This user manual describes all proceedings concerning the operations of this CNC system in detail as much as possible. However, it is impractical to give particular descriptions for all unnecessary or unallowable system operations due to the manual text limit, product specific applications and other causes. Therefore, the proceedings not indicated herein should be considered impractical or unallowable.

Company profile

GSK——GSK CNC Equipment Co,. Ltd is the largest production and marketing enterprise of the CNC system in China currently. It is the Numerical Control industrial base of South China, and the undertaking enterprise of the 863 national main project Industrialization Support Technology for Medium Numerical Control System as well as one of the 20 basic equipment manufacture enterprises in Guangdong province. It has been taking up the research and development, design and the manufacture of machine CNC system (CNC device, drive unit and servo motor) in recent 10 years. Now it has developed into a large high-tech enterprise integrated with research, education, industry and trade by enhancing the popularization and trade of CNC machine tools. There are more than 1400 staffs in this company that involves 4 doctors, more than 50 graduate students and 500 engineers and more than 50 among them are qualified with senior technical post titles. The high performance-cost ratio products of GSK are popularized in China and Southeast Asia. And the market occupation of GSK's product dominates first and the turnout and sale ranks the top in internal industry for successive 7 years from the year 2000 to 2006, which makes it the largest CNC manufacture base throughout China.

The main products provided by our company includes the NC equipments and devices such as GSK series turning machine, milling machine, machining center CNC system, DA98, DA98A, DA98B, DA98D series full digital stepper motor drive device, DY3 series compound stepper driver device, DF3 series response stepper motor driver device, GSK SJT series AC servo motors, CT-L NC slider and so on. The current national standard (and international standard), industry standard, as well as the enterprise standard (or enterprise internal standard) as a supplementary, are completely implemented in our production process. The capability of abundant technology development and complete production and quality system qualified by us will undoubtedly ensure the reliable product to serve our customers. 24~48 hours technological support and service can be easily and promptly provided by our complete service mechanism and tens of service offices distributed in provinces around China and abroad. The pursuit of "excellent product and superexcellent service" has made the GSK what it is now, and we will spare no efforts to continue to consummate this South China NC industry base and enhance our national NC industry by our managerial concept of "Century Enterprise, Golden Brand".

Technological Spot Service

You can ask for spot service if you have the problems that can't be solved by telephone. We will send the authorized engineers to your place to resolve the technological problems for you.

Preface

Your excellency,

It's our pleasure for your patronage and purchase of this GSK GSK218M CNC system made by GSK CNC Equipment Co., Ltd.

This book is "Programming and Operation" manual.

! Accident may occur by improper connection and operation！This system can only be operated by authorized and qualified personnel. Please carefully read this manual before usage！

This manual is reserved by final user.

All specifications and designs herein are subject to change without further notice.
We are full of heartfelt gratitude to you for supporting us in the use of GSK's products.

Warning and Precautions

Warning, note and explanation

This manual contains the precautions to protect user and machine. The precautions are classified as warning and note by safety, and supplementary information is regarded as explanation. Read the warnings, notes and explanations carefully before operation.

Warning

Personnel may be hurted or equipment be damaged if operations and steps are not observed.

Note

Equipment may be damaged if operation instructions or steps are not observed by user.

Explanation

It is used for the supplementary information except for warning and note.

CONTENT

Ⅰ OVERVIEW

1. Overview

This manual is comprised by following parts:

I Overview

It describes the chapter structure, system model available, relative instructions and the note.

Ⅱ Programming

It describes G functions and the programming format, characteristics and restrictions by NC language.

Ⅲ Operation

It describes the manual and auto operation, program input/output and editing methods.

Appendix

It describes parameter list, alarm list and programming data table.

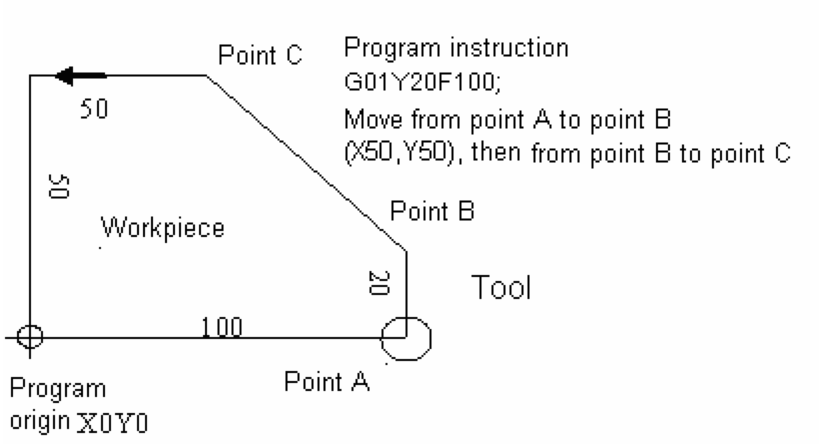
The manual is used for GSK218M CNC system.

Ⅱ PROGRAMMING

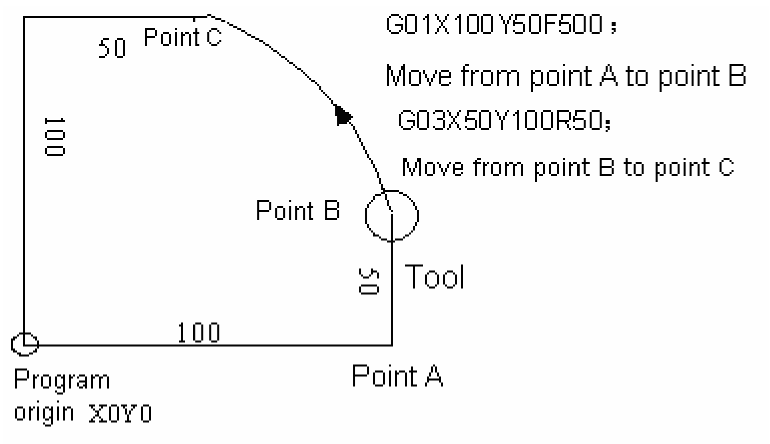
1 General

1.1 Tool movement along workpiece contour —interpolation

1） Tool movement along a straight line

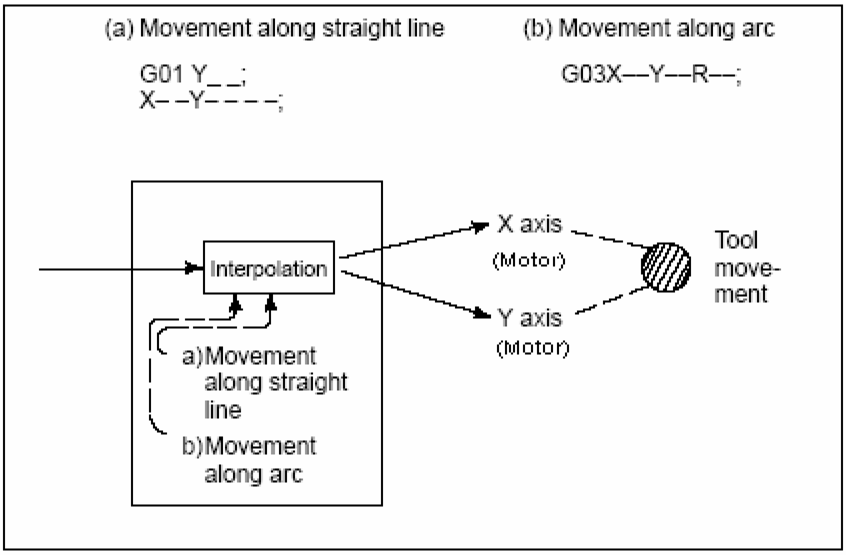


2） Tool movement along an arc



The tool linear and arc motion function is called interpolation.

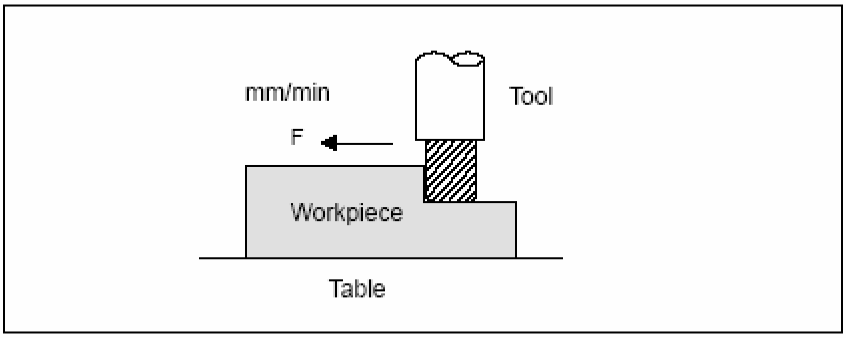
The programming instructions such as G01, G02 are called preparatory function, which is used for interpolation for CNC device.



Note **For some machines, it is the worktable moving other than tool moving in practice. It is assumed that the tool moves relative to the workpiece in this manual. Refer to the machine actual movement direction in practice to protect against personnel hurt and machine damage.**

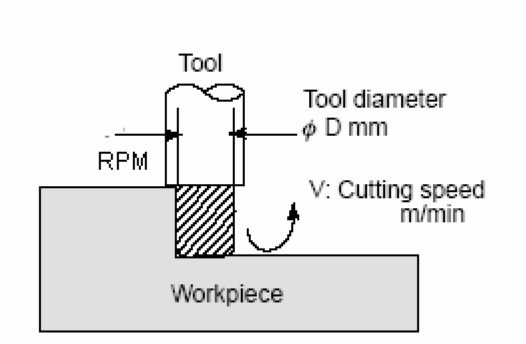
1.2 Feed——Feed function

The feedrate specification is called feed function.



To specify a speed to machine the part by tool is called feed and the machine speed is instructed by a numerical value. For example, the program instruction is F150 if tool feeds by 150mm/min.

1.3 Cutting feedrate, spindle speed function



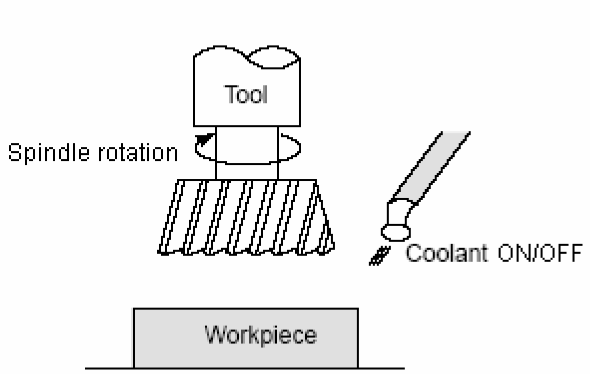
The speed of tool relative to workpiece in cutting is called cutting feedrate. It can be instructed by spindle speed RPM(r/min) by CNC.

Example: If the tool diameter is 10mm, cutting linear speed is 8 m/min, the spindle speed is about 255RPM according to N=1000V/πD, so the instruction is: S255

Instructions related to spindle speed are called spindle speed function.

1.4 Operation instruction——miscellaneous function

When the workpiece is to be machined, to make the spindle run and supply coolant, the machine spindle motor and cooling pump switches must be controlled by actual requirement.

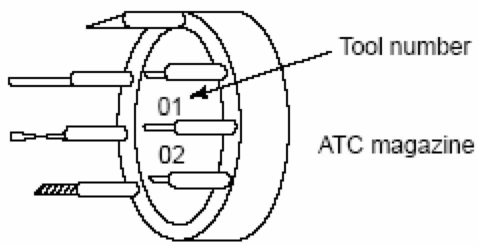


The programs or machine on-off actions controlled by system NC instructions are called miscellaneous functions, which are instructed by M code.

Example If M03 is instructed, the spindle rotates clockwise by the speed specified. (Clockwise direction means the direction viewed from the spindle –Z direction.)

1.5 Tool selection for various machining——Tool function

It is necessary to select a proper tool when drilling, tapping, boring, milling, etc. is performed. When a number is assigned for each tool and the number is specified in the program, the corresponding tool is selected.



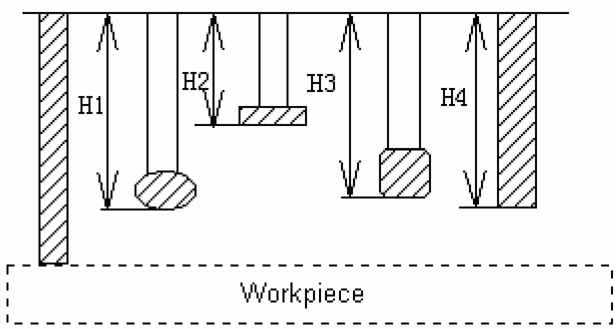
Example When No.01 is assigned to a drilling tool,

When the tool is stored at location 01 in the ATC magazine, the tool can be selected by specifying T01. This is called the tool function.

1.6 Tool figure and tool motion by program

1.6.1 Tool length compensation

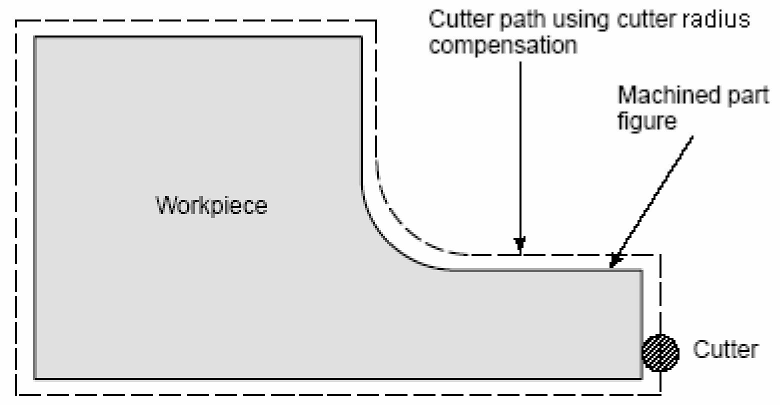
Usually several tools are used for machining one workpiece. If instructions such as G0Z0 are executed in a same coordinate system, because tools have different tool lengths, the distances from tool end to workpiece are different. So it is very troublesome to change the program frequently.



Therefore, the length of each tool used should be measured in advance. By setting the difference between the length of the standard tool and the length of each tool in the CNC (usually the 1st tool), machining can be performed without altering the program even when the tool is changed. After the tool positioning in Z axis (e.g. G0Z0), the distances of the tool end to the workpiece are identical. This function is called tool length compensation.

1.6.2 Tool radius compensation

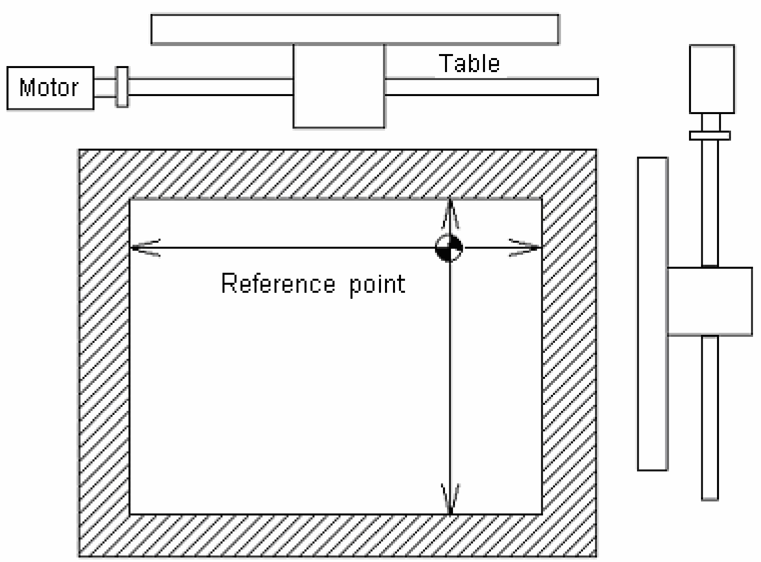
Because a tool has a radius, if the tool goes by the path given by program, the workpiece will be cut off a part for a radius wide. To simplify the programming, the program can be run by CNC around the workpiece with the tool radius deviated, while the transient path of the intersections of the lines or the arcs can be processed automatically by system.



If diameters of tools are stored in the CNC tool compensation list, the tool can be moved by tool radius apart from the machining part figure by calling different radius compensation according to program. This function is called tool radius compensation.

1.7 Tool movement range——stroke

The parameter setting can specify the safe tool running range, if the tool exceeds the range, the system stops all the axes moving with overtravel alarm given. This function is called stroke verification, namely, the software limit.



2 Part Program Composition

2.1 Program composition

A program is composed by many blocks which are formed by words. The blocks are separated by the end code (LF for ISO,CR for EIA). In this manual the end code is represented by "；"character.

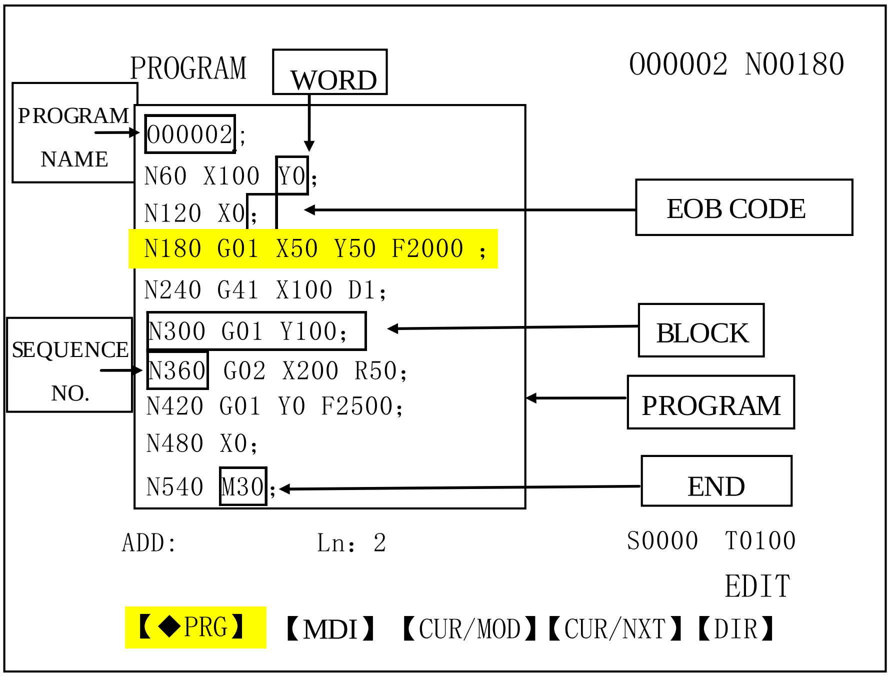


Fig. 2-1 Program structure

The set instructions to control the CNC machine tool to machine the parts are called program. After the program edited is entered into the CNC system, the system controls the tool to move along straight line, arc or make the spindle run or stop by these instructions. And the instructions should be edited by the machine actual movement sequence. The program structure is shown in Fig.2-1.

2.1.1 Program name

In this system the system memory may store many programs. In order to differentiate these programs, address O with five figures behind it is headed in the beginning of the program. And it is shown in Fig. 2-2.

Fig. 2-2 Program name composition

2.1.2 Sequence number and block

A program is consisted by many instructions, and an instruction unit is called block (see Fig. 2-1). The blocks are separated by program end code (see Fig. 2-1). In this manual the block end code is represented by character"；".

Address N with 4 figures sequence number behind it can be used at the beginning of the block (see Fig. 2-1), and the leading zero can be omitted. The sequence of the sequence number (insertion set by bit parameter No. 0 # 5) can be arbitrary, and the intervals between them can be inequal (set by Parameter P210). Sequence number can be either in all blocks, or in some important blocks. But by common machining sequence, the number should be arranged by ascending. That the sequence number is placed in important part of the program is for convenience. (e.g. in tool changing, or worktable indexed to a new plane).

2.1.3 Instruction word

Word is a factor to block composition. It is formed by an address and figures behind it (sometimes +, - added before figures)

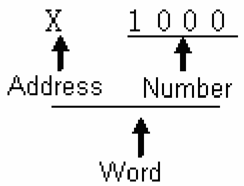


Fig.2-3 Word composition

The address is a character from English alphabetic table which defines the meaning of the figure behind it. In this system, the usable addresses and their meaning as well as value range are shown as Table2-1:

Sometimes an address has a different meaning for different preparatory function.

If 2 or more identical addresses appear in an instruction, the alarm for it will be set by parameter **N0. 32#6.**

Table 2-1

Address	Range	Meaning
O	0～99999	Program name
N	0～99999	Sequence number
G	00～99	Preparatory function
X	-99999.999～99999.999（mm）	X coordinate address
	0.001～9999.999（s）	Dwell time
Y	-99999.999～99999.999（mm）	Y coordinate address
Z	-99999.999～99999.999（mm）	Z coordinate address
R	-99999.999～99999.999（mm）	Arc radius/angle displacement
	-99999.999～99999.999（mm）	R level in canned cycle

Address	Range	Meaning
I	-99999.999～99999.999（mm）	Arc center vector in X axis relative to start point
J	-99999.999～99999.999（mm）	Arc center vector in Y axis relative to start point
K	-99999.999～99999.999（mm）	Arc center vector in Z axis relative to start point
F	0～99999（mm/min）	Feed in a minute
	0.001～500(mm/r)	Feed in a revolution
S	0～99999（r/min）	Spindle speed
	00～04	Multi-gear spindle output
T	0～128	Tool function
M	00～99	Miscellaneous function output, program executing process, subprogram calling
P	1～9999999（ms）	Dwell time
	1～99999	Subprogram number calling
Q	-99999.999～99999.999（mm）	Cutting depth or hole bottom offset in canned cycle
H	01～99	Operator for G65
	00～99	Length offset number
D	00～99	Radius offset number

Special attention should be paid that the limits in table 2-1 are all for CNC device, but not for machine tool. Therefore, programming should be done on a basis of good understanding of the programming limitation of machine builder manual besides this manual.

2.2 General structure of a program

The program is classified for main program and subprogram. Generally, the CNC system is acutated by the main program. If the main program contains the subprogram call, the CNC system acts by the subprogram. If the subprogram contains the instruction of returning to main program, the CNC system returns to the main program to go on execution. The program execution sequence is shown as Fig.2-4.

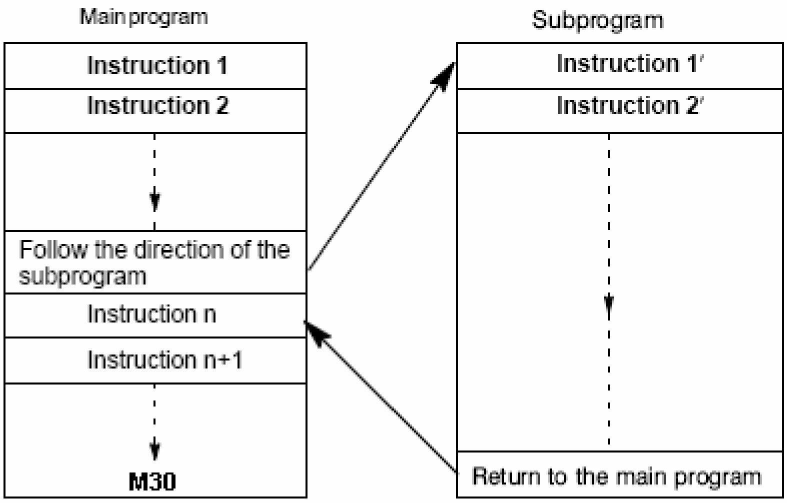


Fig.2-4 Program execution sequence

The structure of the subprogram is same as that of the main program.

If there are fixed sequence blocks occurring repeatedly in a program, it can be taken as a subprogram which can be stored in the memory in advance with no need to be edited repeatedly. So it can simplify the program. The subprogram can be called in Auto mode, usually by M98 in the main program. And the subprogram called can also call other subprograms. The subprogram called from the main program is called the 1st level subprogram. 4 levels subprogram at most can be called in a program (Fig.2-5). The last block in the subprogram must be the returning instruction M99. After M99 execution, the control returns to next block following the block that calls the subprogram in the main program to go on execution. If the main program end is M99, the program execution can be repeated.

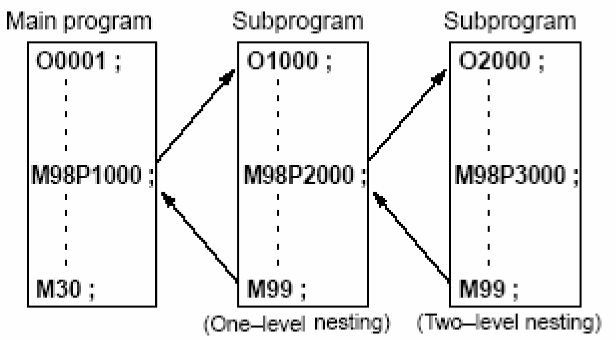


Fig. 2-5 Two-level subprogram nesting

A single subprogram call instruction can be continuously and repeatedly used to call a subprogram up to 999 times.

2.2.1 Subprogram edit

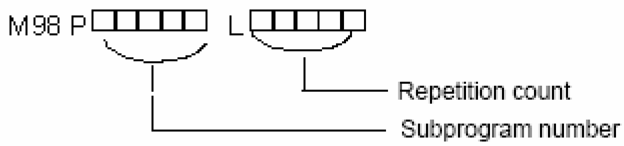
Write out a subprogram by following format:

```
O □□□□□ ;      Subprogram number
.............. ;
..................;
..................
         ...                    Subprogram
         ...
         ...
...................;
..........     .;
M99 ;          Subprogram end
```

Write out the subprogram number behind the address O at the subprogram beginning, and the M99 instruction at the subprogram end (M99 format as above).

2.2.2 Subprogram call

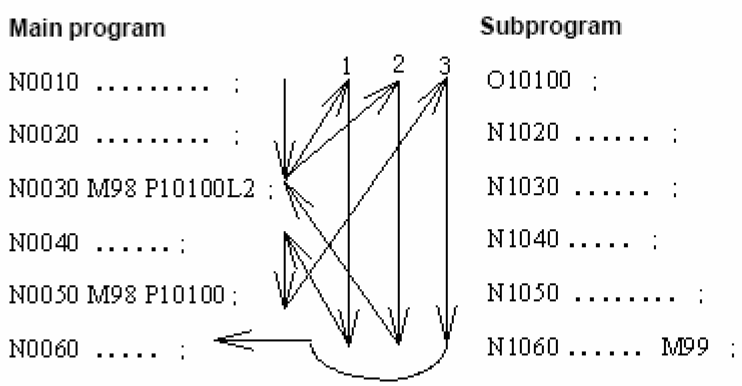
The subprogram is called out for execution by the main program or the subprogram. The instruction format is as following:



- If the repeat time is omitted, the default is 1.

Example M98 P1002L5 ；(It means No.1002 subprogram is continuously called for 5 times.)

- M98 P__ cann’t be in a block with movement instruction.
- Execution sequence of subprogram call from main program



Subprogram call from subprogram are identical with that from main program.

Note Alarm (PS 078) occurs if subprogram number specified by address P is not found.

2.2.3 Program end

The program begins with program name, ends with M02, M30 or M99 (see Fig.2-2). For the end code M02,,M30 or M99 detected in program execution: if M02, M30 specifies the end, the program finishes and reset; if M99 specifies the end, the control returns to the program beginning to restart the program; if M99 is at the end of the subprogram, the control returns to the program that calls the subprogram. M30 can be set by bit parameter N0.33#4 for returning to the program beginning, and M02 can be set by bit parameter N0.33#4 for returning to the program beginning.

3 Programming Fundamentals

3.1 Controlled axis

Table 3-1

Item	218M
Basic controlled axes	3 axes（X, Y, Z）
Extended controlled axes (total)	5 axes

3.2 Axis name

The 3 primary axis names are always X, Y, or Z. And the controlled axes are set by number parameter No.5. The additional axis names are set by number parameter No.6 accordingly, such as A, B, C.

3.3 Coordinate system

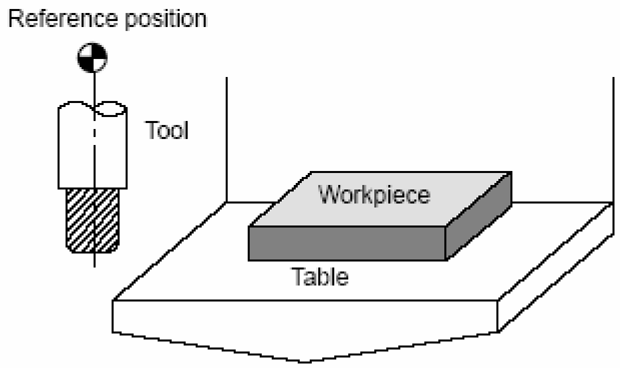
3.3.1 Machine coordinate system

A special point on machine used as machine benchmark is called machine zero, which is set by the machine builder. The coordinate system set by machine zero taken as origin is called machine coordinate system. It is set up by manual machine zero return after power is on. Once set, it remains unchanged till the power off, system reset or emergency stop.

This system uses right-hand Cartesian coordinate system. The motion along spindle is Z axis motion. Viewed from spindle, the motion of headstock approaching the workpiece is negative Z axis motion, and departing for positive. The other directions are determined by right-hand Cartesian coordinate system.

3.3.2 Reference point

There is a special point on CNC machine tool for tool change and coordinate system setup, which is called reference point. It is a fixed point in machine coordinate system set by machine builder. By reference point return, the tool can easily move to this position. Generally this point in CNC milling system coincides with the machine zero, while the reference point of Machine Center is usually the tool change point.

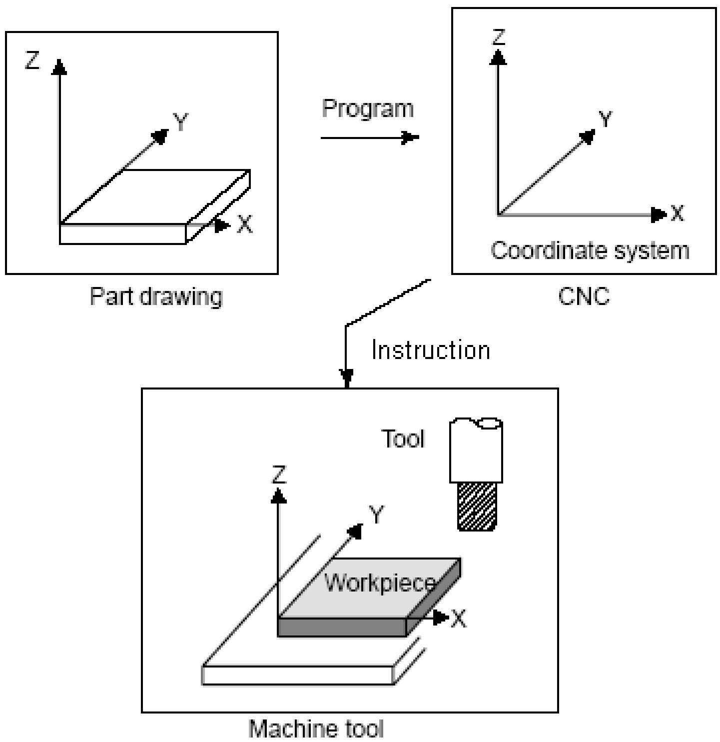


There are two methods to traverse the tool to reference point:

1. Manual reference point return (see"Manual reference point return"in Operation Manual)
2. Auto reference point return

3.3.3 Workpiece coordinate system

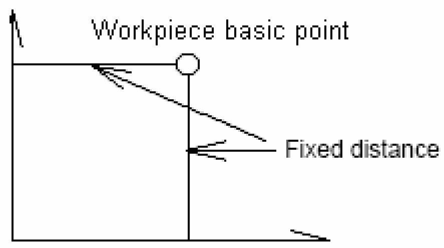
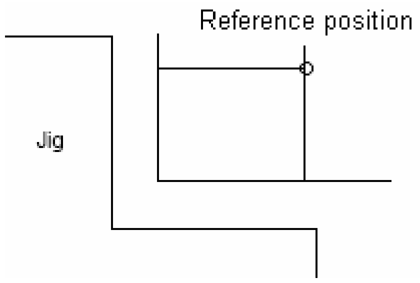
The coordinate system used for workpiece machining is called workpiece coordinate system (or part coordinate system), which is preset by CNC system.



In order to make the tool to cut the workpiece to the figure on drawing by instruction program according to drawing in the workpiece coordinate system specified by CNC, the relation of the machine coordinate system and the workpiece coordinate system must be determined.

The method to determine the relation of these two coordinate systems is called alignment. It can be done by different methods such as part figure, workpiece quantity.

.

Ⅰ) By part base point	Ⅱ) When part is fixed on jig
	
To align the tool center to the workpiece base point, specify the workpiece coordinate system by CNC instructions at this position, and the workpiece coordinate system coincides with the programming coordinate system.	Because the tool center can't be located at the workpiece base point, locate the tool to a position (or reference point) that has a distance to the base point, set the workpiece coordinate system by this distance(e.g. G92)

Workpiece coordinate system should be set for each processing program (to select a workpiece coordinate system). The workpiece coordinate system set can be changed by moving its origin.

There are two methods to set the workpiece coordinate system:

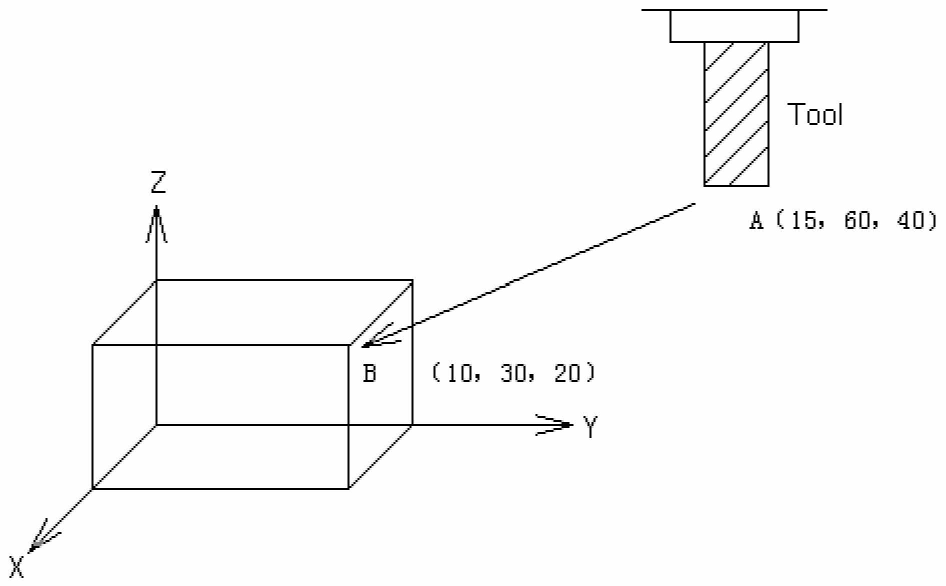
1. By G92, see 4.2.11 for details.
2. By G code from 54 to 59, see 4.2.8 for details.

3.3.4 Absolute programming and relative programming

There are absolute and relative definitions to define the axis moving. The absolute definition is the method of programming by the axis moving final point, which is called absolute programming. The relative definition is the method of programming by the axis moving, which is called incremental programming.

1) Absolute coordinate

It is the target position coordinate in the specified workpiece coordinate system, namely the position the tool to move to.

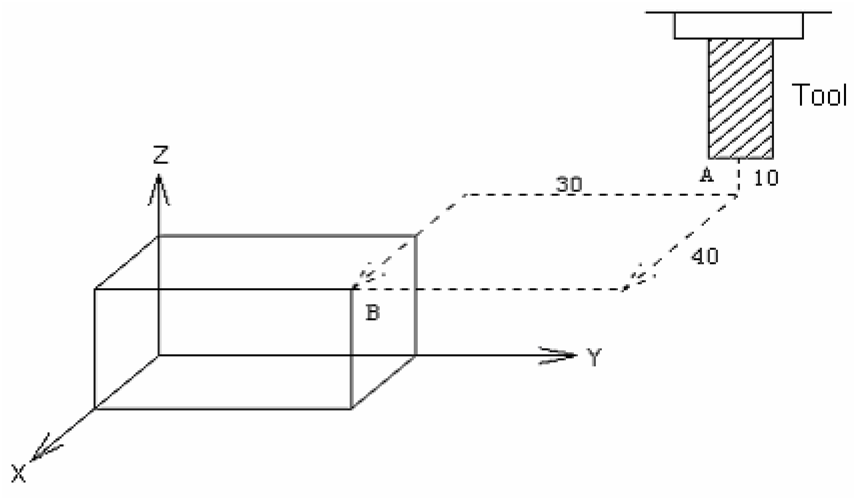


Move the tool from point A to point B, using the B coordinate in G54 workpiece coordinate system, the instruction is as following:

G90 G54X10 Y30 Z20 ;

2) Relative coordinate

It is the target position coordinate relative to the current position by taking the current position as the origin.



For traversing the tool from point A to point B, the instruction is as following:

G0 G91 X-40 Y-30 Z-10;

3.4 Mode and non-mode

The mode means that the address value set by a block is effective till it is reset by another block. Another significance of it is that if a functional word is set, it doesn't need to be input again if it is used in the following blocks.

- e.g. for following program:

  G0 X100 Y100； (rapid positioning to the location X100 Y100)

  X20 Y30； (rapid positioning to the location X120 Y30, G0 specified by mode can be omitted)

  G1 X50 Y50 F300； (interpolate to location X50 Y50 by straight line with the feedrate 300mm/min G0→G1)

  X100； (interpolate to location X100 Y50 by straight line with the feedrate 300mm/min , G1, Z50,F300 are all specified by mode and can be omitted)

  G0 X0 Y0； (rapid positioning to the location X0 Y0)

The initial state is the default state after the system power-on. See table 4-1.

- For following program:

  O00001

  X100 Y100； (rapid positioning to the location X100 Y100, G0 is the initial state)

  G1 X0 Y0 F100； (interpolate to location X0 Y0 by straight line with the feedrate 100mm/min, G98 is the initial power-on state)

Non-modal means that the relevant address value is effective only in the block contains this address, if it is used in following blocks, it must be respecified. e.g. G functional instructions of 00 group in Table 4-1.

Refer to Table 3-4 for mode and non-modal description for functional word.

Table 3-4 Mode and non-modal for functional instruction

Mode	Modal G function	A group of G functions that can be cancelled by each other, once executed, they are effective till they are cancelled by other G functions in the same group.
	Modal M function	A group of M functions that can be cancelled by each other, once executed, they are effective till they are cancelled by other G functions in the same group.
Non-modal	Non-modal G function	They are only effective in the block they are specified and cancelled at the block end.
	Non-modal M function	They are only effective in the block they are specified.

3.5 Decimal point programming

Numerical values can be entered with a decimal point. A decimal point can be used when entering a distance, time, or speed. Decimal points can be specified with the following addresses:

X, Y, Z, A, B, C, I, J, K, R, P, Q, and F.

Explanation:

1、 The decimal point programming are set by bit parameter NO.33#1. If bit parameter NO.33#1=1, the programming value unit is mm, inch, or deg; if bit parameter NO.33#1=0, the programming value unit is the min. moving unit which is set by bit parameter NO.5#1.

2、 The decimal part that is less than the min. input incremental unit should be omitted.

Example:

X9.87654； When the min. input incremental unit is 0.001mm, it should be X 9.876.

When the min. input incremental unit is 0.0001mm, it should be X 9.8765.

4 Preparatory Function: G code

4.1 Classification of G code

Preparatory function is represented by G code with the number behind it, which defines the meaning of the block that contains it. G codes are devided by the following two types:

Classification	Meaning
Non-modal G code	Effective in the block in which it is specified
modal G code	Effective till another G code of the same group is specified

Example G01 and G00 are modal G code in the same group.

G01 X __ ；

Z ______ ； G01 effective

X ______ ； G01 effective

G00 Z____； G00 effective

Note Refer to system parameter list (modal list) for details.

Table 4-1 G codes and their functions

G code	Group	Instruction format	Function
*G00	01	G00 X_Y_Z_	Positioning (traverse)
G01		G01 X_Y_Z_F_	Linear interpolation(cutting feed)
G02		G02 X_Y_ R_ F_；	Circular interpolation CW
G03		G03 X_Y_ I_J_ F_；	Circular interpolation CCW
G04	00	G04 P_ or G04 X_	Dwell, exact stop
G10	00	G10L_；N_P_R_	Programmable data input
*G11	00	G11	Programmable data input cancel
*G12	16	G12 X_Y_Z_ I_J_K_	Storage stroke detection on
G13		G13 X_Y_Z_ I_J_K_	Storage stroke detection off
*G15	11	G15	Polar coordinate instruction cancel

G16		G16			Polar coordinate instruction
*G17 G18 G19	02	Write in with other program in block, used for circular interpolation and tool radius compensation			XY plane selection ZX plane selection YZ plane selection
G20	06	Specified by a single block at the program beginning before the coordinate system set			Inch input
*G21					Metric input
G27	00	G27	X_Y_Z_		Reference point return detection
G28		G28			Reference point return
G29		G29			Return from reference point
G30		G30Pn			2^{nd} ,3^{rd}, 4^{th} reference point return
G31		G31			Skip function
G39		G39	I_J_; I_J_; J_K_ or G39		Corner offset circular interpolation
*G40	07	G17	G40 G41 G42	X_Y_	Tool radius compensation cancel
G41		G18		X_Z_	Left-hand tool radius compensation
G42		G19		Y_Z_	Right-hand tool radius compensation
G43	08	G43		Z_	Positive tool length compensation
G44		G44			Negative tool length compensation
*G49		G49			Tool length compensation cancel
*G50	12	G51			Scaling cancel
G51		G51 X_ Y_ Z_ P_			Scaling
G53	00	Write into the program			Machine coordinate system selection
*G54	05	Write into the block with other program, usually placed at the program beginning			Workpiece coordinate system 1
G55					Workpiece coordinate system 2
G56					Workpiece coordinate system 3
G57					Workpiece coordinate system 4
G58					Workpiece coordinate system 5
G59					Workpiece coordinate system 6
G60	00	G60 X_ Y_ Z_ F_			Unidirectional position
G61	14	G61			Exact stop mode
G62		G62			Automatic corner override
G63		G63			Tapping mode
*G64		G64			Cutting mode
G65	00	G65 H_P# i Q# j R# k			Macro program instruction
G68	13	G68 X_ Y_ R_			Coordinate system rotation
*G69		G69			Coordinate system rotation cancel
G73	09	G73 X_Y_Z_R_Q_F_;			Peck drilling cycle
G74		G74 X_Y_Z_R_P_F_;			Lef-hand tapping cycle
G76		G76 X_Y_Z_R_P_F_K_;			Fine boring cycle
*G80		Write into the block with other program			Canned cycle cancel
G81		G81 X_Y_Z_R_F_;			Drilling cycle(spot drilling cycle)

G82		G82 X_Y_Z_R_P_F_；	Drilling cycle (counter boring cycle)
G83		G83 X_Y_Z_R_Q_F；	Peck drilling cycle
G84		G84 X_Y_Z_R_P_F_；	Tapping cycle
G85		G85 X_Y_Z_R_F_；	Boring cycle
G86		G86 X_Y_Z_R_F_；	Drilling cycle
G87		G87 X_Y_Z_R_Q_P_F_；	Back boring cycle
G88		G88 X_Y_Z_R_P_F_；	Boring cycle
G89		G89 X_Y_Z_R_P_F_；	Boring cycle
*G90	03	Write into the block with other program	Absolute programming
G91			Incremental programming
G92	00	G92 X_Y_Z_	Coordinate system set
*G94	04	G94	Feed per minute
G95		G95	Feed per revolution
G96	15	G96S_	Constant surface speed control （cutting speed）
*G97		G97S_	Constant surface speed control cancel（cutting speed）
*G98	10	Write into the block with other program	Return to initial point in canned cycle
G99			Return to point R level (in canned cycle)

Note 1 For the G code with * sign, when the power is switched on, the system is in the state of this G code.

Note 2 G codes except G10, G11 in 00 group are all non-modal G code.

Note 3 Alarm occurs if G code not listed in this table is used or G code without the selection function is specified.

Note 4 G codes from different groups can be specified in a block, but 2 or more G codes from the same group can’t be specified in a block, otherwise alarm or tool abnormity occurs.

Note 5 In canned cycle, if G code from 01 group is specified, the canned cycle will be cancelled automatically and system turns into G80 state. But G codes in 01 group are not affected by G codes in canned cycle.

Note 6 G codes are represented by group numbers repectively according to their types. All G codes can be cleared by bit parameter No.35#0~7 and No.36#0~7 setting at system reset and emergency stop.

4.2 Simple G code

4.2.1 Rapid positioning G00

Format: G00 X_Y_Z_

Function: G00 instruction moves the tool to the position in the workpiece system specified with an absolute or an incremental instruction at a traverse speed by linear interpolation. It is set by bit parameter NO.12#1 and uses the following two path.(Fig. 4-2-1-1).

1. Linear interpolation positioning: The tool path is the same as in linear interpolation（G01）. The tool is positioned within the shortest possible time at a speed not more than the traverse speed of each axis.

2. Nonlinear interpolation positioning: The tool is positioned with the traverse speed of each axis respectively. The tool path is usually not straight.

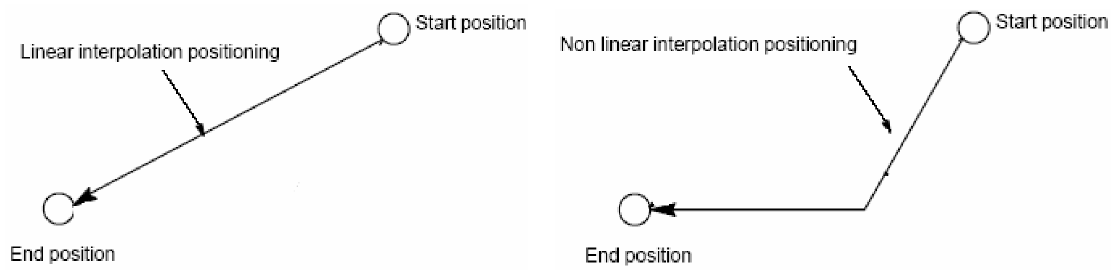


Fig. 4-2-1-1

Explanation:

1 After G00 is executed, the system change the tool current move mode for G00 mode. The G00 (parameter value is 0) or G01 (parameter value is 1)default mode can be set by bit parameter No.031#0 while the power is switched on.

2 The tool doesn’t move if positioning parameter is not specified, and the system only change the current tool move mode for G00.

3 G00 are identical with G0.

4 G0 speed for X,Y, Z axis is set by number parameter P88~P92.

Restrictions

1 The traverse speed is set by parameter, if F is specified in G0 instruction, it is used for the following cutting feedrate. For example:

G0 X0 Y10 F800； rapid traversing by system parameter set

G1 X20 Y50； by F800 feedrate

The rapid feedrate is adjusted by the key on operator panel with following override : F0，25， 50， 100%, see Fig. 2-4-1-2. The speed for F0 is set by number parameter P93, and they are used by all axes.

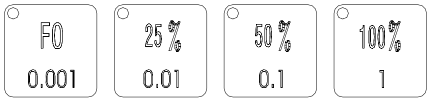


Fig. 2-4-1-2 Rapid feedrate override key

2 G00 is unallowed to be programmed in a block with the same group modal G codes such as G01, G02, G03, otherwise alarm is issued by system.

4.2.2 Linear interpolation G01

Format: G01 X_ Y_ Z_ F_

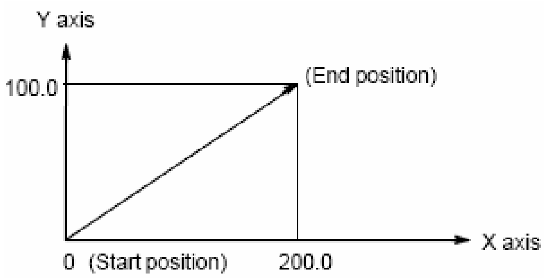
Function: The tool moves along a line to the specified position at the feedrate (mm/min)specified by parameter F.

Explanation:

1 X_ Y_ Z_ are the final point coordinate which concerns the coordinate system, refer to 3.3.1～3.3.3 sections.

2 The feedrate specified by F is effective till the new F code is specified. The feedrate by F code is got by an interpolation along a line, if F code is not specified in program, the feedrate uses the default value when the power is on.(see number parameter P87 for the setting)

Program example (see Fig. 4-2-2-1)



G01 X200 Y100 F200 ；

Note：

Each axis feedrate is as following：

G01 Xα Yβ ZγFf ；

In this block：

Feedrate in X axis： $F_X = \frac{\alpha}{L} \times f$

Feedrate in Y axis： $F_Y = \frac{\beta}{L} \times f$

Feedrate in Z axis： $F_Z = \frac{\gamma}{L} \times f$

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

Fig. 4-2-2-1

Note 1 The instruction parameters except F are all positioning parameter. And the upper limit of the feedrate F can be set by number parameter P94. If the actual federate (using override) exceeds the upper limit, it is restricted to the upper limit and its unit is mm/min. The lower limit of the feedrate F can be set by number parameter P95. If the actual federate (using override) exceeds the lower limit, it is restricted to the lower limit and its unit is mm/min.

Note 2 If the positioning parameter behind G01 is not specified, the tool doesn't move, and the system only changes the tool current mode for G 01 mode. The system default mode at power-on can be set for G00 (value is 0) or G01 (value is 1) by altering the system bit parameter NO.31#0.

4.2.3 Circular (helical) interpolation G02/G03

A Circular interpolation G02/G03

Prescriptions for G02/G03:

The plane circular interpolation means that the arc path is to be finished by the specified rotation and radius or circle center from the start point to the end point in the specified plane.

Because the arc path can't be defined only by the start point and the end point, other conditions are needed:

- Arc rotation direction（G02，G03）
- Circular interpolation plane（G17, G18, G19）
- Circle center coordinate or radius, which gives two programming Format: Circle center coordinate I, J ,K or radius R programming

Only the three points above are all confirmed, could the interpolation operation be done in coordinate system.

The circular interpolation can be done by the following instructions to make the tool to go along an arc, it is shown as follows:

Arc in XY plane

```
     G02         R_
G17        X_Y_        F_;
     G03         I_J_
```

Arc in ZX plane

```
     G02         R_
G18        X_Z_        F_;
     G03         I_K_
```

Arc in YZ plane

```
     G02         R_
G19        Y_Z_        F_;
     G03         J_K_
```

Item	Content	Instruction	Description
1	To specify plane	G17	Arc specification on XY plane
		G18	Arc specification on ZX plane
		G19	Arc specification on YZ plane
2	To specify rotation direction	G02	CW
		G03	CCW
3	G90 Final position G91	Two axes of X,Y, Z axis	End point coordinate in workpiece coordinate system
		Two axes of X,Y, Z axis	Coordinate of end point relative to start point
4	Distance from start point to circle center	Two axes of I,J, K axis	Coordinate of circle center relative to start point
	Arc radius	R	Arc radius
5	Feedrate	F	Arc tangential speed

CW and CCW mean the directions viewed from the positive Z(or Y, Z) axis to the negative in the right-hand Cartesian coordinate system regarding to XY (or ZX, YZ)plane , as shown in Fig. 4-2-3-1.

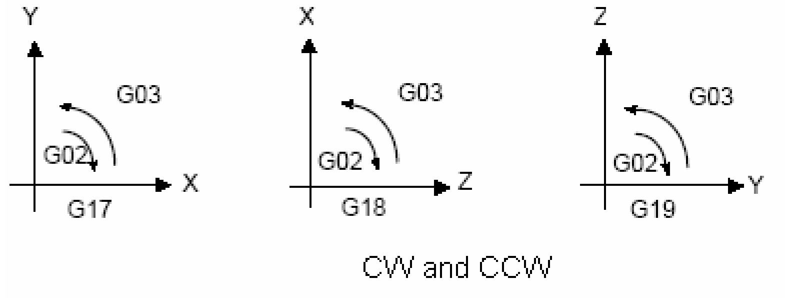


Fig. 4-2-3-1

The default plane mode at power-on can be set by bit parameters **NO.31#1, #2, #3.**

The arc end point can be specified by parameter words X, Y, Z. It is an absolute value in G90, an incremental value that is a coordinate of the end point relative to the start point in G91. The circle center is specified by parameter words I, J, K, corresponding to X, Y, Z respectively. Either in absolute mode G90, or in incremental mode G91, parameter values of I, J, K are coordinates of circle center relative to the arc start point (for simplicity, the circle center coordinate when taking the start point as origin). They are incremental values with signs. See Fig. 4-2-3-2.

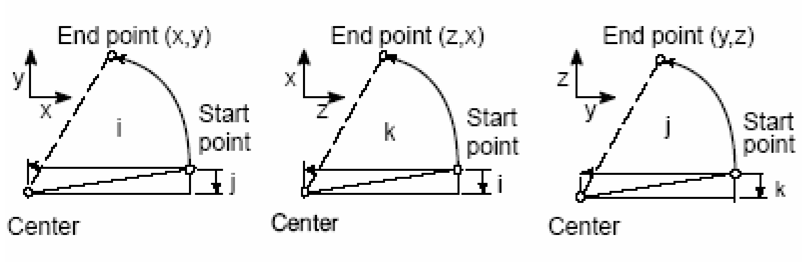


Fig. 4-2-3-2

I, J, K are assigned with sign according to the circle center relative to the start point. The circle center can also be specified by radius R besides I, J, K.

G02 X_ Y_ R_ ；

G03 X_ Y_ R_ ；

1 Two arcs can be drawn out as following, one arc is more than 180°, the other one is less than 180°. The radius of the arc more than 180° should be specified by a negative value.

(e.g. Fig. 2-4-4-3) as arc ① is less than 180°

G91 G02 X60 Y20 R50 F300 ；

as arc ② is more than 180°

G91 G02 X60 Y20 R-50 F300 ；

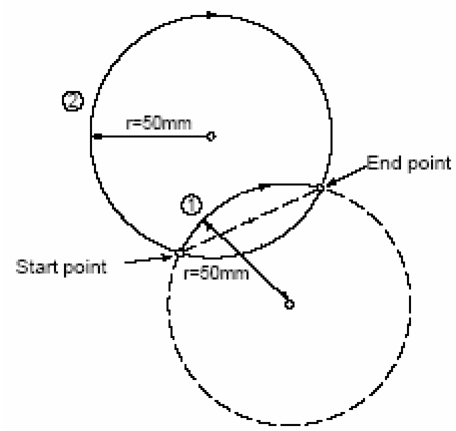


Fig. 2-4-4-3

2 The arc equal to 180° can be programmed either by I, J, K, or by R.

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

Equal to G90 G0 X0 Y0；G2 X20 R10 F100

or G90 G0 X0 Y0；G2 X20 R-10 F100

Note For the arc 180°, the positive or negative value of R doesn't affect the arc path.

3 The arc equal to 360° can only be programmed by I, J, K.

(Program example)

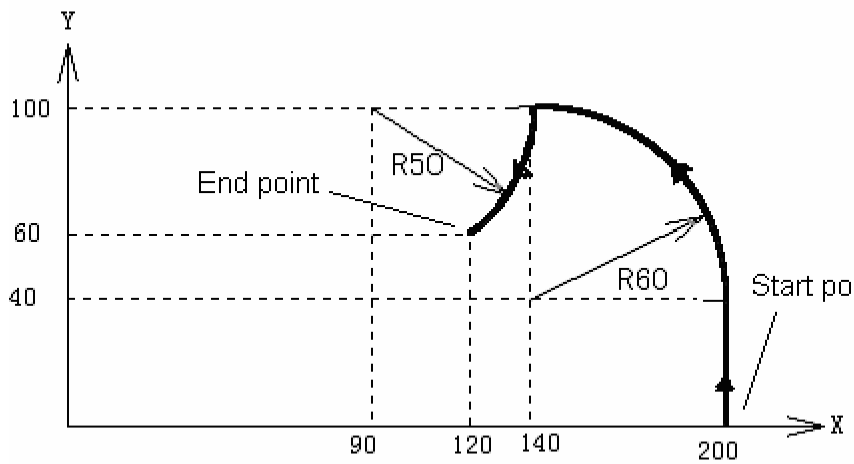


Fig. 2-4-4-4

The tool path programming for Fig. 2-4-4-4 is as following:

1. Absolute programming

G90 G0 X200 Y40 Z0；

G3 X140 Y100 R60 F300；

G2 X120 Y60 R50；

Or

G0 X200 Y40 Z0；

G90 G3 X140 Y100 I-60 F300；

G2 X120 Y60 I-50；

2． Incremental programming

G0 G90 X200 Y40 Z0；

G91 G3 X-60 Y60 R60 F3000；

G2 X-20 Y-40 R50；

Or

G0 G90 X200 Y40 Z0；

G91 G3 X-60 Y60 I-60 F300；

G2 X-20 Y-40 I-50；

Restriction

1. If address I, J, K and R are specified together in program, the arc specified by R is in priority and others are ignored.
2. If both arc radius parameter and the parameter from the start point to the circle center are not specified, error message will be issued by system.
3. If the circle is to be interpolated, only the parameters I, J, K from start point to circle center but the parameter R can be specified.
4. Attention should be paid to the coordinate plane selection when the circular interpolation is being done.
5. If X, Y, Z are all omitted, i.e. the start point and the final point coincides, as well as R is specified (e.g. G02R50), the tool doesn't move.

B Helical interpolation

Format: G02/G03

Arc of XY plane

$$G17 \begin{Bmatrix} G02 \\ G03 \end{Bmatrix} X_p\_ Y_p\_ \; Z_p\_ \begin{Bmatrix} I\_ J\_ \\ R\_ \end{Bmatrix} \quad F\_$$

Arc of ZX plane

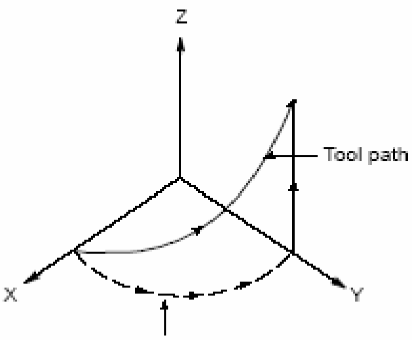
$$G18 \begin{Bmatrix} G02 \\ G03 \end{Bmatrix} X_p\_ Y_p\_ \; Z_p\_ \begin{Bmatrix} I\_ K\_ \\ R\_ \end{Bmatrix} \quad F\_$$

Arc of YZ plane

$$G19 \begin{Bmatrix} G02 \\ G03 \end{Bmatrix} X_p\_ Y_p\_ \; Z_p\_ \begin{Bmatrix} J\_ K\_ \\ R\_ \end{Bmatrix} \quad F\_$$

Function: It is used to move the tool to a position specified from current position by a feedrate specified by parameter F in a helical path.

Explanation:



The feedrate along the circumference of two circular interpolated axes is the specified feedrate.

The first two bits of the instruction parameter are positioning parameter. The parameter words are the two axes name (X, Y or Z) in current plane. These two positioning parameters specify the position the tool is to go to. The third parameter word of the instruction parameter is a linear axis except the circular interpolation axis. Its value is the helical height. The significance and restriction for other instruction parameters are identical with circular interpolation.

If the circle can't be machined by the system specified instruction parameter, the system will give error message. And the system changes the current tool moving mode for G02/G03 mode.

Feedrate along the two circular interpolation axes are specified

A moving axis that is not circular interpolation axis is added as for the instruction method, and F instruction specifies the feedrate along an arc. So the feedrate of this linear axis is as following:

$$F * \frac{\text{Length of circular arc}}{\text{Length of linear axis}}$$

The feedrate should be ensured that the linear axis feedrate are not beyond any limit.

Restriction Attention should be paid to the coordinate plane selection set when the helical interpolation is being done.

4.2.4 Absolute/ incremental programming G90/G91

Format: G90/G91

Function: There are 2 instructions for axis moving, the absolute instruction and the incremental instruction.

The absolute instruction is a method of programming by the axis moving end point coordinate, which is concerned with coordinate system. Refer to section 3.3.1～3.3.4.

The incremental instruction is a method of programming by the axis relative moving. The incremental value is irrelevant with the coordinate system concerned, it only uses moving direction and distance of the end point relative to the start point.

The absolute instruction and the incremental instruction are specified by G90 and G91 respectively.

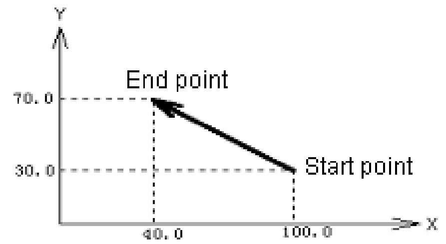


Fig. 2-4-3-1

For the moving from start point to end point in Fig. 2-4-3-1, the programming by absolute instruction G90 and incremental instruction G91 are as follows:

G90 G0 X40 Y70；

or G91 G0 X－60 Y40 ；

The action can be performed by both programming methods that can be expediently used by operator.

Explanation:

- No instruction parameter. It can be written into the block with other instructions.
- G90 and G91 are the same group mode, i.e. if G90 is specified while G91 not, the mode is G90(default). If G91 specified while G90 not, the mode is G91.

System parameter

G90 or G91 mode specified for the default positioning parameter at power-on can be set by bit parameter **NO.31#4(** parameter is 1**).**

4.2.5 Dwell(G04)

Format: G04 X_ or P_

Function: The dwell is executed by G04, and the execution of next block is delayed by the time specified. In addition, a dwell can be specified to make an exact stop check in cutting mode G64.

G04	**X** **P**	0～9999.999 0～99999.9999	X for second P for millisecond

Explanation:

1 G04 is non-modal instruction, which is only effective in current line.

2 Alarm occurs if parameter X, P both appear.

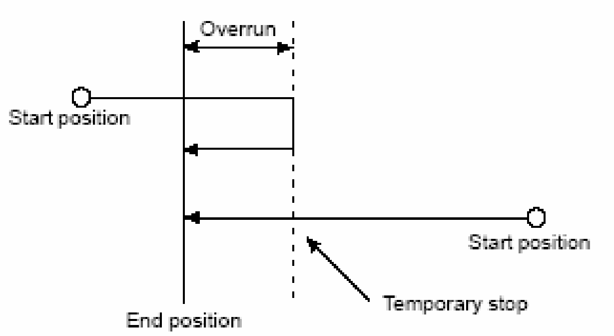
3 Only X or P can follow G04 instruction, alarm occurs if other code follows it.

4 Alarm occurs if X, P value is set for negative.

5 Exact stop is executed if neither X nor P is specified.

4.2.6 Unidirectional positioning (G60)

Format: G60 X_ Y_ Z_ F_



Function: For accurate positioning to eliminate machine backlash, G60 can be used for accurate positioning in a direction.

Explanation:

G60 is non-modal code, which is only effective in a specified block.

For parameter X, Y, Z, they represent the end point coordinate in absolute programming; and moving distance of tool in incremental programming.

When using unidirectional positioning in tool offset, the path of unidirectional positioning is the tool compensation path.

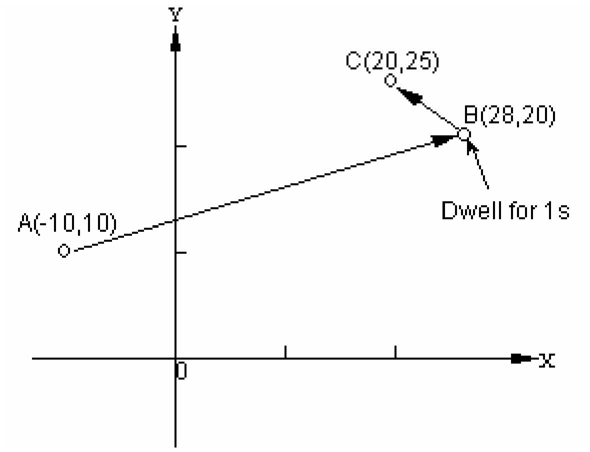
The overrun marked in above figure can be set by system parameter P335，P336，P337，P338，P339, and the dwell time can be set by parameter P334. The positioning direction can be defined by the set positive or negative overrun, refer to system parameter for details.

Example 1:

G90 G00 X-10 Y10;

G60 X20 Y25; (1)

If the system parameter P334 = 1, P335 = -8, P336 = 5; as for statement (1), the tool path is AB→dwell for 1s→BC



System parameter:

P335	Overrun and unidirectional positioning direction in X axis（unit：mm）
P336	Overrun and unidirectional positioning direction in Y axis（unit：mm）
P337	Overrun and unidirectional positioning direction in Z axis（unit：mm）
P338	Overrun and unidirectional positioning direction in 4th axis（unit：mm）
P339	Overrun and unidirectional positioning direction in 5th axis（unit：mm）
P334	Dwell time of unidirectional positioning（unit：mm）

4.2.7 System parameter online modification (G10)

Function: It is used to set or modify the values of pitch error compensation, radius, length offset, external zero offset, workpiece zero offset, additional workpiece zero offset, number parameter, bit parameter and so on in program.

Format:

G10 L50 N_P _R_； Set or modify bit parameter
G10 L51 N_ R_； Set or modify number parameter
⋮
G11； Parameter input mode cancel

Parameter definition:

N： Parameter number. Sequence number to be modified.

P： Parameter bit number. Bit number to be modified.

R： Value. Parameter value after it modified.

The values can also be modified by following instructions, refer to relative sections for details:

G10 L2 P_X_Y_Z_A_B_; Set or modify external zero offset or workpiece zero offset
G10 L10 P_R_; Set or modify length offset
G10 L11 P_R_; Set or modify length wear
G10 L12 P_R_; Set or modify radius offset
G10 L13 P_R_; Set or modify radius wear
G10 L20 P_ X_Y_Z_A_B_; Set or modify additional workpiece zero offset

Note:

In parameter input mode, except annotation statement, other NC statement can't be specified.

G10 must be specified in a single block or the alarm occurs. It should be noted that the parameter input mode must be cancelled by G11 for after G10 for program normal use.

The parameter value modified by G10 must be within the system parameter range. If not, alarm occurs.

The canned cycle mode must be cancelled prior to G10 execution, or alarm occurs.

4.2.8 Workpiece coordinate system G54～G59

Format: G54～G59

Function: It specifies the current workpiece coordinate system. It is used to select workpiece coordinate system by specifying workpiece coordinate system G code in program.

Explanation:

1. No instruction parameter.
2. 6 workpiece coordinate system can be set in the system, any of which can be selected by G54~G59 instruction.
3. G54 (workpiece coordinate system 1) is selected automatically by system after machine zero return at power-on. The absolute position on displayer is the coordinate set in G54 coordinate system.

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

4. When different workpiece coordinate system is called by block, the axis for move by instruction will be located in the new workpiece coordinate system; for the coordinate of the axis not move, it turns to the corresponding coordinate in the new workpiece coordinate

system and the actual machine position doesn't alter.

e.g. The corresponding machine coordinate for G54 coordinate system origin is （10，10，10）.

The corresponding machine coordinate for G55 coordinate system origin is（30，30，30）.

When the program is executed by sequence, the absolute coordinate and the machine coordinate of the end point are shown as follows:

Program	Absolute coordinate	Machine coordinate
G0 G54 X50 Y50 Z50	50，50，50	60，60，60
G55 X100 Y100	100，100，70	130，130，60
X120 Z80	120，100，80	150，130，110

5. The external workpiece zero offset or workpiece zero offset can be altered by G10, which is shown as following:

By instruction G10 L2 Pp X_Y_Z_

P=0 ： External workpiece zero offset

P=1 to 6 ： Workpiece zero offset of workpiece coordinate system from 1 to 6

X_Y_Z_ ： For absolute instruction（G90）, it is workpiece zero offset of each axis

For incremental instruction（G91）, it is workpiece zero offset set plusing each axis(the result is the new workpiece zero offset).

By G10 instruction, each coordinate system can be altered respectively.

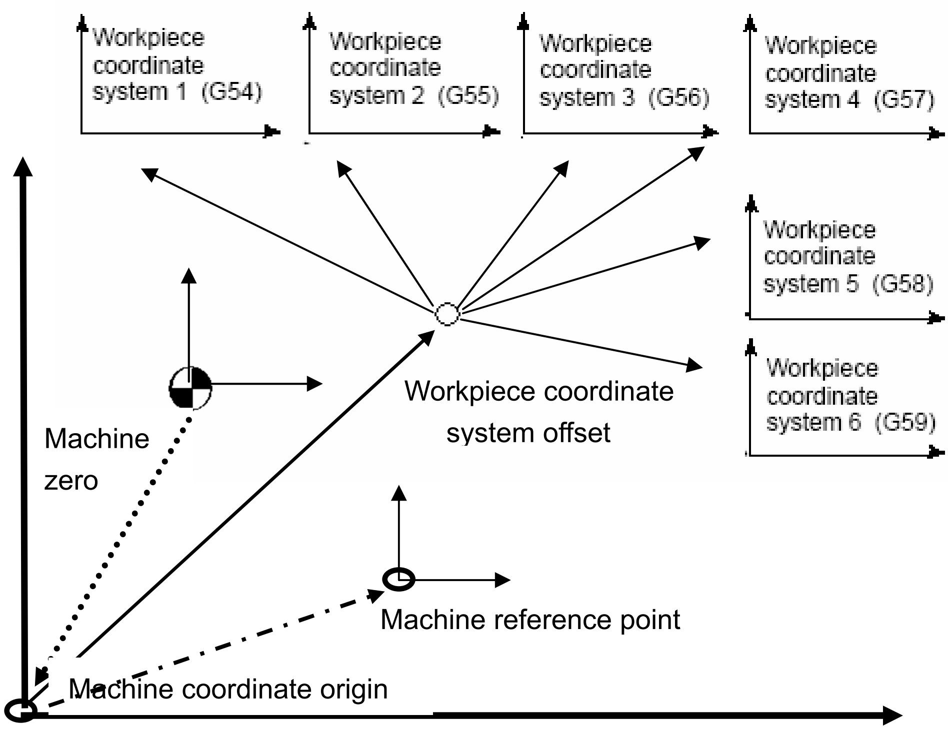
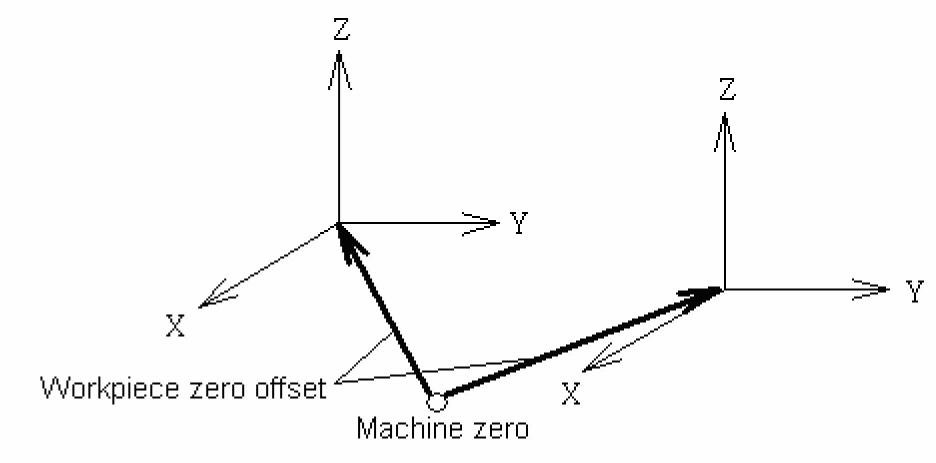


Fig. 4-2-8-1

As shown in Fig. 4-2-8-1, after power-on, the machine returns to machine zero by manual zero return. The machine coordinate system is set up by machine zero with the machine reference point generating and workpiece coordinate system to be defined. The corresponding values of offset number parameter P10～14 in workpiece coordinate system are the integral offset of the 6 workpiece coordinate system. The 6 workpiece coordinate system origins can be specified by coordinate offset input in MDI mode or set by number parameter P15～44. These 6 workpiece coordinate systems are set up by the distances from machine zero to each coordinate system origin.



Example：

N10 G55 G90 G00 X100 Y20；

N20 G56 X80.5 Z25.5；

For the example above, when N10 block is being executing, it rapidly traverses to a position （X=100，Y=20）in G55 workpiece coordinate system.

When N20 block is being executing, the absolute coordinate value automatically turns to the coordinate value （X=80.5，Z=25.5）in G55 workpiece coordinate system for rapid positioning.

4.2.9 Additional workpiece coordinate system

Except 6 workpiece coordinate system (standard workpiece coordinate system) from G54 to G59, 50 additional workpiece coordinate systems can be used.

Format: G54 Pn

Pn: specified additional workpiece coordinate system code

Range ： 1～50

The setting and restriction of the additional workpiece coordinate system are the same as that of workpiece coordinate system from G54 to G59.

The workpiece zero offset in additional workpiece coordinate system can be set by G10, as following:

By instruction G10 L20 Pn X_Y_Z_

Pn=0 ： The workpiece zero offset code for workpiece coordinate system specified.

n=1 to 50： Additional workpiece coordinate system code

X_Y_Z_ ： Set axis address and offset value for workpiece zero offset.

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

For incremental instruction (G91), the new offset value can be gotten by adding the value specified to the current offset value.

By G10 instruction, each workpiece coordinate system can be changed respectively.

4.2.10 Machine coordinate system selection G53

Format: G53 X_ Y_ Z_

Function: To rapidly position the tool to the corresponding coordinate location in the machine coordinate system.

Explanation:

1 While G53 is used in program, the instruction coordinate behind it should be the coordinate in the machine coordinate system and the machine will position to the location specified.

2 G53 is a non-modal instruction, which is effective in block containing it, and it doesn't affect the coordinate system defined before.

Restriction

Machine coordinate system selection G53

When the position in the machine coordinate system is specified, the tool rapidly traverse to this position. The G53 used for selecting machine coordinate system is a non-modal G code, which is only effective for the block specifying the machine coordinate system. Absolute G90 should be specified for G53; if G53 is specified in incremental mode (G91), G91 is neglected (G53 is still in G90 mode without changing G91 mode). The tool can be specified to move to a special position, e.g. in, G53 can be used in program to position the tool to the tool changing point.

After power on

Machine coordinate system must be set before G53 is specified after power on. Therefore, manual reference point return must be performed after power on(zero return in manual mode) or auto reference point return must be specified by G28. If an absolute position encoder is used, this operation is unneeded.

4.2.11 Floating coordinate system G92

Format: G92 X_ Y_ Z_

Function: It is used to set floating workpiece coordinate system. The current tool absolute coordinate values in the new workpiece coordinate system are specified by 3 instruction parameters. And this instruction doesn't' result in the axis movement.

Explanation:

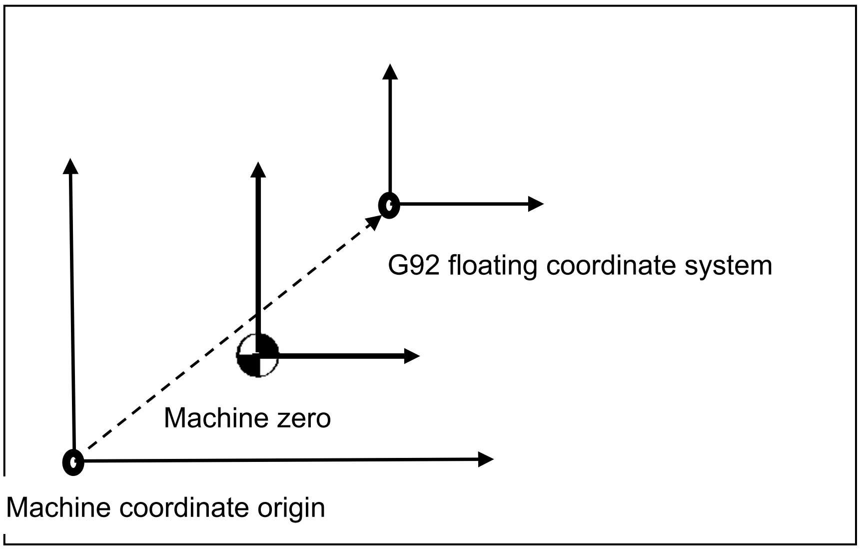


Fig. 4-2-11-1

1、As the figure shows, the origin of the G92 floating coordinate system is the value in machine coordinate system, which is irrelevant to the workpiece coordinate system, it can be set up after the machine zero return.

G92 setting is effective in the following conditions:

1) Before system power off
2) Before workpiece coordinate system is called
3) Before machine zero return

The G92 floating coordinate system is usually used for the alignment of temporary workpiece machining and it will be lost after the power is off. And G92 is usually used at the program beginning or specified in MDI mode before the program auto run.

2、There are two methods for defining the floating coordinate system:

（1）By tool nose:

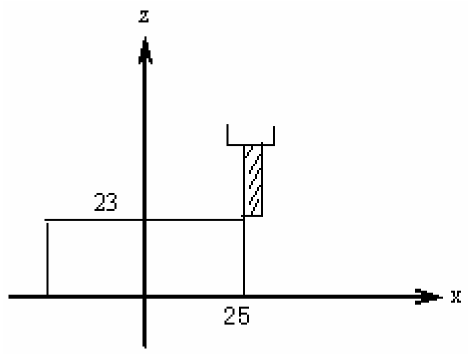


Fig. 4-2-11-2

As fig. 4-2-11-2 shows, for G92 X25.3 Z23，take the position the tool locates at as the point（X25.3， Z23）in the floating coordinate system,

（2）By a fixed point in the arbor as a basic point:

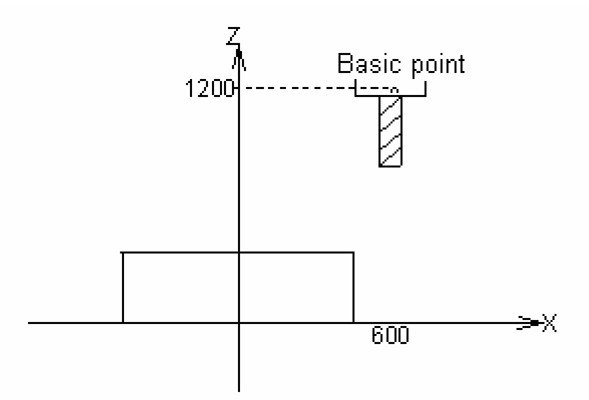


Fig. 4-2-11-3

As Fig. 4-2-11-3 shows, specify the workpiece coordinate system by block “G92 X600 Y1200”(by a basic point in the arbor as a start point). Regarding a basic point as the start point, if the motion is specified by the absolute value in the program, the basic point is moved to the specified position and it must be added the tool length compensation value, which is the difference of the basic point to the tool nose.

Note 1　If G92 is used for coordinate system setting in tool offset, the coordinate system is the one set by G92 as to the tool length compensation without the offset value added.

Note 2　For tool radius compensation, the tool offset should be cancelled if G92 is used.

Restriction

After floating coordinate system is set, the 1st canned cycle instruction should be in a complete format, or the tool move will be wrong.

4.2.12 Plane selection G17/G18/G19

Format: G17/G18/G19

Function: For circular interpolation, tool radius compensation, drilling or boring, plane selection is needed, which can be selected by G 17/G18/G19.

Explanation:

It has no instruction parameter. The system default at power-on is G17 plane if parameter is not specified. It can also be set by bit parameter NO.31#1, #2, #3. The relation of the instruction and the plane is as following:

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

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

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

Plane is not changed if G17，G18，G19 is not specified in the block.

For example:

G18 X_ Z_； ZX plane

G0 X_ Y_； Plane unchanged (ZX plane)

In addition the moving instruction is irrelevant to the plane selection. e.g. in the following instruction, Y axis is not in the ZX plane, so the Y axis moving is irrelevant to ZX plane.

G18Y_；

Annotation: Only the canned cycle in G17 plane is available in this system at present. For criterion or astringency, plane should be expressly defined in the corresponding block, especially in a system used by many users, which can avoid the incident or abnormity caused by programming error.

4.2.13 Polar coordinate system setup/cancel G16/G15

Format: G16/G15

Function:

G16 is used for the setup of the polar coordinate system of the positioning parameter.

G15 is used for the cancellation of the polar coordinate system of the positioning parameter.

Explanation:

No command parameter.

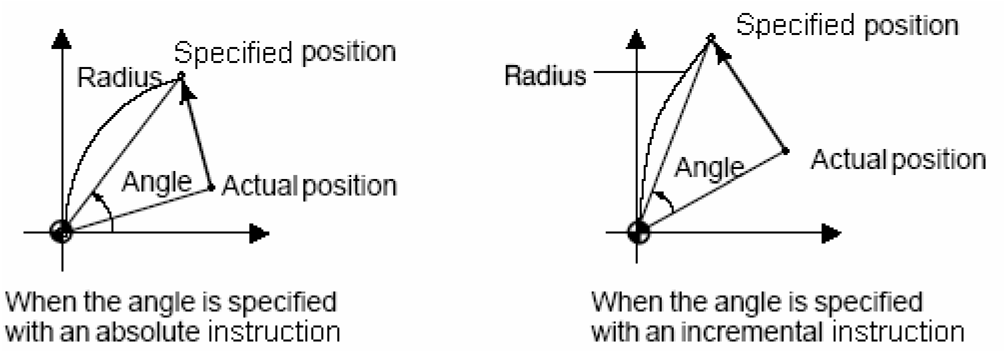
If G16 is set, the coordinate value can be input by polar coordinate radius and angle. The positive of angle is the CCW direction of the 1st axis positive direction in a plane selected; while the negative is CW direction. Both the radius and angle can use the absolute or incremental instructions（G90 ，G91）.

If G16 is used, the 1st axis of the positioning parameter of the tool moving command represents the polar radius in polar coordinate system, the 2nd axis of that represents the polar angle in polar coordinate system.

If G15 is specified, the polar coordinate system can be cancelled and the control returns to the Cartesian coordinate system.

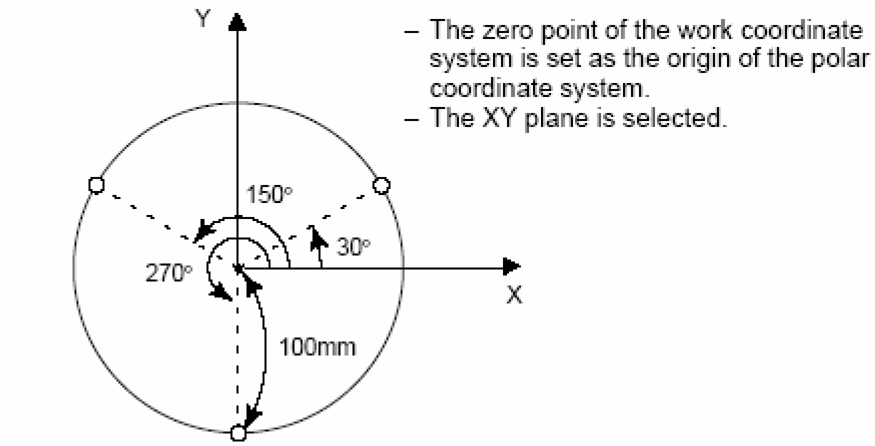
The definition of the polar coordinate system origin:

1 In G90 absolute mode, if G16 is specified, the workpiece coordinate system origin is regarded as the polar coordinate system origin.



2 In G91 incremental mode, if G16 is specified, the current point is regarded as the polar coordinate system origin.

Example: Bolt hole circle (the workpiece coordinate system zero point set as the polar coordinate system origin, selecting X-Y plane)



- To specify angle and radius by absolute value

G17 G90 G16; To specify polar coordinate system and take the workpiece coordinate system zero point in X-Y plane as the polar coordinate system origin

G81 X100.0 Y30.0 Z-20.0 R-5.0 F200.0; To specify 100mm distance and 30°angle

Y150; To specify 100mm distance and 150°angle

Y270; To specify 100mm distance and 270°angle

G15 G80; To cancel the polar coordinate system

- To specify angle by incremental value, polar radius by absolute value

G17 G90 G16; To specify the polar coordinate system and take the workpiece coordinate system zero point in X-Y plane as the polar coordinate system origin

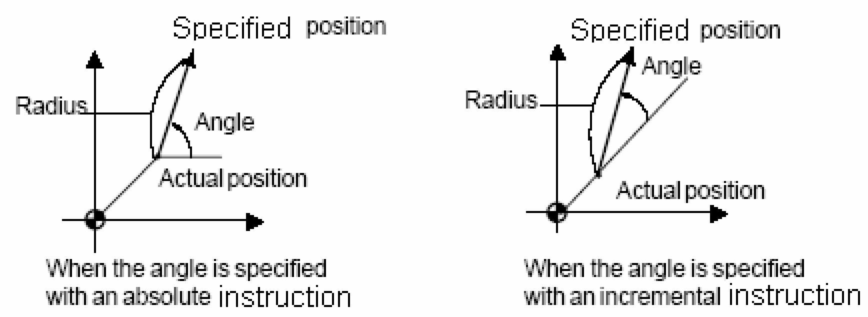
G81 X100.0 Y30.0 Z-20.0 R-5.0 F200.0; To specify 100mm distance and 30°angle

G91 Y120; To specify 100mm distance and 150°angle

Y120; To specify 100mm distance and 270°angle

G15 G80; To cancel the polar coordinate system

Moreover, when programming by polar coordinate system, the current coordinate plane setting should be considered. And the polar coordinate plane and the current coordinate plane are relevant. e.g. in G91 mode, if the current coordinate plane is specified by G17, the origin of it is defined by the X,Y axis components of the current tool position. If the current coordinate plane is specified by G18, the origin of it is defined by the Z, X axis components of the current tool position.



If the positioning parameter of the 1st hole cycle after G16 instruction is not specified, the tool current position is the default positioning parameter of the hole cycle. The 1st canned cycle instruction after the current polar coordinate must be complete, or the tool moving will be wrong.

After G16 instruction, except the hole cycle, the words of the positioning parameter for tool moving involves with the special plane selection mode. While the polar coordinate system is cancelled by G15 which followed by a moving instruction, the tool current position is defaulted as the start point of the moving instruction.

4.2.14 Scaling in plane G51/G50

Format:

G51 X_ Y_ Z_ P_ (Absolute instruction for scaling center coordinate, P: axis scaling by a same ratio)

… Scaling processing blocks

G50 Scaling cancel

or G51 X_ Y_Z_ I_ J_ K_（scaling by different ratios （I, J, K）by each axis）

… Scaling processing block

G50 Scaling cancel

Function:

G51 is used for the programming figure scaling in a same or different ratio by a position specified as the center. G51 is needed to be specified in a single block and cancelled by G50.

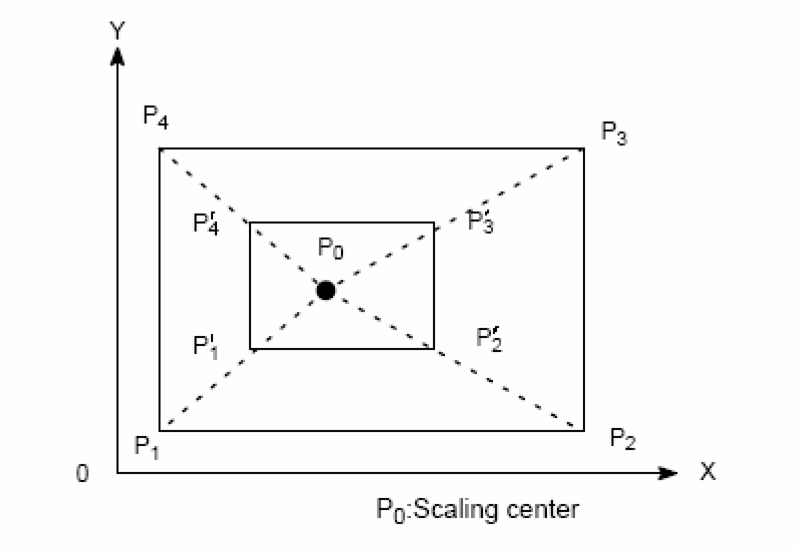


Fig. 4-2-14-1 Scaling (P1'P2P3P4→ P1'P2'P3'P4')

Explanation:

1 Scaling center: G51 can be specified with 3 positioning parameters X_Y_Z_, which are optional. These positioning parameters are used to specify the scaling center of G51. If they are not specified, the tool current position will be specified for the scaling center. Whether the positioning mode is absolute or incremental, the scaling center is specified by the absolute positioning mode. Moreover, in polar coordinate system G16 mode, the parameters in G51 are expressed by Cartesian coordinate system.

Example:

G17 G91 G54 G0 X10 Y10;

G51 X40 Y40 P2; Though in incremental mode, the scaling center means the absolute coordinate（40,40）in G54 coordinate system

G1 Y90; By incremental mode as for parameter Y

2 Scaling: whether the current mode is G90 or G91, the scaling are always expressed by absolute mode.

Except specified in program, the scaling can also be specified in parameters. The number parameters P331~335 correspond to the scaling ratios of X, Y, Z, 4^{TH} and 5^{th} respectively. If no scaling is specified, the number parameter P330 can be used for scaling setting.

If the parameter P or I, J, K value specified are negative, the mirror image is made for the corresponding axis.

3 Scaling setting: The effectiveness of the single axis scaling is set by bit parameter NO.47#3, the effectiveness of the axis scaling mirror image is set by bit parameter NO.47#6, and the ratio

unit of it is set by bit parameter NO.47#7.

4 Scaling cancellation: After the scaling is cancelled by G50 followed by a moving instruction, if the coordinate rotation is cancelled by default, the current tool position is regarded as the start point of this moving instruction.

5 In scaling mode, G codes for reference point return （G27～G30 etc.）and coordinate system specification（G52～G59 , G92 etc.）can't be specified. If needed, they should be specified after the scaling is cancelled.

6 Even different scalings are specified for circular interpolation and axes, the ellipse path cann't be made by tool.

If the scaling ratios of the axes are different and the circular interpolation are programmed by R, the interpolation figure is shown as Fig. 4-2-14-2, (below the scaling ratio of X is 2, that of Y is 1)

```
G90 G00 X0.0 Y100.0 ;
G51 X0.0 Y0.0 Z0.0 I2000 J1000;
G02 X100.0 Y0.0 R100.0 F500 ;
```

Above instructions are equivalent to the following instruction:

```
G90 G00 X0.0 Y100.0 Z0.0 ;
G02 X200.0 Y0.0 R200.0 F500 ;
```

Magnification of radius R depends on I, or J whichever is larger.

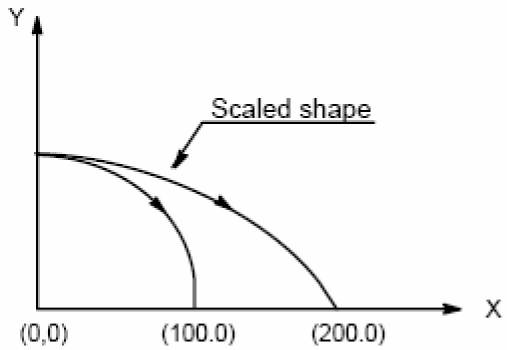


Fig. 4-2-14-2 Scaling of circular interpolation 1

If the axes scaling ratio are different, and the circular interpolation is programmed by I, J, K. the interpolation figure is shown as Fig. 4-2-14-3(in following example, X scaling ratio is 2, Y scaling ratio is 1).

```
G90 G00 X0.0 Y0.0 ;
G51 X0.0 Y0.0 I2000 J1000;
G02 X100.0 Y0.0 I0.0 J–100.0 F500 ;
```

Above instructions are equivalent to the following instructions.

```
G90 G00 X0.0 Y100.0;
G02 X200.0 Y0.0 I0.0 J–100.0 F500 ;
```

In this case, the end point does not beet the radius, a linear section is included.

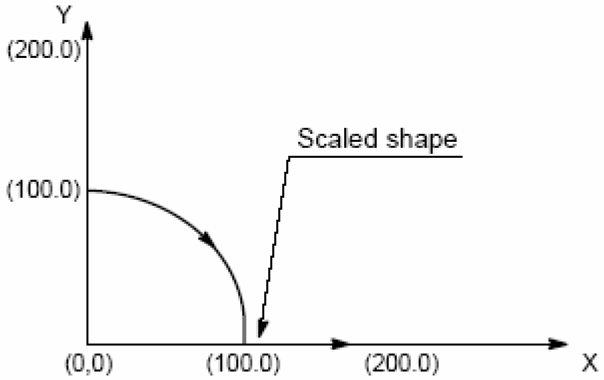


Fig. 4-2-14-3 Scaling of circular interpolation 2

7 Scaling is ineffective for the tool radius compensation, tool length compensation and tool offset, which is shown in Fig. 4-2-14-4.

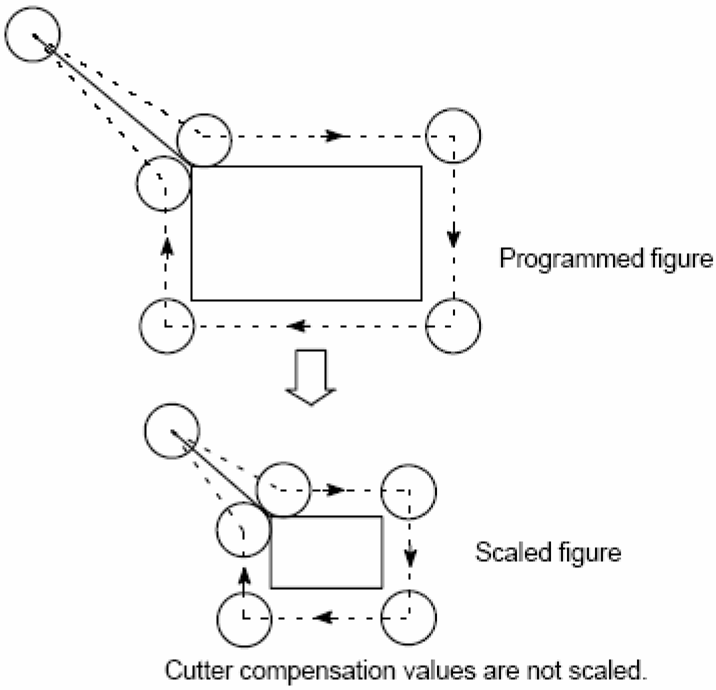


Fig. 4-2-14-4 Scaling of tool radius interpolation

Example for mirror image program:

Main program

G00 G90;

M98 P9000;

G51 X50.0 Y50.0 I1 J-1;

M98 P9000;

G51 X50.0 Y50.0 I-1 J-1;

M98 P9000;

G51 X50.0 Y50.0 I-1 J1;

M98 P9000;

G50;

Subprogram

O9000

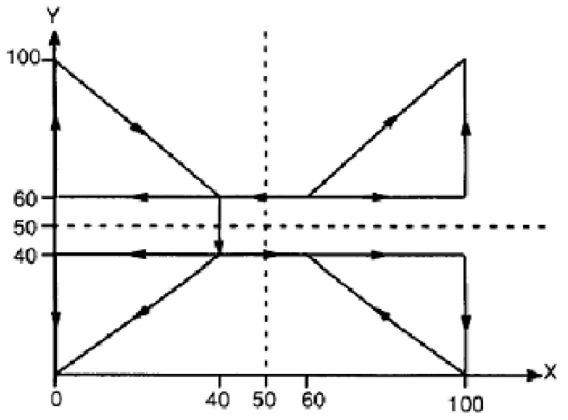
G00 G90 X60.0 Y60.0;

G01 X100.0 F100;

G01 Y100;

G01 X60.0 Y60.0;

M99;



Restriction

1 The moving scaling of Z axis is ineffective in following canned cycles:

1） The cut-in value Q and retraction value d of peck drilling cycle（G83, G73）

2） Fine boring cycle（G76）.

3） Offset value of X axis and Y axis in back boring cycle（G87）.

2 In JOG mode, the traverse distance can't be increased or decreased by scaling.

Note 1 The position is displayed by scaling coordinates.

Note 2 The result for an axis performing mirror image in a specified plane is as following:

1）Circular instruction……………….reverse direction of rotation

2）Tool radius compensation C……….reverse direction of offset

3）Coordinate system rotation…………….reverse direction of rotation angle

4.2.15 Coordinate system rotation G68/G69

A programmed shape can be rotated. When a workpiece comprises some identical shapes, this function can be used for programming by prepairing a subprogram for the shape unit, then calling it by rotation function.

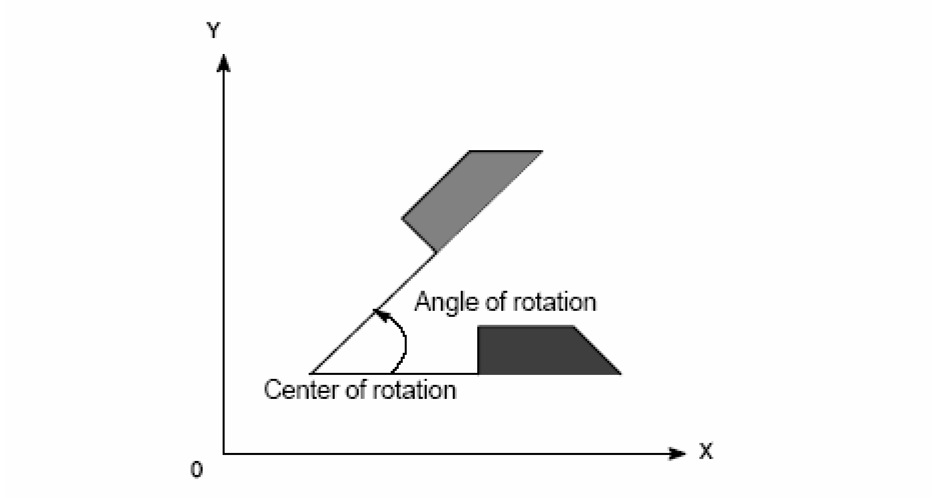
Format: G17 G68 X_ Y_ R_

or G18 G68 X_ Z_ R_

or G19 G68 Y_ Z_R_

G69

Function: G68 is used for the programming shape in plane rotating by a center point specified as an origin. G69 is used for cancellation of coordinate system rotation.



Explanation:

1 G68 is an optional parameter with 2 positioning parameters that are used for specifying the rotation center. If the rotation center is not specified, the tool current position is regarded as the center by system. The positioning parameters are relevant to the current coordinate plane, while X, Y for G17; Z, X for G18; Y, Z for G19.

2 Whether the current positioning mode is absolute or incremental, the rotation center can only be specified by absolute positioning of Cartesian coordinate system.

G68 can be followed by a command parameter R, the value of the parameter is the angle to be rotated. The positive value is for CCW rotation and the angle unit is degree. If no rotation angle is specified in this function, the angle will be set by number parameter P329.

3 In G91 mode, the rotation angle=last rotation angle +current angle specified by R in G68 instruction.

4 When the system is in rotation mode, plane selection is not allowed, or errors will be shown. Attention should be paid in programming.

5 In coordinate system rotation mode, G codes for reference point return （G27～G30 etc.）and coordinate system specification（G52～G59 , G92 etc.）can’t be specified. They should be specified after the scaling is cancelled if needed.

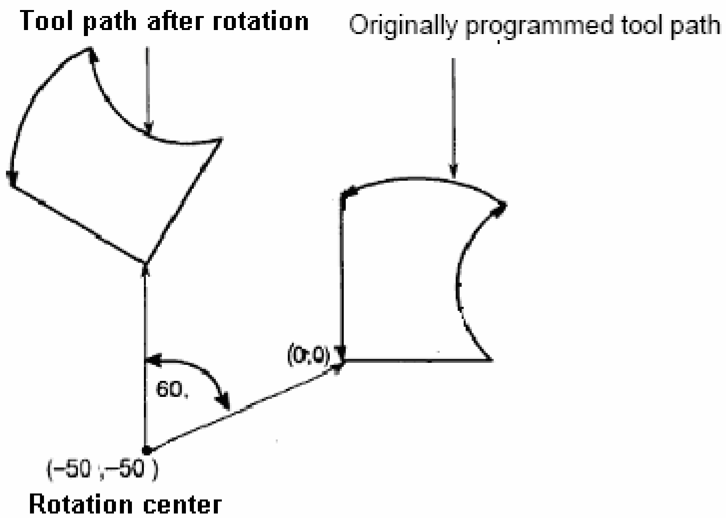
6 After coordinate system rotation, the tool radius compensation, tool length

compensation, tool offset and other compensation operation will be performed.

7 If coordinate system rotation is performed in scaling mode(G51), the rotation center coordinate values will be scaled. If the rotation angle is not scaled, when the moving instruction is given, the scaling will be executed first, then the coordinate system rotation. In scaling mode(G51), the coordinate system rotation instruction （G68）can't be given in tool radius compensation（G41, G42）, it should always be specified before tool radius compensation.

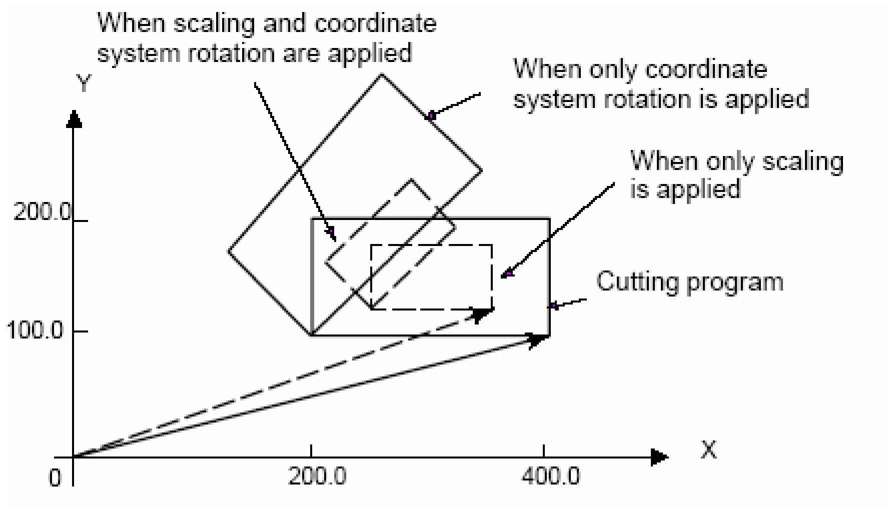
Example 1: Rotation

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



Example 2: Scaling and rotation

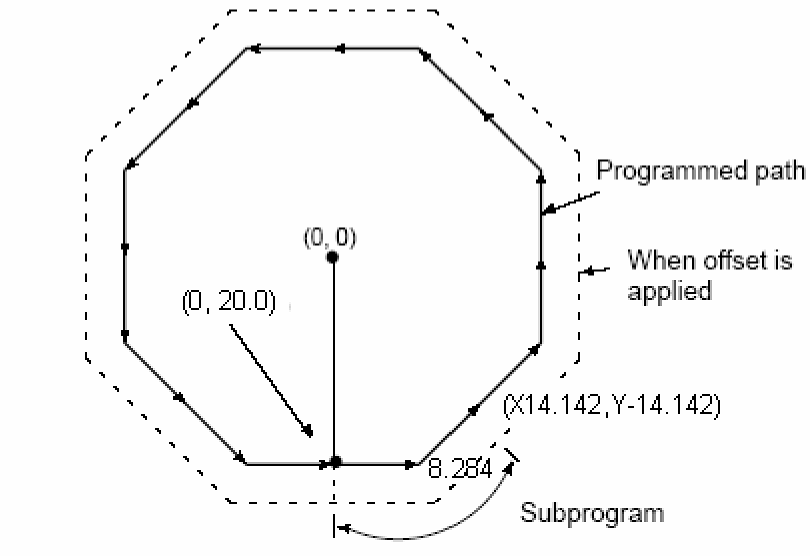
```
G51 X300 Y150 P0.5;
G68 X200 Y100 R45;
G01 G90 X400 Y100;
G91 Y100;
X-200;
Y-100;
X200;
G69 G50;
```



Example 3 : Repetition of G68

By program (main program)

```
G92 X0 Y0 Z20 G69 G17;
M3 S1000;
G0Z2 ;
G51   X0 Y0 I1.2 J1.2
G42   D01；                     (offset setting)
M98   P2100 (P02100);           (subprogram call)
M98   P2200L7；                 (calling for 7 times)
G40
G50
G0    G90 Z20;
X0Y0
M30;

Subprogram 2200
O2200 G68 X0 Y0 G91 R45.0;          (relative rotation angle)
G90;
M98 P2100;                          (subprogram O2200 calling subprogram O2100)
M99；

Subprogram O2100
O2100 G90 G0 X0 Y-20；           (Right-hand tool compensation setup)
G01Z－2 F200；
X8.284；
X14.142 Y-14.142；
M99;
```



4.2.16 Skip function G31

Format: G31 X_Y_Z_

Function: The linear interpolation can be specified like G01 after G31 instruction. During the execution of G31, the current instruction execution will be interrupted to execute next block if an external skip signal is entered. While the working end point is specified not by programming but by signals from machine, this function can be used (e.g. used for grinding). It can also be used for measuring the workpiece dimensions.

Explanation:

1、G31 is a non-modal G code that is only effective in a specified block.

2、Alarm occurs if G31 is given during the tool radius compensation. The tool radius compensation should be cancelled before G31 instruction.

Example:

The block after G31 is a single axis moving specified by incremental values, as Fig. 4-2-16-1 shows:

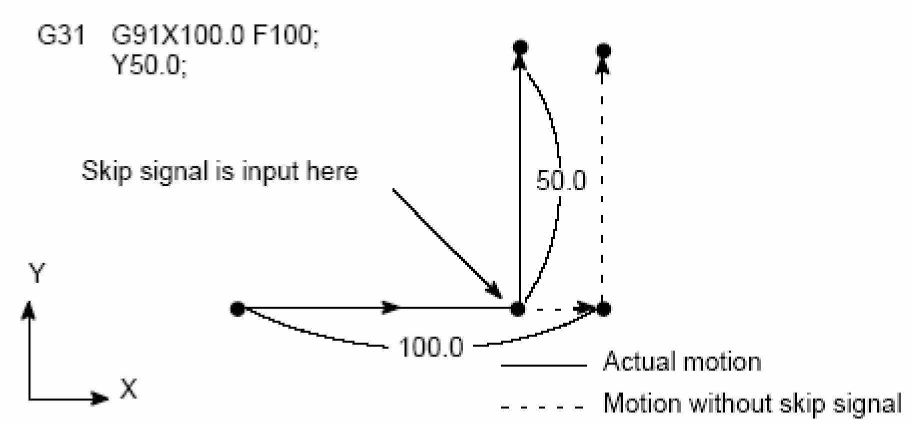


Fig. 4-2-16-1 A single axis moving specified by incremental values of next block

The block after G31 is a single axis moving specified by absolute values, as Fig. 4-2-16-2 shows:

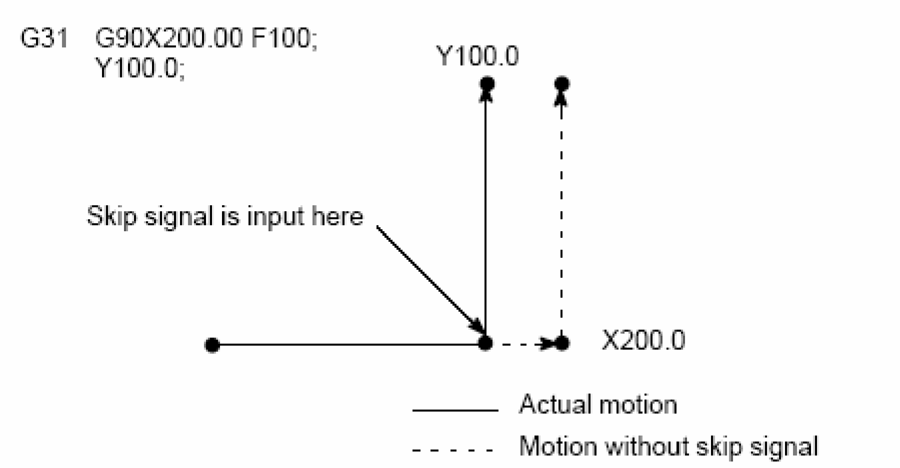


Fig. 4-2-16-2 Single axis moving specified by absolute values of next block

The block after G31 is 2-axis moving specified by absolute values, as Fig. 4-2-16-3 shows:

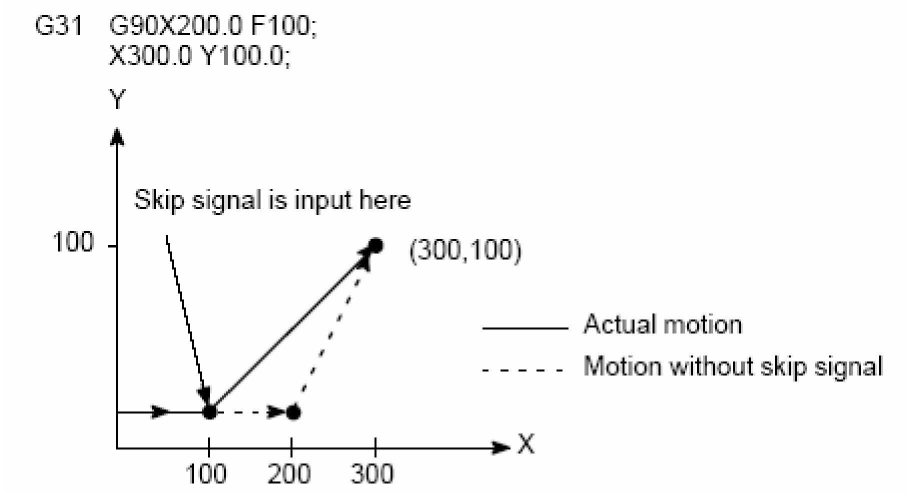


Fig. 4-2-16-3 2-axis moving specified by absolute values of next block

4.2.17 Inch/metric conversion G20/G21

Format: G20： input by inch system

G21： input by metric system

Function: They are used for the inch/metric input conversion in program.

Explanation:

1 This function must be specified by a single block at the beginning of the program before the coordinate system setup.

2 Change the unit of the following item after the inch/metric conversion:

Feedrate specified by F code

Position instruction

Workpiece zero offset value

Tool compensation value

Scale unit of MPG

Moving distance in incremental feeding

Some parameters

The G code status at power-on is the same as that in power-off.

Note 1 Inch/metric conversion can't be executed during the program execution.

Note 2 The tool compensation value must be preset by the minimum incremental input unit when inch system is converted to metric system or the reverse.

Note 3 For the 1st G28 instruction, the running from the intermediate point is the same as the JOG reference point return when inch system is converted to metric system or the reverse.

Note 4 When the minimum incremental input unit is different from the minimum command unit, the maximum error that is not accumulated is the half of the minimum command unit .

Note 5 The inch/metric system for program input can be set by bit parameter NO.00#2.

Note 6 The inch/metric system for program output can be set by bit parameter NO.03#0.

4.2.18 Optional angle chamfering/corner rounding

Format: L_：chamfering

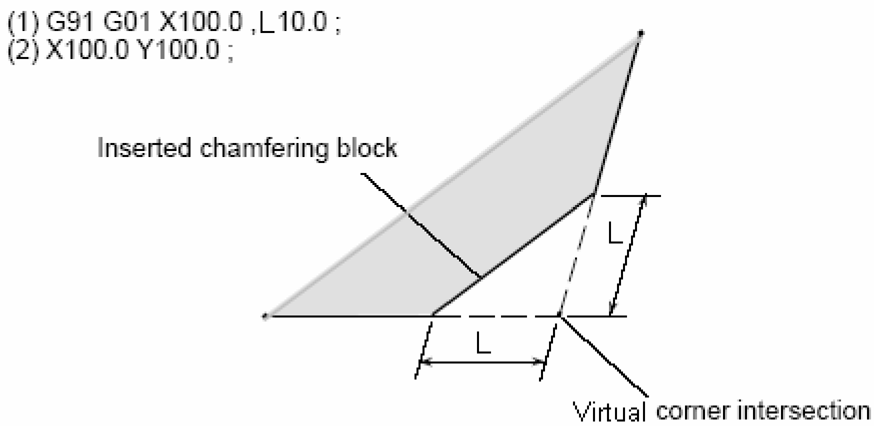
R_：corner rounding

Function: When the above instruction is —added to the end of a block that specifies linear interpolation（G01）or circular interpolation（G02, G03）, a chamfering or corner rounding is automatically done in the machining. Blocks specifying chamfering and corner rounding can be specified consecutively.

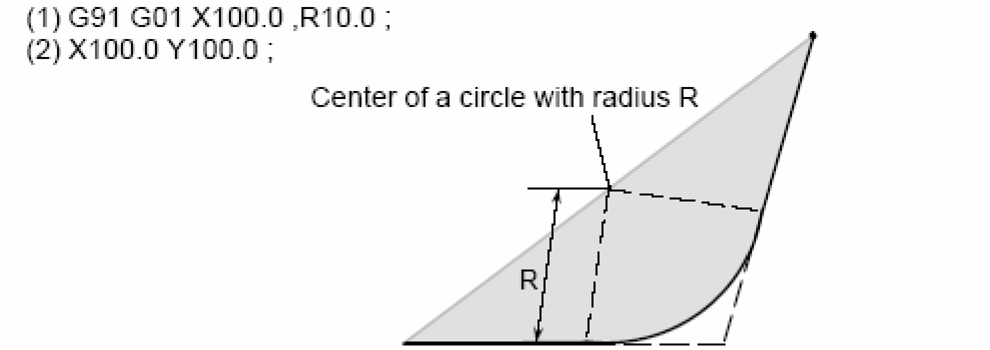
Explanation:

1、Blocks specifying chamfering and corner rounding can only be inserted between the linear interpolation blocks.

2、The chamfering after L is used to specify the distance from the virtual corner point to the start and the end point. The virtual corner point is the corner point that exists if chamfering is not performed. As the following figure shows:



3、The corner rounding after R is used to specify the radius for corner. As the following figure shows:



Restriction

1 Chamfering and corner rounding can only be performed in the plane specified, and these functions can't be performed for parallel axes.

2 A block specifying chamfering or corner rounding must be followed by a block that specifies a linear interpolation. If next block is not linear block, alarm is issued.

3 A chamfering or corner rounding block can be inserted only for move instructions which are performed in the same plane. If plane is switched, neither chamfering nor corner rounding can be specified in a block.

4 If the inserted chamfering or corner rounding block causes the tool to go beyond the original interpolation move range, alarm is issued.

5 In a block that comes after the coordinate system is changed or a reference point return is specified, neither the chamfering nor corner rounding can be specified.

6 Corner rounding can't be specified in a threading block.

7 Optional angle chamfering or corner rounding can't be used in DNC operation.

8 The chamfering and corner rounding value can't be negative, or alarm is issued.

4.3 Reference point G code

The reference point is a fixed point on a machine tool to which the tool can easily be moved by the reference point return function. There are 3 instructions for reference point as is shown in Fig. 4.3.1.1, the tool can be automatically moved to the reference point via an intermediate point along an axis specified by G28; or from the reference point automatically to a specified point via an intermediate point along a specified axis by G29.

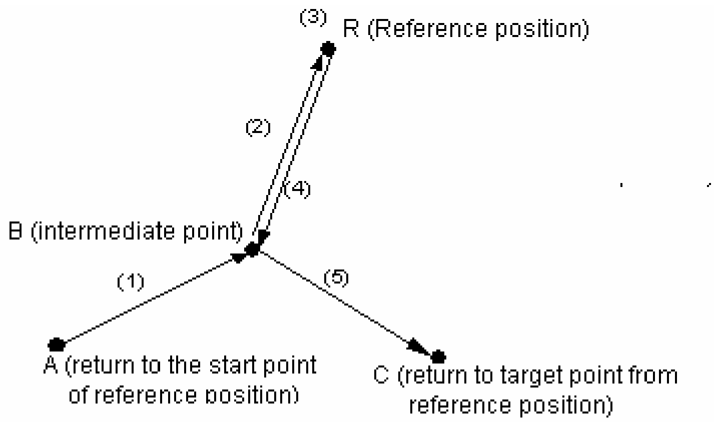


Fig. 4-3-1

4.3.1 Reference point return G28

Format: G28 X_ Y_ Z_

Function: It is used for the operation to return to the reference point (a special point on machine) via an intermediate point.

Explanation:

Intermediate point:

An intermediate point is specified by an instruction parameter in G28, which can be expressed by absolute or incremental instructions. During the execution of this block, the coordinate value of the intermediate point of the axis specified is stored that is to be used for the G29(returning from the reference point) instruction.

Note: **The coordinate value of the intermediate point is stored in the CNC system. Only the axis coordinate value specified by G28 is stored each time, for the other axes not specified by G28, the coordinate values specified by G28 before are used. If the intermediate point defaulted by the system is not ensured by user when using G28 instruction, it is better to specify all the axes. Take a consideration by N5 block in the following example 1.**

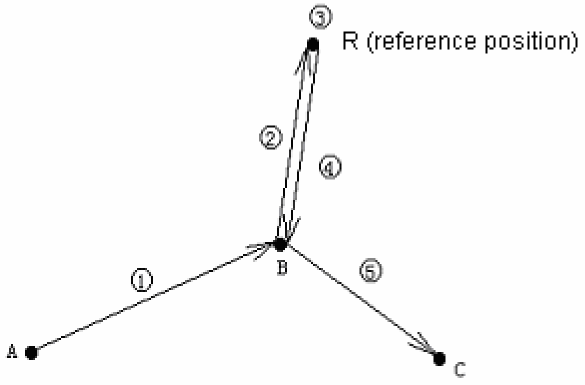


Fig. 4-3-1-1

1 The action of the G28 block can be analyzed as following: （refer to Fig.4-3-1-1）:

(1) Positioning to the intermediate point of the specified axis from the current position (point A→point B) at a traverse speed.

(2) Positioning to the reference point from the intermediate point (point B →point R) at a traverse speed.

2 G28 is a non-modal instruction which is only effective in current block.

3 The combined reference point return of a single axis or multiple axes is available in this system. And the intermediate point coordinate is saved by system during the workpiece coordinate system change.

Example 1:

N1 G90 G54 X0 Y10;

N2 G28 X40 ; Set the intermediate point of X axis for X40 in G54 workpiece coordinate system, and return to reference point via point（40,10）, i.e. reference point return of single X axis

N3 G29 X30 ; Return to the point (30，10) via point（40,10）from reference point, i.e. target point return of single X axis

N4 G01 X20;

N5 G28 Y60 ; Intermediate point(X40，Y60), which is substituted by X40 specified by G28 before due to it is not specified in X axis.

Note The intermediate point is not （20，60）.

N6 G55; Due to workpiece coordinate system change, the intermediate point (40，60) in G54 workpiece coordinate system is changed for (40，60) in G55 workpiece coordinate system.

N7 G29 X60 Y20; Return to the point (60, 20) via the intermediate point (40，60) in G55 workpiece coordinate system from reference point

The G28 instruction can automatically cancel the tool compensation and this instruction is only used in automatic tool change mode(changing tool at the reference point after reference point return). So the tool radius compensation and tool length compensation should be cancelled before using this instruction. See the 1^{st} reference point setting in number parameter P45～P49.

4.3.2 2nd, 3rd, 4th reference point return G30

There are 4 reference points in machine coordinate system. In a system without an absolute-position detector, the 2^{nd}, 3^{rd}, 4^{th} reference point return functions can be used only after the auto reference point return（G28） or manual reference point return is performed.

Format: G30 P2 X_ Y_ Z_； the 2^{nd} reference point return (P2 can be omitted)

G30 P3 X_ Y_ Z_； the 3^{rd} reference point return

G30 P4 X_ Y_ Z_； the 4^{th} reference point return

Function: It is used for the operation of returning to the specified point via the intermediate point specified by G30 from the reference point.

Explanation:

1 X_ Y_ Z_； Instruction for specifying the intermediate point (absolute/ incremental)

2 The specification and restriction for G30 instruction is the same as G28 instruction. See number parameter P50～64 for the 2^{nd}, 3^{rd}, 4^{th} reference point setting.

3 The G30 code can also be used together with G29 code (return from reference point), whose setting and restriction are identical with G28 code.

4.3.3 Automatic return from reference point G29

Format: G29 X_ Y_ Z_

Function: It is used for the operation of returning to a specified point via the intermediate point specified by G28, G30 from the reference point (or current point).

Explanation:

1 The action of the G29 block can be analyzed as following: （refer to Fig.4-3-1-1）：

(1) Positioning to the intermediate point (point R→point B) specified by G28, G30 from the reference point at a traverse speed.

(2) Positioning to a specified point from the intermediate point (point B →point C) at a traverse speed.

2 G29 is a non-modal instruction which is only effective in current block. Usually return from reference point should be specified immediately after G28, G30 instruction.

3 The optional parameters X,Y and Z in G29 instruction are used for specifying the target point (i.e. point C in Fig. 4-3-1-1) from the reference point, which can be expressed by absolute or incremental instruction. The instruction specifies the incremental value from the intermediate point in incremental programming. If an axis is not specified it means the axis

has no moving relative to the intermediate point. The G29 instruction followed by an axis is a single axis return with no action taken by other axes.

Example 1

G90 G0 X10 Y10；

G91 G28 X20 Y20； Reference point return via the intermediate point(30，30)

G29 X30； Return to (60，30) from the reference point via the intermediate point(30，30). **Note:** The component in X axis should be 60 in incremental programming.

The intermediate point of G29 instruction is assigned by G28, G30. Refer to G28 explanation for the definition, criterion and system default of the intermediate point.

4.3.4 Reference point return check G27

Format: G27 X_ Y_ Z_

Function: It is used for the reference point return check, the reference point is specified by X_ Y_ Z_ (absolute/incremental instruction).

Explanation:

1、G27 instruction positions the tool at a traverse speed. If the tool reaches the reference point, the reference point return indicator lights up. However, if the position reached by the tool is not the reference point, an alarm is issued.

2、In machine lock mode, even G27 is specified and the tool has automatically returned to the reference point, the indicator for return completion doesn’t light up.

3、In an offset mode, the position to be reached by the tool with G27 instruction is the position obtained by adding the offset. Therefore, if the position with the offset added is not the reference point, the indicator does not light up, and an alarm is issued. Usually the tool offset should be cancelled before G27 instruction.

4.4 Canned cycle G code

Canned cycle make it easier for the programmer to creat programs. With a canned cycle, a machining operation by multiple blocks can be realized by a single block which contains G function. (In this system only canned cycle in G17 plane is available)

The general process of canned cycle:

A canned cycle consists of a sequence of 6 operations, as Fig. 4-4-1 shows:

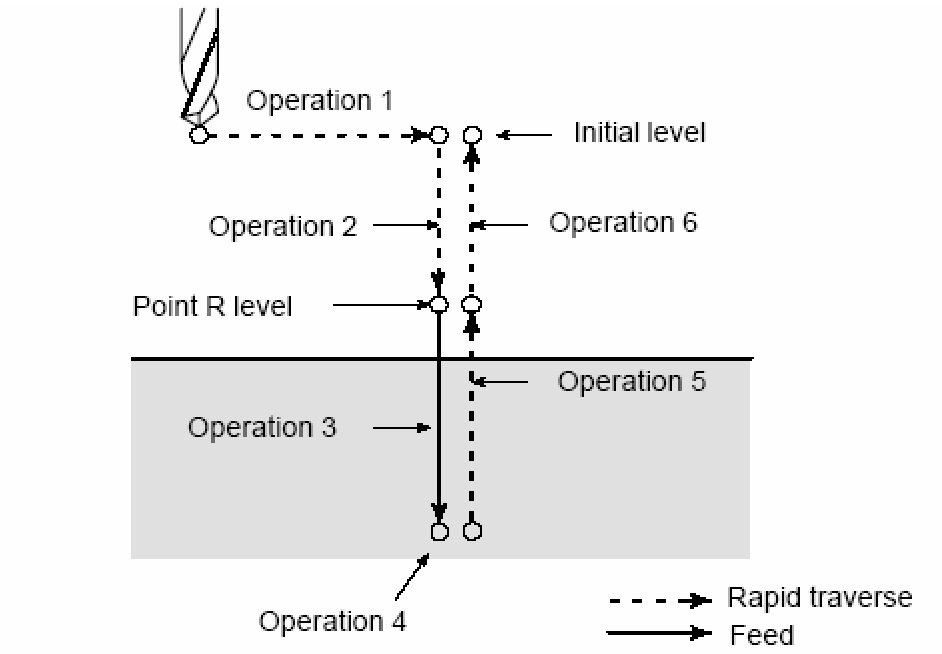


Fig. 4-4-1

Operation 1: Positioning of axes X and Y (may including another axis)

Operation 2: Traverse to point R level

Operation 3: Hole machining

Operation 4: Operation at the bottom of a hole

Operation 5: Retraction to point R level

Operation 6: Traverse to the initial point

The hole machining can be performed in Z axis if positioned in XY plane. It defines that a canned cycle operation is determined by 3 types. They are all specified by G code.

1) Data type

G90 absolute mode； G91 incremental mode

2) Return point plane

G98 initial level； G99 R level

3) Hole machining type

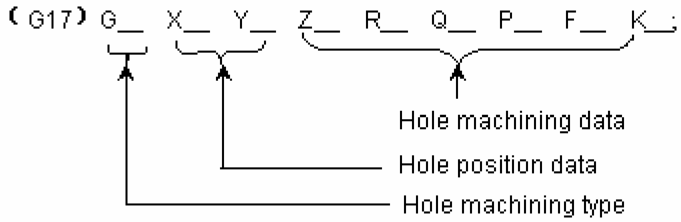
G73， G74， G76， G81～G89

Initial level and R level

Initial level It is the absolute position where the tool locates in Z axis before the canned cycle.

R level It is also called safe plane, it is a position in Z axis when the traverse is switched to the feeding in canned cycle, which is usually positioned at a distance from the workpiece surface to prevent the tool from colliding with the workpiece and provide a sufficient distance to finish the acceleration. The instructions of G73/G74 /G76/G81～G89 specify all the data(hole location data, hole machining data, repetition) , by which a block is constituted.

The format for hole machining is shown as following:



Therein, the significance of the hole location data and machining data is as following Table 4-4-1:

Table 4-4-1

Designation	Parameter word	Explanation
Hole machining	G	Refer to Table 4.4.3，note the restrictions above。
Data for hole location	X，Y	The hole location is specified by absolute value or incremental value and the control is identical to the G00 positioning.
Data for hole machining	Z	As Fig. 4.4.2(A) shows, the distance from point R level to the hole bottom is specified by incremental value, or the hole bottom coordinate is specified by absolute value. And the feedrate is the speed specified by F in operation 3; while in operation 5, it is a traverse speed or a speed specified by F code due to the different machining type.
	R	In Fig. 4.4.2(B), the distance from the initial level to point R level is specified by incremental value or point R level coodinate is specified by absolute value. The speeds in operation 2 and 6 are both traverse.
	Q	It is used to specify the cut-in value or the parallel moving value in G76 or G87.
	P	It is used to specify the dwell time at the hole bottom. The canned cycle instruction can be followed by a parameter P_ , which specifies the dwell time after the tool reaches the Z plane. The time unit is ms. The min. value of the parameter can be set by number parameter P281, and the max. value by number parameter P282.
	F	It is used to specify the cutting feedrate.
	K	The repetition is specified in parameter K_, which is effective only in the specified block. It can be omitted and the default is one time. The max. drilling times are 99999. If a negative value is specified, it executes by absolute values. If zero is specified, the mode is changed without drilling operation.

Restriction

- Drilling instruction G_ _ can't be specified in a single block or alarm is issued by the system.
- The canned cycle is modal instruction, which is effective till it is cancelled by a G code.
- G80 and G codes in 01 group are used for cancelling canned cycle.
- The processing data once specified in canned cycle are effective till the canned cycle is cancelled. Therefore, after all the processing data required for hole machining are specified in the beginning of the canned cycle, only the data to be changed is needed to be respecified in the following canned cycle.

Note The feedrate specified by F remains effective even the canned cycle is cancelled.

In single mode, the canned cycle has 3 stage working type, positioning→R level→initial level

In canned cycle, the data of hole machining and hole position will be eliminated if the system is reset. The instance of dada retained and eliminated is shown as following table:

Table 4-4-2

No.	Designation of data	Explanation
①	G00X-M3；	
②	G81X-Y-Z-R-F-；	Specify values for Z, R, F in the beginning.
③	Y-；	G81，Z-R-F- can be omitted due to the identical hole machining mode and data specified in ②. Drill the hole for the length Y once by G81.
④	G82X-P-；	Move in X axis relative to hole ③. Do the hole machining by G82 and data Z，R，F specified in ② and P in ④.
⑤	G80X- Y-	Hole machining is not performed. Cancel all the hole data.
⑥	G85X-Z-R-P-；	Because all data are cancelled in ⑤, Z, R needs to be respecified and F that remains can be omitted. P is saved but not needed in this block.
⑦	X- Z-；	It is a hole machining with a different Z value to ⑥. And there is moving only in X axis.
⑧	G89X-Y-；	Do the hole machining by G89 according to the data Z specified in ⑦, R, P in ⑥ and F in ②.
⑨	G01X-Y-；	Cancel the hole machining mode and data.

A Absolute instruction and incremental instruction in canned cycle G90/G91

The change of G90/G91 along drilling axis is shown as Fig. 4-4-2. (Usually it is programmed by G90, if it is programmed by G91, Z and R are regarded as negative values.)

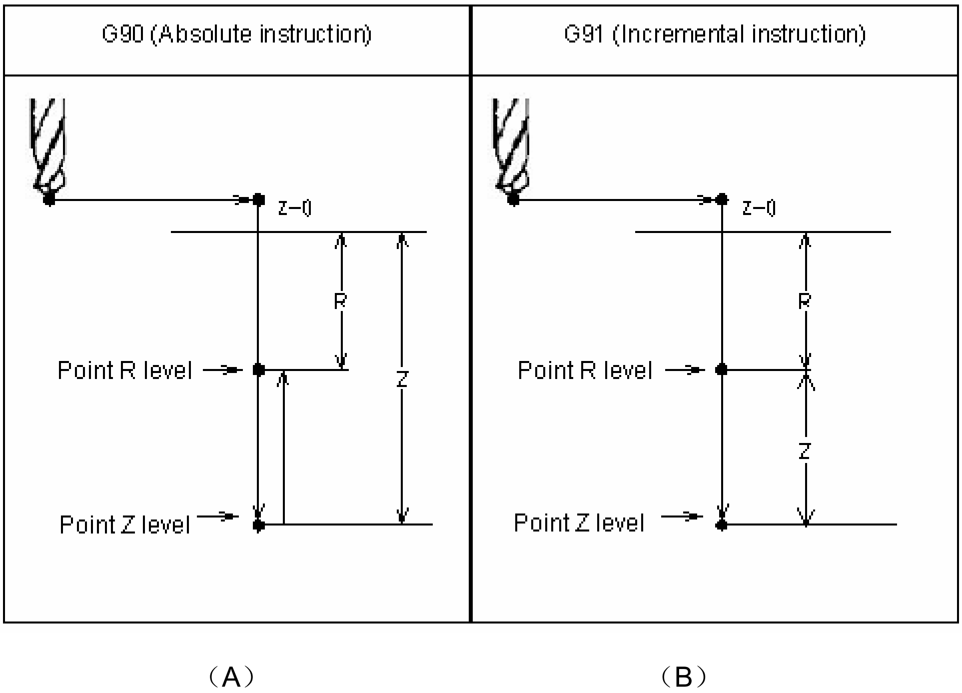


Fig. 4-4-2

B Return to initial level in canned cycle G98/G99

After the tool reaches the bottom of a hole, it may return to the point R level or the initial level. These operations can be specified by G98 and G99.

Generally, G99 is used for the 1st drilling operation and G98 is used for the last drilling operation. The initial level does not change even drilling is performed in G99 mode. The following figure illustrates the operation of G98 and G99.

G98 is the system default mode.

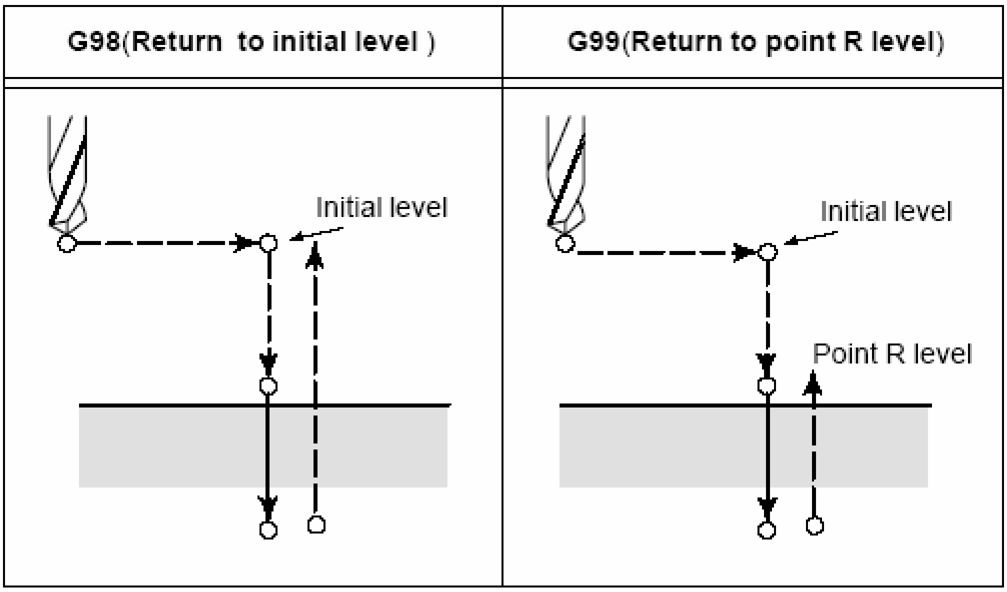


Fig. 4-4-3

The following symbols are used for the canned cycle illustration:

- - -→	Positioning (rapid traverse G00)
——→	Cutting feed (linear interpolation G01)
~~→	Manual feed
⇨	**Offset** (rapid traverse G00)
P	Dwell

Canned cycle comparison table（G22～G89）

Table 4-4-3

G code	Drilling (-Z direction)	Operation at the hole bottom	Retraction(+Z direction)	Application
G73	Intermittent feed		Rapid feed	High-speed peck drilling cycle
G74	Feed	Dwell→spindle CW	Feed	Counter tapping cycle
G76	Feed	Oriented spindle stop	Rapid feed	Fine boring
G80				Cancel
G81	Feed		Rapid feed	Drilling，spot drilling
G82	Feed	Dwell	Rapid feed	Drilling，counterboring
G83	Intermittent feed		Rapid feed	Peck drilling cycle
G84	Feed	Dwell → spindle CCW	Feed	Tapping
G85	Feed		Feed	Boring
G86	Feed	Spindle stop	Rapid feed	Boring
G87	Feed	Spindle CCW	Rapid feed	Boring
G88	Feed	Dwell → spindle CCW	JOG	Boring
G89	Feed	Dwell	Feed	Boring

Restriction

In canned cycle, tool offset is ignored.

In canned cycle mode, R can’t be specified in a single block. i.e. after canned cycle starts, R instruction can’t be programmed by a single block.

4.4.1 Rough milling of circular groove G22/G23

Format:

G98/G99 G22/G23 X_ Y_ Z_ R_ I_ L_ W_ Q_ V_ D_ F_ K_

Function: They are used for circular interpolations from the circle center by helical type till the circular groove programmed is machined.

Explanation:

G22： CCW inner circular groove rough milling

G23： CW inner circular groove rough milling

I： Circular groove radius, it should be over the current tool radius

L：Cut width increment within XY plane, less than tool diameter but more than 0;

W：Initial cut depth in Z axis, which is the distance below R level and it is over 0(if the initial cut depth exceeds the groove bottom, it should machine by this bottom) ;

Q：Cut depth of each feed;

V：Distance to the end surface at rapid tool traverse, which is over 0;

D：Tool diameter number, ranging within 0 ~ 128, D0 is defaulted for 0. The current tool diameter value is got by the given number.

K：Repetitions.

Cycle process:

⑴ Rapid to a location in XY plane;

⑵ Rapid down to R level;

⑶ To cut W depth downward by cutting feedrate;

⑷ From center outward to mill a circle surface with a radius I helically by a L increment each time;

⑸ Z axis rapidly returns to R level;

⑹ X, Y axes rapidly position to the circle center;

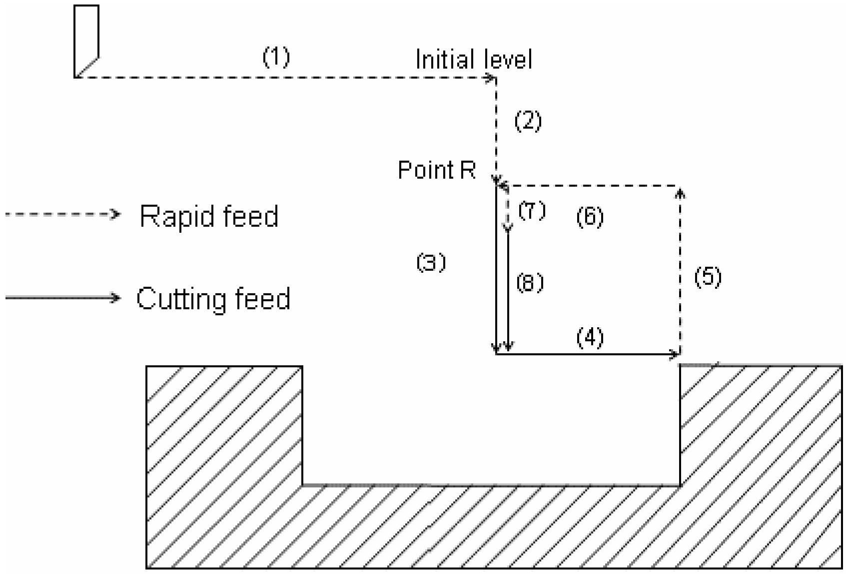
⑺ Z axis rapid downward to a location with a distance V to the end surface;

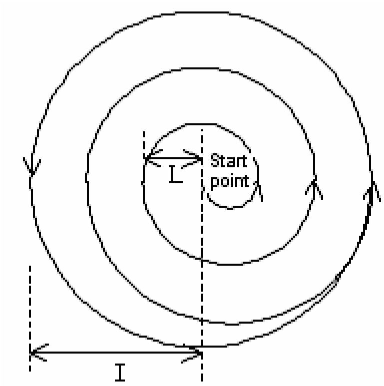
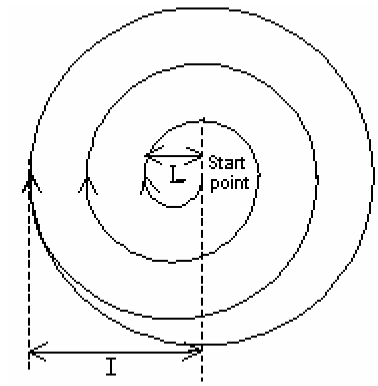
⑻ To cut a（Q+V）depth downward in Z axis;

⑼ Repeat the actions from （4）~（8）till the total depth of circle surface is finished;

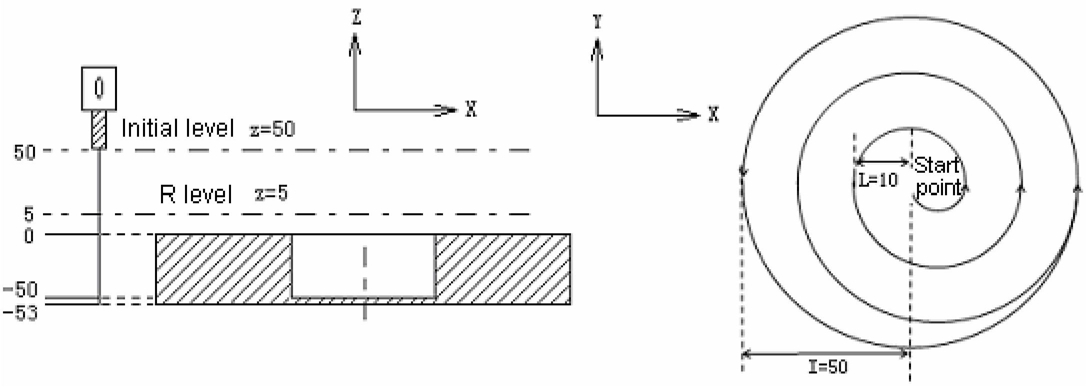
⑽ Return to initial level or R level according to G98 or G99 instruction.

Instruction path:



G22	G23
	

Example: To rough mill a groove within a circle by canned cycle G22 instruction, which is as follows:



G90 G00 X50 Y50 Z50; (G00 rapid positioning)

G99 G22 X25 Y25 Z-50 R5 I50 L10 W20 Q10 V10 F800;

(Groove rough milling cycle within a circle)

G80 X50 Y50 Z50; (Canned cycle cancel and return from R level)

M30;

4.4.2 Fine milling cycle within a circle G24/G25

Format:

G24

G98/G99 **X_ Y_ Z_ R_ I_ J_ D_ F_ K_**

G25

Function: They are used to fine mill a circle by a radius I and direction specified and the tool returns after milling.

Explanation:

G24: CCW fine milling within a circle

G25: CW fine milling within a circle

I: Milling circle radius, ranging within 0 mm ~9999.999mm, use absolute value if it is a negative one;

J: Distance of fine milling start point to circle center, ranging with 0 mm ~9999.999mm, use absolute value if it is a negative one;

D: Tool diameter number, ranging within 0 ~128. D0 is defaulted for 0. The tool diameter value is obtained by the given number.

K: Repetitions

Cycle process:

(1) Rapid to a location within XY plane;

(2) Rapid down to R level;

(3) Feed to the hole bottom;

(4) To position to the start point from current position at the bottom;

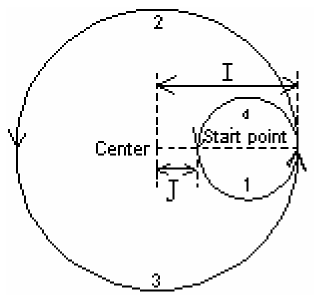
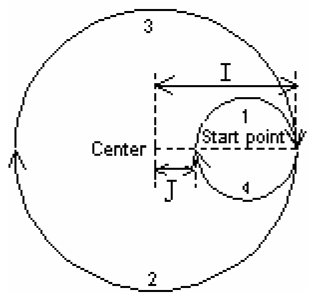
(5) To interpolate by the transition arc 1 from the start point;

(6) To make circular interpolation for the whole circle by arc 2, arc 3

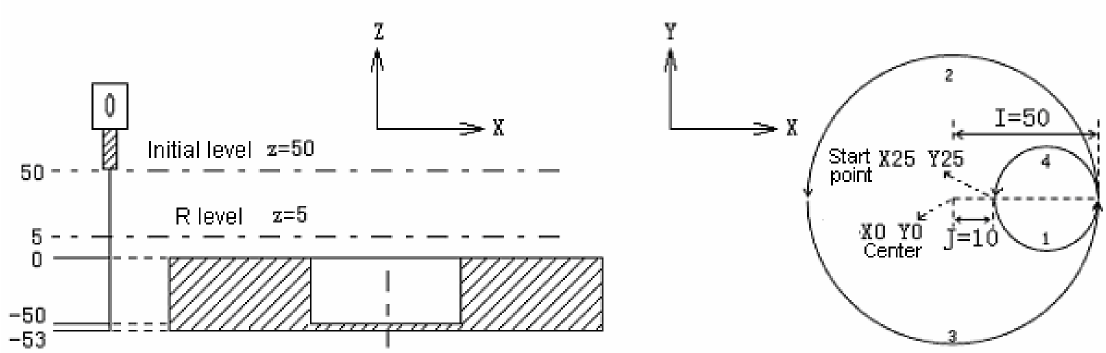
(7) To make circular interpolation by transition arc 4 and return to the start point;

(8) Return to the initial level or R level according to G98 or G99 instruction.

Instruction path:

G24	G25
	

Example: To fine mill a circular groove that has been rough milled as following by canned cycle G24 instruction:



G90 G00 X50 Y50 Z50； (G00 rapid positioning)

G99 G24 X25 Y25 Z-50 R5 I50 J10 F800；

(Canned cycle starts, and goes down to the bottom to perform the inner circle fine milling)

G80 X50 Y50 Z50； (To cancel canned cycle and return from R level)

M30；

4.4.3 Outer circle fine milling cycle G26/G32

Format:

G98/G99 G26/G32 X_ Y_ Z_ R_ I_ J_ D_ F_ K_；

Function: They are used to fine mill a circle outside a circle by the specified radius and direction and the tool returns after milling.

Explanation: For these instructions, refer to canned cycle explanation in Table 13.1.7.

G26： CCW outer circle fine milling

G32： CW outer circle fine milling

I： Fine milling circle radius, ranging within 0 mm ~9999.999mm, use the absolute value if it is a negative one.

J： Distance from the milling start point to milling circle center, ranging within 0 mm ~9999.999mm, use the absolute value if it is a negative one

D：Tool radius number, ranging within 0 ~128, D0 is defaulted for 0. The current tool radius value is obtained by the given number.

K：Repetitions.

Cycle process:

(1) Rapid to a location within XY plane;

(2) Rapid down to R level;

(3) Feed to the hole bottom;

(4) To position to the start point from current position at the bottom;

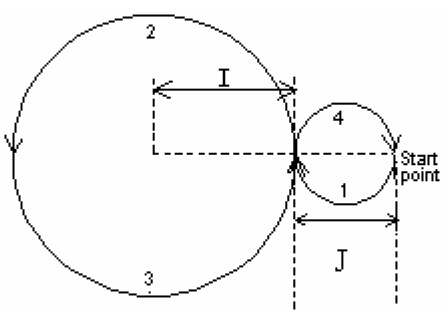
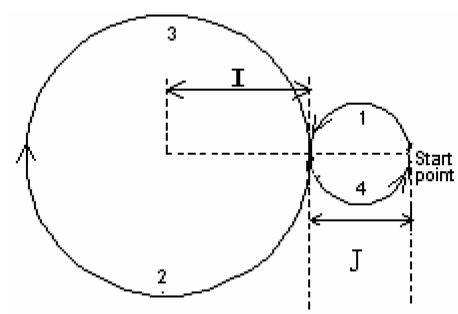
(5) To interpolate by the transition arc 1 from the start point;

(6) To make circular interpolation for the whole circle by arc 2, arc 3

(7) To make circular interpolation by transition arc 4 and return to the start point;

(8) Return to the initial level or R level according to G98 or G99 instruction.

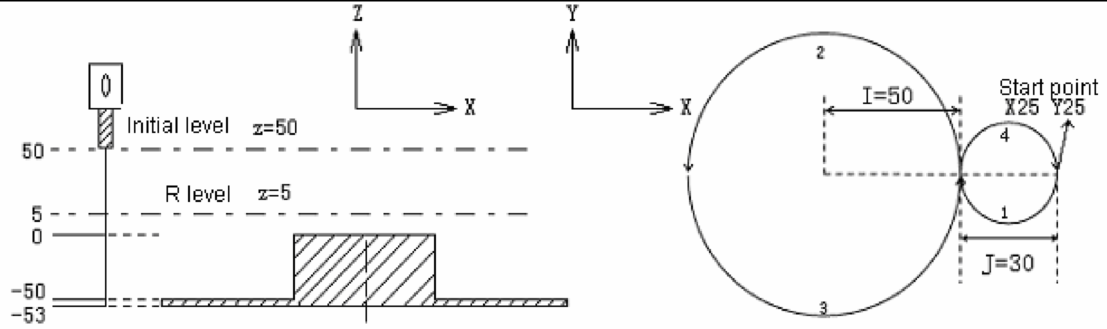
Instruction path:

G26	G32
	

Explanation:

In outer circle fine milling, the interpolation directions of transition arc and fine milling arc are different, while the interpolation direction in the instruction means the interpolation direction of the fine milling.

Example: To fine mill a circular groove that has been rough milled as following by canned cycle G26 instruction:



G90 G00 X50 Y50 Z50; (G00 rapid positioning)

G99 G26 X25 Y25 Z-50 R5 I50 J30 F800; (Canned cycle starts, and goes down to the bottom to perform the outer circle fine milling)

G80 X50 Y50 Z50; (To cancel canned cycle and return from R level)

M30;

4.4.4 Rectangular groove rough milling G33/G34

Format:

G33

G98/G99 X_ Y_ Z_ R_ I_ J_ L_ W_ Q_ V_ U_ D_ F_ K_

G34

Function: These instructions are used for linear cutting cycle from the rectangle center by the parameter data specified till the rectangular groove programmed is machined.

Explanation: For these instructions, refer to canned cycle explanation in Table 13.1.7.

G33: CCW rectangular groove rough milling

G34: CW rectangular groove rough milling

I: Rectangular groove width in X axis

J: Rectangular groove width in Y axis

L: Cutting width increment within a specified plane, which should be less than the tool diameter and over 0

W: Initial cut depth in Z axis, which is a downward distance from R level and is over 0 (if the initial cut exceeds the groove bottom, it will cut at the bottom position)

Q: Cut depth of each cutting feed

V: Distance to the end surface to be machined in rapid feed, which is over 0

U: Corner radius, no corner transition if omitted

D: Tool diameter number, ranging within 0 ~ 128, D0 is defaulted for 0. The current tool diameter value is given by the number specified.

K: Repetitions

Cycle process:

(1) Rapid to a location within XY plane;

⑵ Rapid down to R level;

⑶ To cut a W depth downward by a federate;

⑷ To mill a rectangular surface helically from center outward by L increment each time;

⑸ Z axis rapids to R level;

⑹ X, Y axes rapidly locates to the rectangle center;

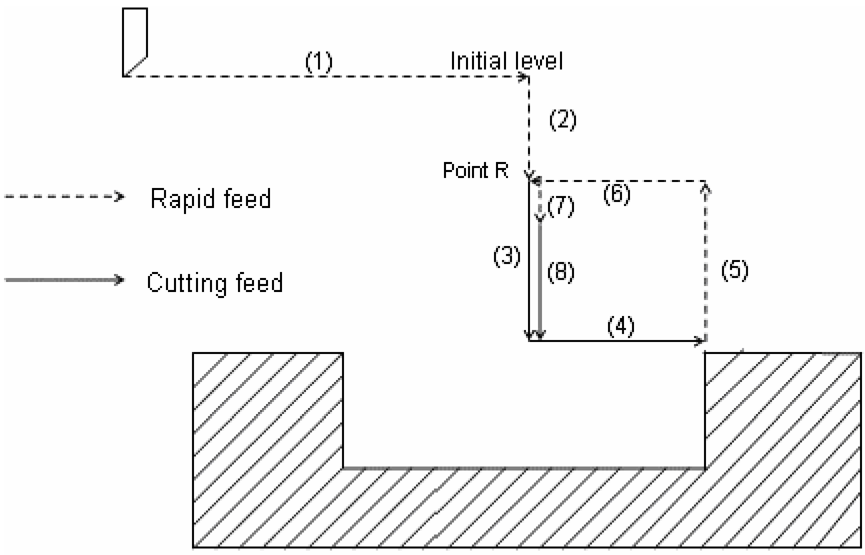
⑺ Z axis rapids down to a position that has a V distance to the end surface;

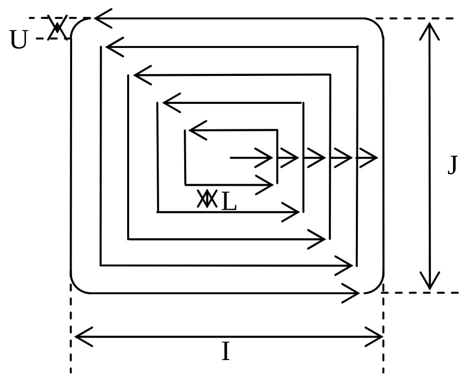
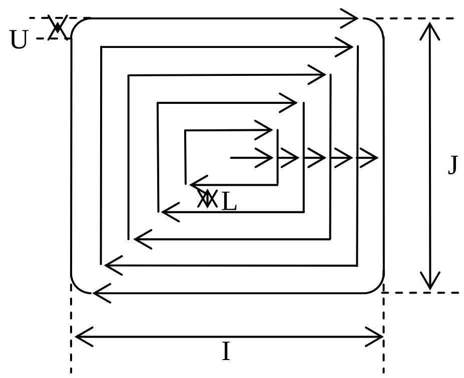
⑻ Z axis cuts downward for a（Q+V）depth;

⑼ Repeat the actions of（4）~（8）till the rectangular surface with the total depth is machined;

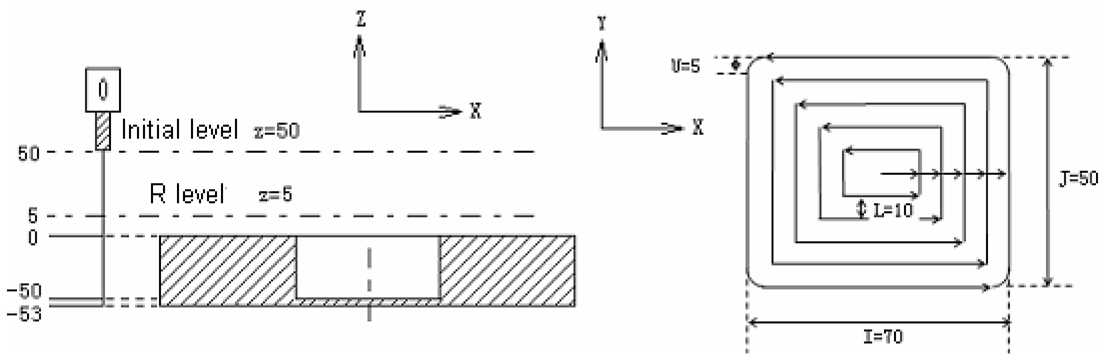
⑽ Return to the initial level or R level according to G98 or G99 instruction.

Instruction path:



G33	G34
	

Example: To rough mill an inner rectangular groove as shown in the following by canned cycle G33 instruction:



G90 G00 X50 Y50 Z50; (G00 rapid positioning)

G99 G33 X25 Y25 Z-50 R5 I70 J50 L10 W20 Q10 V10 U5 F800;

(To run the inner rectangular groove rough milling cycle)

G80 X50 Y50 Z50; (To cancel canned cycle and return from R level)

M30;

4.4.5 Inner rectangular groove fine milling cycle G35/G36

Format:

G35

G98/G99 **X_ Y_ Z_ R_ I_ J_ L_ U_ D_ F_ K_;**

G36

Function: They are used for fine milling within a rectangle by the width and direction specified, and the tool returns after fine milling.

Explanation: For these instructions, refer to canned cycle explanation in Table 13.1.7.

G35: CCW inner rectangular groove fine milling cycle

G36: CW inner rectangular groove fine milling cycle

I: Rectangular width in X axis, ranging within 0~9999.999mm

J: Rectangular width in Y axis, ranging within 0~9999.999mm

L: Distance of start point to rectangular side in X axis, ranging within 0~9999.999mm;

U: Corner radius, no corner transition if omitted. Alarm is issued if U is omitted or equal to 0 and the tool radius is over 0.

D: Tool diameter number, ranging within 0 ~ 128, D0 is defaulted for 0. The current tool diameter value is given by the number specified.

K: Repetitions.

Cycle process:

(1) Rapid to a location within XY plane;

(2) Rapid down to R level;

(3) Feed to the hole bottom;

(4) To position to the start point from current position at the bottom;

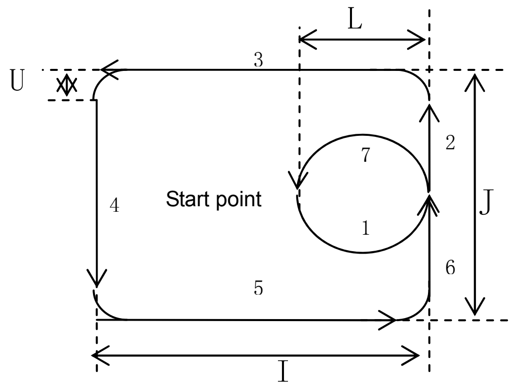
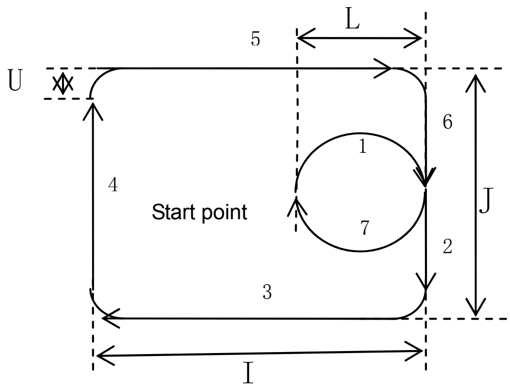
(5) To make circular interpolation by the transition arc 1 from the start point;

(6) To make linear and circular interpolation by the path 2-3-4-5-6;

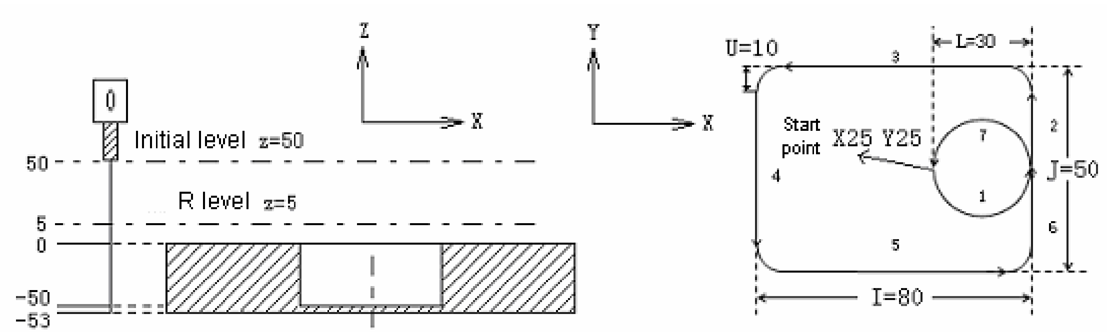
(7) To make circular interpolation by the path of transition arc 7 and return to the start point;

(8) Return to the initial level or R level according to G98 or G99 instruction.

Instruction path:

G35	G36
	

Example: To fine mill a circular groove that has been rough milled as following by canned cycle G35 instruction:



G90 G00 X50 Y50 Z50； (G00 rapid positioning)

G99 G35 X25 Y25 Z-50 R5 I80 J50 L30 U10 F800；

(Canned cycle starts, and go down to the bottom to perform the rectangular groove fine milling)

G80 X50 Y50 Z50； (To cancel canned cycle and return from R level)

M30；

4.4.6 Rectangle outside fine milling cycle G35/G36

Format:

G37

G98/G99 **X_ Y_ Z_ R_ I_ J_ L_ U_ D_ F_ K_**

G38

Function: They are used for fine milling outside a rectangle by the width and direction specified, and the tool returns after fine milling.

Explanation:

G37： CCW rectangle outside fine milling cycle

G38： CW rectangle outside fine milling cycle

I： Rectangular width in X axis, ranging within 0 mm ~9999.999mm

J： Rectangular width in Y axis, ranging within 0 mm ~9999.999mm

L：Distance of start point to rectangular side in X axis, ranging within 0~9999.999mm

U：Corner radius, no corner transition if omitted

D：Tool diameter number, ranging within 0 ~ 128, D0 is defaulted for 0. The current tool diameter value is given by the number specified

K：Repetitions

Cycle process:

(1) Rapid to a location within XY plane;

(2) Rapid down to R level;

(3) Feed to the hole bottom;

(4) To position to the start point from current position at the bottom;

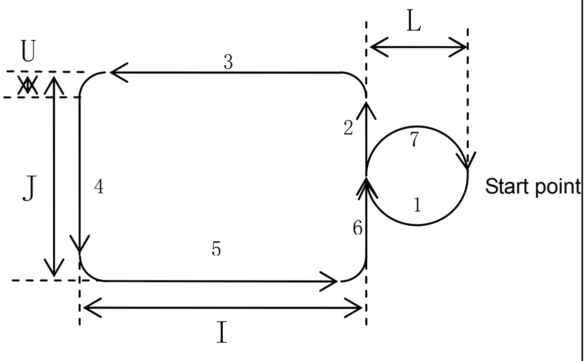
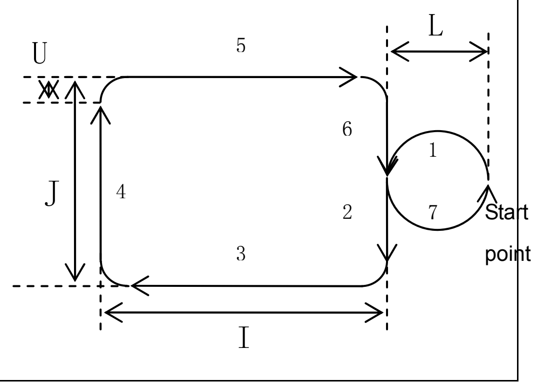
(5) To make circular interpolation by the transition arc 1 from the start point;

(6) To make linear and circular interpolation by the path 2-3-4-5-6;

(7) To make circular interpolation by the path of transition arc 7 and return to the start point;

(8) Return to the initial level or R level according to G98 or G99 instruction.

Instruction path:

G37	G38
	

Explanation:

For the rectangle outside fine milling, the interpolation direction of the transition arc is not consistent with that of the fine milling arc, and the interpolation direction in explanation means that of the fine milling arc.

Example: To fine mill a circular groove that has been rough milled as following by canned cycle G37 instruction:

G90 G00 X50 Y50 Z50； (G00 rapid positioning)

G99 G37 X25 Y25 Z-50 R5 I80 J50 L30 U10 F800；

(Canned cycle starts, and go downward to the bottom to perform the rectangular groove fine milling)

G80 X50 Y50 Z50； (To cancel canned cycle and return from R level)

M30；

4.4.7 High-speed peck drilling cycle G73

Format: G73 X_Y_Z_R_Q_F_K_

Function: This cycle is especially defined for high-speed peck drilling, it performs intermittent cutting feed to the bottom of a hole while removing chips from the hole by rapid retraction. The operation illustration is shown as Fig. 4-4-1-1.

Explanation:

X_Y_： Hole positioning data

Z_： In incremental programming it specifies the distance from point R level to the bottom of the hole; in absolute programming it specifies the absolute coordinate of the hole bottom.

R_： In incremental programming it specifies the distance from the initial level to point R level; in absolute programming it specifies the absolute coordinate of point R.

Q_： Depth of cut for each cutting feed

F_： Cutting feedrate

K_： Number of repeats

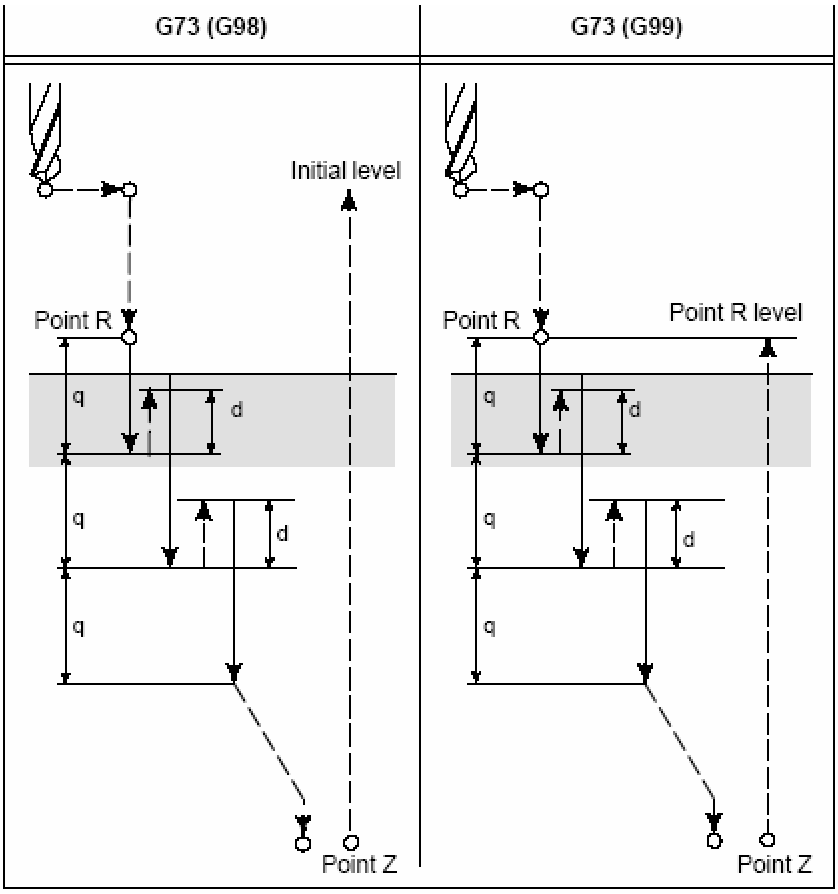


Fig. 4-4-1-1

Z, R: The hole bottom parameter Z and R must be correctly specified while performing the 1st drilling operation (omitting unallowable) or the alarm is issued.

Q: If parameter Q is specified, the intermittent feed is performed as shown in above figure. And the retraction is performed by the retraction value d (Fig.4.4.1.1) set in number parameter P270. The rapid tool retraction for a distance d is performed in each intermittent feeding.

If G73 and M codes are specified in a same block, M code is executed during the 1st hole positioning operation, then the system goes on the next drilling operation.

If the repetition K is specified, M code is only executed for the first hole.

Note 1 If parameter Q is not specified, alarm "address Q not found(G73/G83)" will be issued. If Q value is specified for a negative, the intermittent feed will be performed by the absolute value of Q.

Note 2 In canned cycle, if the tool length compensation (G43，G44 or G49) is specified, the offset value is either added or cancelled while positioning to point R level.

Restriction

Cancellation: Do not specify a G code in 01 group(G00, G01, G02, G03) or G60 in a same block, otherwise alarm occurs.

Tool offset: In canned cycle the tool radius compensation is ignored.

Example 1

M3 S1500	Spindle running start
G90 G99 G73 X0 Y0 Z-15. R-10.Q5. F120.	Positioning and drill hole 1 then return to point R level
Y-50；	Positioning and drill hole 2 then return to point R level
Y-80；	Positioning and drill hole 3 then return to point R level
X10；	Positioning and drill hole 4 then return to point R level
Y10；	Positioning and drill hole 5 then return to point R level
G98 Y75；	Positioning and drill hole 6 then return to initial level
G80；	
G28 G91 X0 Y0 Z0；	Return to reference point
M5；	Spindle stop
M30；	

Note The chip removal operation is still performed though Q is omitted in the machining of the holes from 2 to 6.

4.4.8 Drilling cycle, spot drilling cycle G81

Format: G81 X_ Y_ Z_ R_ F_ K_

Function: It is used for normal drilling feed to the hole bottom, then the tool rapidly retracts from the hole bottom.

Explanation:

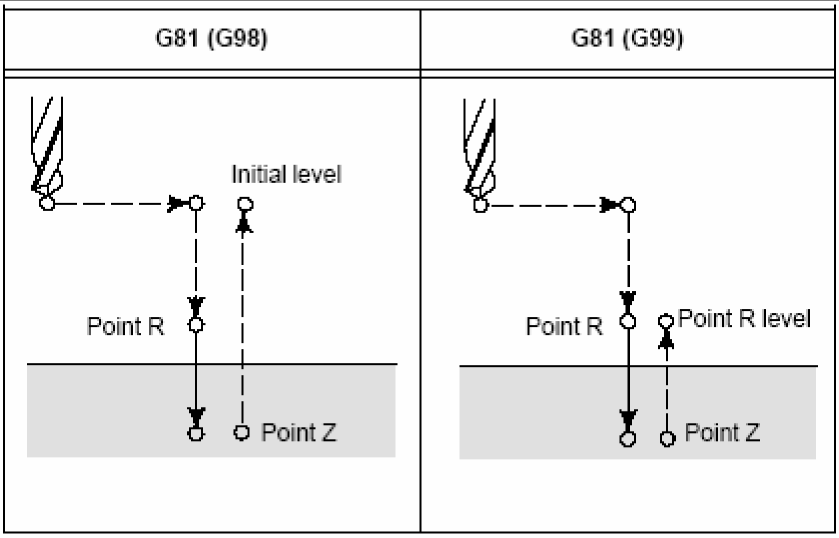
X_Y_： Hole positioning data

Z_： In incremental programming it specifies the distance from point R level to the bottom of the hole; in absolute programming it specifies the absolute coordinate of the hole bottom.

R_： In incremental programming it specifies the distance from the initial level to point R level; in absolute programming it specifies the absolute coordinate of point R level.

F_： Cutting feedrate

K_： Number of repeats (if necessary)



Z, R: The hole bottom parameter Z and R must be correctly specified while performing the 1st drilling operation(omitting unallowable) or the alarm occurs. If parameter P,Q are specified, they are ignored by system.

After positioning along X and Z axes, the tool traverses to point R level to perform the drilling from point R level to point Z level, then retracts rapidly.

The spindle is rotated by miscellaneous function M code before G81 is specified.

If G81 and M code are specified in a same block, M code is executed while the 1st hole positioning operation is being performed, then the system goes on next drilling operation.

If number of repeats K is specified, M code is only executed for the 1st hole.

If the tool length compensation G43, G44 or G49 is specified in canned cycle, the offset is either added or cancelled while positioning to point R level.

Example

M3 S2000 Spindle running start

G90 G99 G81 X300. Y-250. Z-150. R-10. F120. Positioning, drill hole 1, then return to point R level

Y-550.; Positioning, drilling hole 2, then return to point R level

Y-750.; Positioning, drill hole 3, then return to point R level

X1000.; Positioning, drill hole 4, then return to point R level

Y-550.; Positioning, drill hole 5, then return to point R level

G98 Y-750.; Positioning, drill hole 6, then return to initial level

G80;

G28 G91 X0 Y0 Z0 ; Return to reference point

M5; Spindle stops

M30;

Restriction

Cancellation: Do not specify a G code in 01 group (G00, G01, G02, G03) or G60 in a same block, otherwise alarm occurs.

Tool offset: In canned cycle the tool radius compensation is ignored.

4.4.9 Drilling cycle, counterboring G82

Format: G82 X_ Y_ Z_ R_ P_ F_ K_；

Function: It is used for normal drilling to feed to the hole bottom and dwell, then retract the tool rapidly from hole bottom.

Explanation:

X_Y_： Hole positioning data

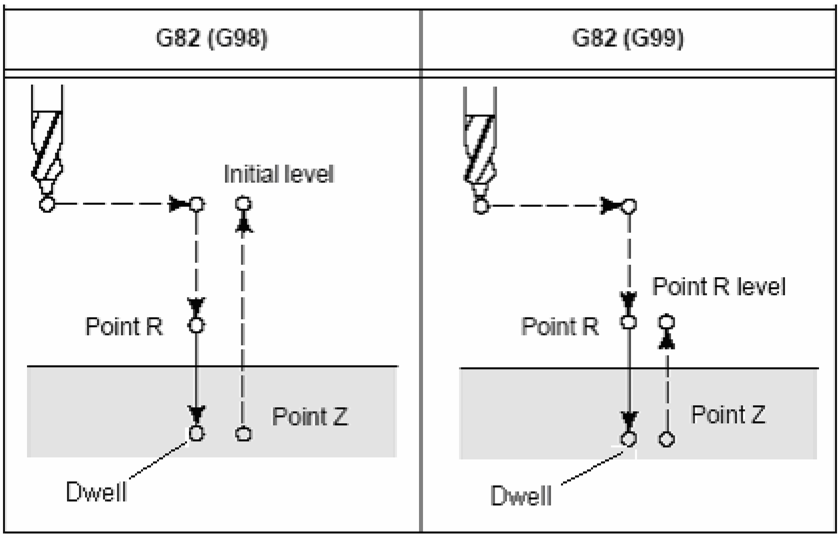
Z_： In incremental programming it specifies the distance from point R level to the bottom of the hole; in absolute programming it specifies the absolute coordinate of the hole bottom.

R_： In incremental programming it specifies the distance from the initial level to point R level; in absolute programming it specifies the absolute coordinate of point R.

F_： Cutting feedrate

P_： Dwell time

K_： Number of repeats



After positioning along X and Z axes, the tool traverses to point R level to perform the drilling from point R level to point Z level, then dwells and returns rapidly after the tool reaches the hole bottom.

The spindle is rotated by miscellaneous function M code before G82 is specified.

If G82 and M code are specified in a same block, M code is executed while the 1st hole positioning operation is being performed, then the system goes on next drilling operation.

If number of repeats K is specified, M code is only executed for the 1st hole.

If tool length compensation G43, G44 or G49 is specified in canned cycle, the offset value is either added or cancelled while positioning to point R level.

P is a modal instruction, and the min. value of it is set by number parameter P281, the max. value by P282. If P value is less than the setting by P281, the min. value is effective; if P value is more than the setting by P282, the max. value is effective. If P is specified in a block containing no drilling, it can't be stored as a modal datum.

Example

M3 S2000 Spindle running start

G90 G99 G82 X300. Y-250. Z-150. R-100. P1000 F120 Positioning, drill hole 1 with 1s dwell at the hole bottom, then return to point R level

Y-550； Positioning, drill hole 2 with 1s dwell at the hole bottom, then return to point R level

Y-750； Positioning, drill hole 3 with 1s dwell at the hole bottom, then return to point R level

X1000.； Positioning, drill hole 4 with 1s dwell at the hole bottom, then return to point R level

Y-550； Positioning, drill hole 5 with 1s dwell at the hole bottom, then return to point R level

G98 Y-750； Positioning, drill hole 6 with 1s dwell at the hole bottom, then return to initial level

G80； Cancel canned cycle

G28 G91 X0 Y0 Z0 ； Return to reference point

M5； Spindle stops

M30；

Restriction

Cancellation: Do not specify a G code in 01 group (G00, G01, G02, G03) or G60 in a same block, otherwise alarm occurs.

Tool offset: In canned cycle the tool radius compensation is ignored.

4.4.10 Drilling cycle with chip removal G83

Format: G83 X_ Y_ Z_ R_ Q_ F_ K_

Function: It is used for peck drilling that the tool feeds to the hole bottom by intermittent feeding with chips removed from hole during drilling.

Explanation:

X_Y_： Hole positioning data

Z_： In incremental programming it specifies the distance from point R level to the bottom of the hole; in absolute programming it specifies the absolute coordinate of the hole bottom.

R_： In incremental programming it specifies the distance from the initial level to point R level; in absolute programming it specifies the absolute coordinate of point R.

Q_： Depth of cut for each cutting feed

F_： Cutting feedrate

K_： Number of repeats

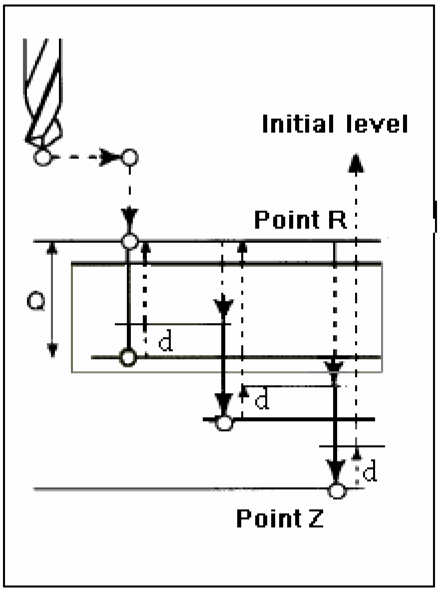
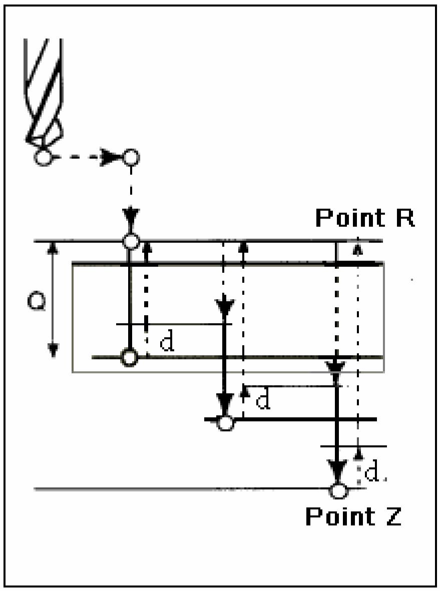
G83 (G98)	G83 (G99)
	

Fig. 4-4-4-1

Q: It specifies each cutting depth expressed by incremental value. In the second and the following feeding, the tool rapidly traverse to the position which has a distance d to the end position of last drilling and still performs the feeding d that is set by parameter P270, as is shown in Fig. 4-4-4-1.

Only positive value can be specified for Q and the negative value is used as a positive one with its negative sign ignored.

Q is specified in drilling block, it can't be stored as a modal datum if it is specified in the block containing no drilling.

The spindle is rotated by miscellaneous function(M code) before G83 is specified.

If G83 and M code are specified in a same block, M code is executed while the 1st hole positioning operation is being performed, then the system goes on next drilling operation.

If number of repeats K is specified, M code is only executed for the 1st hole.

If tool length compensation G43,G44 or G49 is specified in canned cycle, the offset value is either added or cancelled while positioning to point R level.

Example

M3 S2000 Spindle running start

G90 G99 G83 X300. Y-250. Z-150. R-100. Q15 F120; Positioning, drill hole 1, then return to point R level

Y-550; Positioning, drill hole 2, then return to point R level

Y-750；	Positioning, drill hole 3, then return to point R level
X1000；	Positioning, drill hole 4, then return to point R level
Y-550；	Positioning, drill hole 5, then return to point R level
G98 Y-750；	Positioning, drill hole 6, then return to initial level
G80；	
G28 G91 X0 Y0 Z0 ；	Return to reference point
M5；	Spindle stops
M30；	

Restriction

Cancellation: Do not specify a G code in 01 group (G00, G01, G02, G03) or G60 in a same block, otherwise alarm occurs.

Tool offset: In canned cycle the tool radius compensation is ignored.

4.4.11 Right-handed tapping cycle G84

Format: G84 X_ Y_ Z_ R_ P_ F_

Function: It is used for tapping. In tapping, when the tool reaches the hole bottom, the spindle runs reversely.

Explanation:

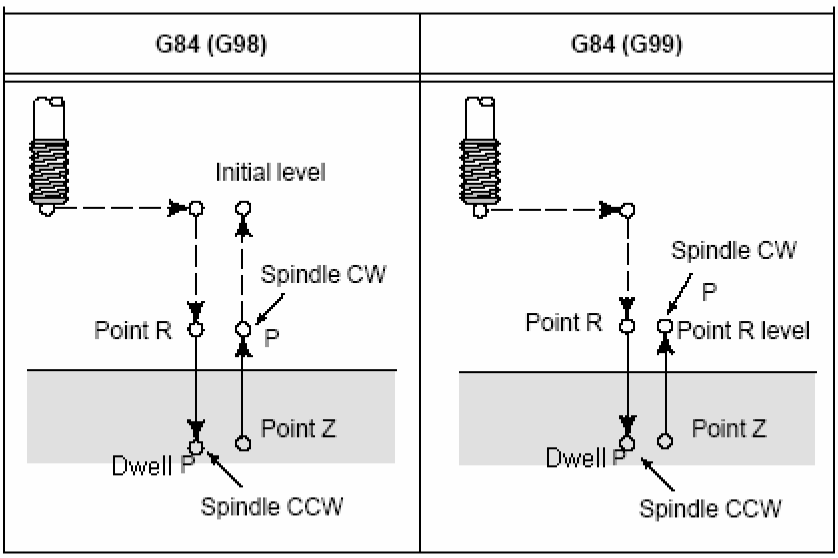
X_Y_： Hole positioning data

Z_： In incremental programming it specifies the distance from point R level to the bottom of the hole; in absolute programming it specifies the absolute coordinate of the hole bottom.

R_： In incremental programming it specifies the distance from the initial level to point R level; in absolute programming it specifies the absolute coordinate of point R.

P_： Dwell time.

F_： Cutting feedrate.



Tapping is performed by rotating the spindle CW, when the tool reaches the hole bottom, the spindle is rotated reversely for retraction. This operation creates threads.

Feedrate overrides are ignored during tapping. A feed hold does not stop the machine until the return operation is finished.

Before specifying G84, use a miscellaneous function (M code) to rotate the spindle. If the spindle CW rotation is not specified, it will be adjusted for CW rotation automatically in R level by the current spindle specification.

If G84 and M code are specified in a same block, M code is executed while the 1st hole positioning operation is being performed, then the system goes on next drilling operation.

If number of repeats K is specified, M code is only executed for the 1st hole.

P is a modal instruction, and the min. value of it is set by number parameter P281, the max. value by P282. If P value is less than the setting by P281, the min. value is used; if P value is more than the setting by P282, the max. value is used. If P is specified in a block containing no drilling, it can't be stored as a modal datum.

If tool length compensation G43, G44 or G49 is specified in canned cycle, the offset value is either added or cancelled while positioning to point R level.

In feeding per minute, the relation between the thread lead and feedrate as well as spindle speed is as following:

Feedrate F=tap pitch×spindle speed S

For example: for the M12×1.5 thread hole on the workpiece, the following parameter can be used:

S500=500 r /min　　F=1.5×500=750mm/min

For multi-start thread, F value can be gotten by multiplying the thread number.

Example:

M3 S100	Spindle running start
G90 G99 G84 X300. Y-250. Z-150. R-120 P300 F120	Positioning, tap hole 1, then return to point R level
Y-550.；	Positioning, tap hole 2, then return to point R level
Y-750.；	Positioning, tap hole 3, then return to point R level
X1000；	Positioning, tap hole 4, then return to point R level
Y-550.；	Positioning, tap hole 5, then return to point R level
G98 Y-750.；	Positioning, tap hole 6, then return to initial level
G80；	
G28 G91 X0 Y0 Z0 ；	Return to reference point
M5；	Spindle stops
M30；	

Restriction

Cancellation: Do not specify a G code in 01 group(G00, G01, G02, G03) or G60 in a same block, otherwise alarm occurs.

Tool offset: In canned cycle the tool radius compensation is ignored.

4.4.12 Left-handed tapping cycle G74

Format: G74 X_ Y_ Z_ R_ P_ F_

Function: It is used for tapping cycle. In this tapping cycle, when the hole bottom is reached, the spindle rotates reversely.

Explanation:

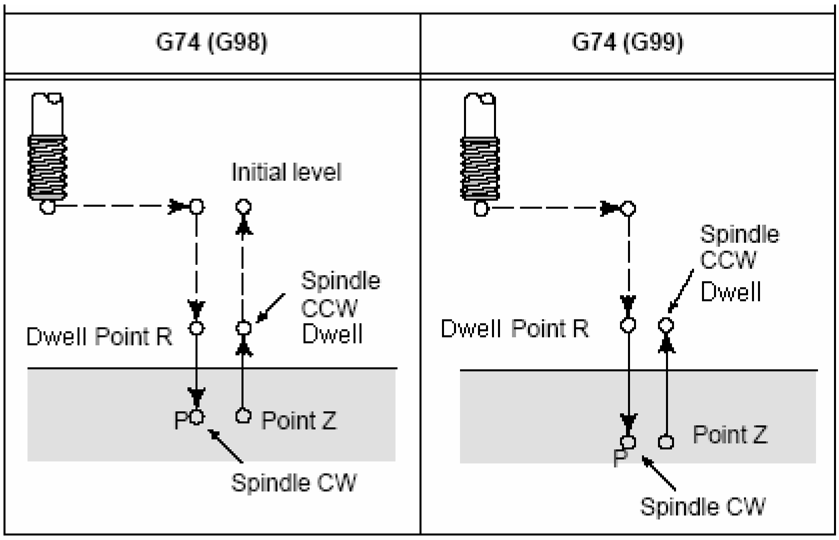
X_Y_： Hole positioning data

Z_： In incremental programming it specifies the distance from point R level to the bottom of the hole; in absolute programming it specifies the absolute coordinate of the hole bottom.

R_： In incremental programming it specifies the distance from the initial level to point R level; in absolute programming it specifies the absolute coordinate of point R.

P_： Dwell time.

F_： Cutting feedrate.



Tapping is performed by rotating the spindle CCW, when the tool reaches the hole bottom, the spindle is rotated reversely for retraction. This operation creates threads.

Feedrate overrides are ignored during tapping. A feed hold does not stop the machine until the retraction operation is finished.

Before specifying G74, use a miscellaneous function (M code) to rotate the spindle. If the spindle CCW rotation is not specified, it will be adjusted for CCW rotation in R level automatically by the current spindle speed specified.

If G74 and M code are specified in a same block, M code is executed while the 1st hole positioning operation is being performed, then the system goes on next drilling operation.

If number of repeats K is specified, M code is only executed for the 1st hole.

P is a modal instruction, and the min. value of it is set by number parameter P281, the max. value by P282. If P value is less than the setting by P281, the min. value is used; if P value is more than the setting by P282, the max. value is used. If P is specified in a block containing no drilling, it can't be stored as a modal datum.

If tool length compensation G43, G44 or G49 is specified in canned cycle, the offset value is either added or cancelled while positioning to point R level.

Example

M4 S100	Spindle running start
G90 G99 G74 X300. Y-250. Z-150. R-120 P300 F120	Positioning, tap hole 1, then return to point R level
Y-550.;	Positioning, tap hole 2, then return to point R level
Y-750.;	Positioning, tap hole 3, then return to point R level
X1000;	Positioning, tap hole 4, then return to point R level
Y-550.;	Positioning, tap hole 5, then return to point R level
G98 Y-750.;	Positioning, tap hole 6, then return to initial level
G80;	
G28 G91 X0 Y0 Z0 ;	Return to reference point
M5;	Spindle stops
M30;	

Restriction

Cancellation: Do not specify a G code in 01 group (G00, G01, G02, G03) or G60 in a same block, otherwise alarm occurs.

Tool offset: In canned cycle the tool radius compensation is ignored.

4.4.13 Fine boring cycle G76

Format: G76 X_Y_Z_Q_R_P_F_K_

Function: It is used for boring a hole precisely. When the tool reaches the hole bottom, the spindle stops and the tool departs from the machined surface of the workpiece and retracts. The retraction trail that affects machined surface finish and the tool damage should be avoided in the operation.

Explanation:

X_Y_: Hole positioning data

Z_: In incremental programming it specifies the distance from point R level to the bottom of the hole; in absolute programming it specifies the absolute coordinate of the hole bottom.

R_: In incremental programming it specifies the distance from the initial level to point

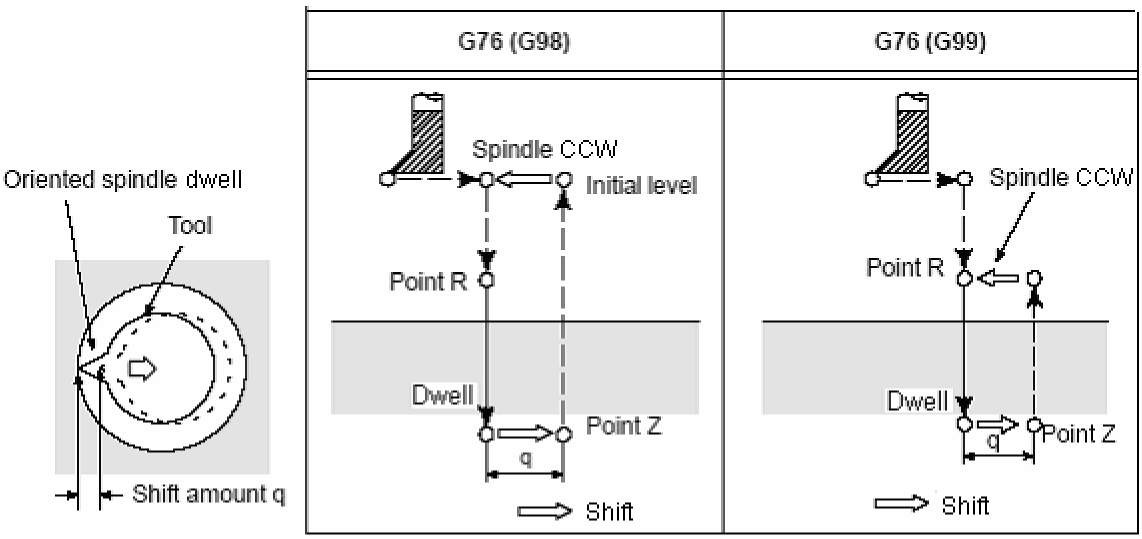
R level; in absolute programming it specifies the absolute coordinate of point R level.

Q_: Offset of the hole bottom

P_: Dwell time.

F_: Cutting feedrate.

K_: Number of fine boring repeats



When the tool reaches the hole bottom, the spindle stops at a fixed rotation position and the tool is moved in the direction opposite to the tool tip and retracted. This ensures that the machined surface is not damaged and enables precise and efficient boring. The parameter Q specifies the retraction distance and the retraction axis and direction are specified by bit parameter NO.42#4 and NO.42#5. And Q is a positive value, if Q is specified with a negative value, the sign is ignored. The hole bottom offset of Q is a modal value saved in canned cycle which should be specified carefully as it is also used for the cutting depth for G73 and G83.

Before specifying G76, use a miscellaneous function (M code) to rotate the spindle.

If G76 and M code are specified in a same block, M code is executed while the 1st hole positioning operation is being performed, then the system goes on next boring operation.

If number of repeats K is specified, M code is only executed for the 1st hole.

If tool length compensation G43,G44 or G49 is specified in canned cycle, the offset value is either added or cancelled while positioning to point R level.

Axis switching: before the boring axis is changed, the canned cycle must be cancelled.

Boring: In a block that does not contain X , Y , Z, R or any additional axes, boring is not performed.

Example

M3 S500 Spindle running start

G90 G99 G76 X300.Y-250. Positioning, bore hole 1, then return to point R level

Z-150. R-100.Q5. Orient at the hole bottom, then shift by 5mm

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

Y-550.;	Positioning, bore hole 2, then return to point R level
Y-750.;	Positioning, bore hole 3, then return to point R level
X1000.;	Positioning, bore hole 4, then return to point R level
Y-550.;	Positioning, bore hole 5, then return to point R level
G98 Y-750.;	Positioning, bore hole 6, then return to initial level
G80 G28 G91 X0 Y0 Z0 ;	Return to reference point
M5;	Spindle stops

Restriction

Cancellation: Do not specify a G code in 01 group (G00, G01, G02, G03) or G60 in a same block , otherwise alarm occurs.

Tool offset: In canned cycle the tool radius compensation is ignored.

4.4.14 Boring cycle G85

Format: G85 X_ Y_ Z_ R_ F_ K_

Function: It is used to bore a hole.

Explanation:

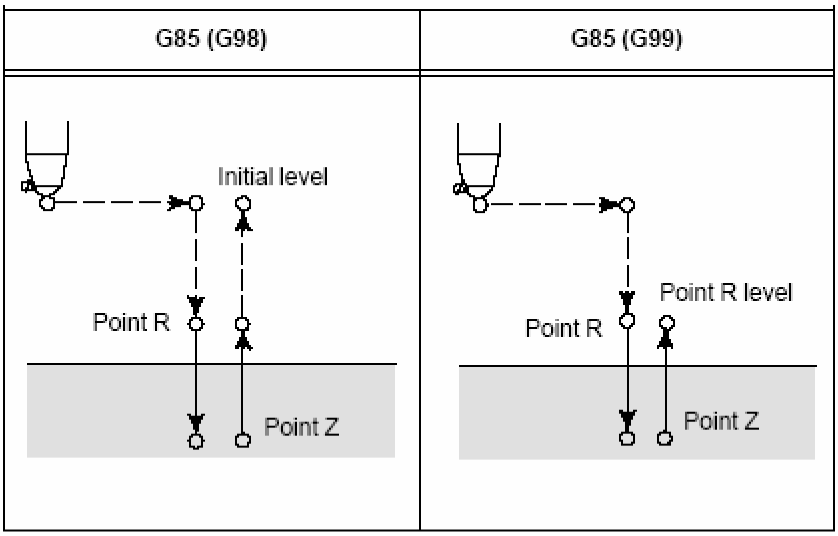
X_Y_： Hole positioning data

Z_： In incremental programming it specifies the distance from point R level to the bottom of the hole; in absolute programming it specifies the absolute coordinate of the hole bottom.

R_： In incremental programming it specifies the distance from the initial level to point R level; in absolute programming it specifies the absolute coordinate of point R.

F_： Cutting feedrate.

K_： Number of repeats



After positioning along X and Y axis, traverse is performed to point R level, and boring is performed from point R level to point Z level. As the tool reaches the hole bottom, cutting feed is performed then return to point R level.

Before specifying G85, use a miscellaneous function (M code) to rotate the spindle.

If G85 and M code are specified in a same block, M code is executed while the 1st hole positioning operation is being performed, then the system goes on next boring operation.

If number of repeats K is specified, M code is only executed for the 1st hole.

If the tool length compensation G43, G44 or G49 is specified in the canned cycle, the offset is added while positioning to point R level.

Axis switching: Before the boring axis is changed, the canned cycle must be cancelled.

Boring: In a block that does not contain X , Y , Z, R or any additional axes, boring is not performed.

Example

M3 S100	Spindle running start
G90 G99 G85 X300. Y-250. Z-150. R-120. F120.	Positioning, bore hole 1, then return to point R level
Y-550.；	Positioning, bore hole 2, then return to point R level
Y-750.；	Positioning, bore hole 3, then return to point R level
X1000.；	Positioning, bore hole 4, then return to point R level
Y-550.；	Positioning, bore hole 5, then return to point R level
G98 Y-750.；	Positioning, bore hole 6, then return to initial level
G80；	
G28 G91 X0 Y0 Z0 ；	Return to reference point
M5；	Spindle stops
M30；	

Restriction

Cancellation: Do not specify a G code in 01 group (G00, G01, G02, G03) or G60 in a same block , otherwise alarm occurs.

Tool offset: In canned cycle the tool radius compensation is ignored.

4.4.15 Boring cycle G86

Format: G86 X_ Y_ Z_ R_ F_ K_；

Function: It is used to perform a boring cycle.

Explanation:

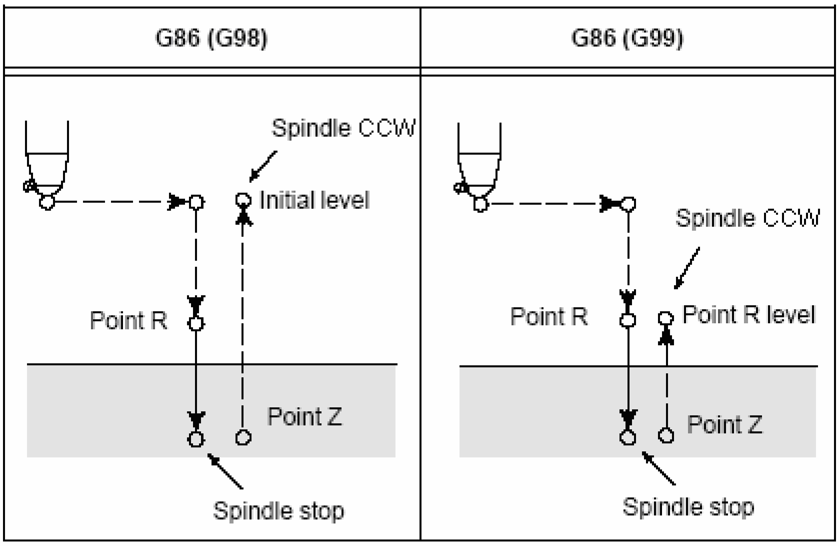
X_Y_： Hole positioning data

Z_： In incremental programming it specifies the distance from point R level to the bottom of the hole; in absolute programming it specifies the absolute coordinate of the hole bottom.

R_: In incremental programming it specifies the distance from the initial level to point R level; in absolute programming it specifies the absolute coordinate of point R.

F_: Cutting feedrate

K_: Number of repeats



After positioning along X and Y axis, the tool rapidly traverses to point R level. And boring is performed from point R level to point Z level. When the tool reaches the hole bottom, it is retracted in traverse.

Before specifying G86, use a miscellaneous function (M code) to rotate the spindle.

If G86 and M code are specified in a same block, M code is executed while the 1st hole positioning operation is being performed, then the system goes on next boring operation.

If number of repeats K is specified, M code is only executed for the 1st hole.

If the tool length compensation G43, G44 or G49 is specified in the canned cycle, the offset value is either added or cancelled while positioning to point R level.

Axis switching: Before the boring axis is changed, the canned cycle must be cancelled.

Boring: In a block that does not contain X, Y, Z, R or any additional axes, boring is not performed.

Example

M3 S2000	Spindle running start
G90 G99 G86 X300. Y-250. Z-150. R-100. F120.	Positioning, bore hole 1, then return to point R level
Y-550.;	Positioning, bore hole 2, then return to point R level
Y-750.;	Positioning, bore hole 3, then return to point R level
X1000.;	Positioning, bore hole 4, then return to point R level

Y-550.; Positioning, bore hole 5, then return to point R level

G98 Y-750.; Positioning, bore hole 6, then return to initial level

G80;

G28 G91 X0 Y0 Z0 ; Return to reference point

M5; Spindle stops

M30;

Restriction

Cancellation: Do not specify a G code in 01 group (G00, G01, G02, G03) or G60,G86 in a same block , otherwise alarm occurs.

Tool offset: In canned cycle the tool radius compensation is ignored.

4.4.16 Boring cycle, back boring cycle G87

Format: G87 X_Y_Z_R_Q_ F_

Function: It is used for accurate boring.

Explanation:

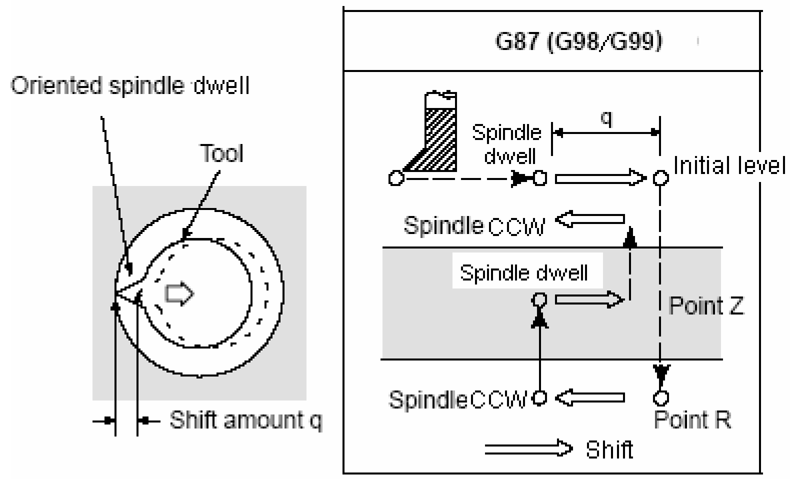
X_Y_: Hole positioning data

Z_: In incremental programming it specifies the distance from point R level to the bottom of the hole; in absolute programming it specifies the absolute coordinate of the hole bottom.

R_: In incremental programming it specifies the distance from the initial level to point R level; in absolute programming it specifies the absolute coordinate of point R.

Q_: Offset of the hole bottom

F_: Cutting feedrate



After positioning along X and Y axis, the tool is stopped after spindle orientation. And the tool is moved in the direction opposite to the tool tip, positioning is performed at the hole bottom point R level. Then the tool is moved in the tool tip direction and the spindle is rotated clockwise. Boring is performed in the positive direction along Z axis until point Z is reached. At point Z, the spindle is stopped at the fixed rotation position after it is oriented again. And the tool is retracted to the initial level in the opposite direction of the tool tip and then is shifted in the direction of the tool tip. And the spindle is rotated clockwise to proceed to the next block operation.

The parameter Q specifies the retraction distance and the retraction direction is set by system parameter NO.42#4 and NO.42#5. Q must be a positive value, if Q is specified with a negative value, the sign is ignored. The hole bottom offset of Q is a modal value saved in canned cycle which should be specified carefully as it is also used for the cutting depth for G73 and G83. Before specifying G87, use a miscellaneous function (M code) to rotate the spindle.

If G87 and M code are specified in a same block, M code is executed while the 1st hole positioning operation is being performed, then the system goes on next boring operation.

If number of repeats K is specified, M code is only executed for the 1st hole.

If the tool length compensation G43, G44 or G49 is specified in the canned cycle, the offset is added while positioning to point R level.

Axis switching: Before the boring axis is changed, the canned cycle must be cancelled.

Boring: In a block that does not contain X, Y, Z, R or any additional axes, boring is not performed.

Annotation: The value of Z and R must be specified in the back boring cycle programming. Alarm occurs if point Z is below point R.

Example

M3 S500 Spindle running start

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

Positioning, bore hole 1, orient at the initial level then shift by 5mm and dwell at point Z for 1s

Y-550.;	Positioning, bore hole 2, then return to point R level
Y-750.;	Positioning, bore hole 3, then return to point R level
X1000.;	Positioning, bore hole 4, then return to point R level
Y-550.;	Positioning, bore hole 5, then return to point R level
G98 Y-750.;	Positioning, bore hole 6, then return to initial level
G80 G28 G91 X0 Y0 Z0 ;	Return to reference point
M5;	Spindle stops

Restriction

Cancellation: Do not specify a G code in 01 group (G00, G01, G02, G03) or G86, G60 in a same block, otherwise alarm occurs.

Tool offset: In canned cycle the tool radius compensation is ignored.

4.4.17 Boring cycle G88

Format: G88 X_Y_Z_R_ P_F_

Function: It is used to bore a hole.

Explanation:

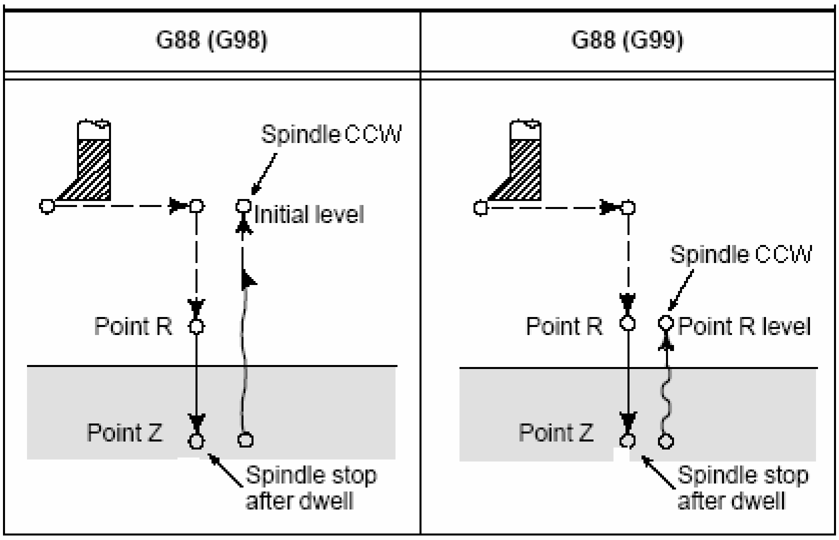
X_Y_: Hole positioning data

Z_： In incremental programming it specifies the distance from point R level to the bottom of the hole; in absolute programming it specifies the absolute coordinate of the hole bottom.

R_: In incremental programming it specifies the distance from the initial level to point R level; in absolute programming it specifies the absolute coordinate of point R.

P_: Dwell time.

F_: Cutting feedrate



After positioning along X and Y axis, the tool rapidly traverses to point R level. Boring is performed from point R level to point Z. When boring is completed, a dwell is performed then the spindle is stopped. The tool is manually retracted from the hole bottom point Z to point R level(in G99) or the initial level(in G98) and the spindle is rotated CCW.

Before specifying G88, use a miscellaneous function (M code) to rotate the spindle.

If G88 and M code are specified in a same block, M code is executed while the 1st hole positioning operation is being performed, then the system goes on next boring operation.

If number of repeats K is specified, M code is only executed for the 1st hole.

If the tool length compensation G43, G44 or G49 is specified in the canned cycle, the offset is added while positioning to point R level.

Axis switching: Before the boring axis is changed, the canned cycle must be cancelled.

Boring: In a block that does not contain X, Y, Z, R or any additional axes, boring is not performed.

Example

M3 S2000 Spindle running start

G90 G99 G88 X300. Y-250. Z-150. R-100. P1000 F120. Positioning, bore hole 1, then return to point R level

Y-550.; Positioning, bore hole 2, then return to point R level

Y-750.; Positioning, bore hole 3, then return to point R level

X1000.; Positioning, bore hole 4, then return to point R level

Y-550.;	Positioning, bore hole 5, then return to point R level
G98 Y-750.;	Positioning, bore hole 6, then return to initial level
G80 G28 G91 X0 Y0 Z0 ;	Return to reference point
M5;	Spindle stops

Restriction

Cancellation: Do not specify a G code in 01 group (G00, G01, G02, G03) or G60, G86 in a same block, otherwise alarm occurs.

Tool offset: In canned cycle the tool radius compensation is ignored.

4.4.18 Boring cycle G89

Format: G89 X_ Y_ Z_ R_ P_ F_ K_

Function: It is used to bore a hole.

Explanation:

X_Y_： Hole positioning data

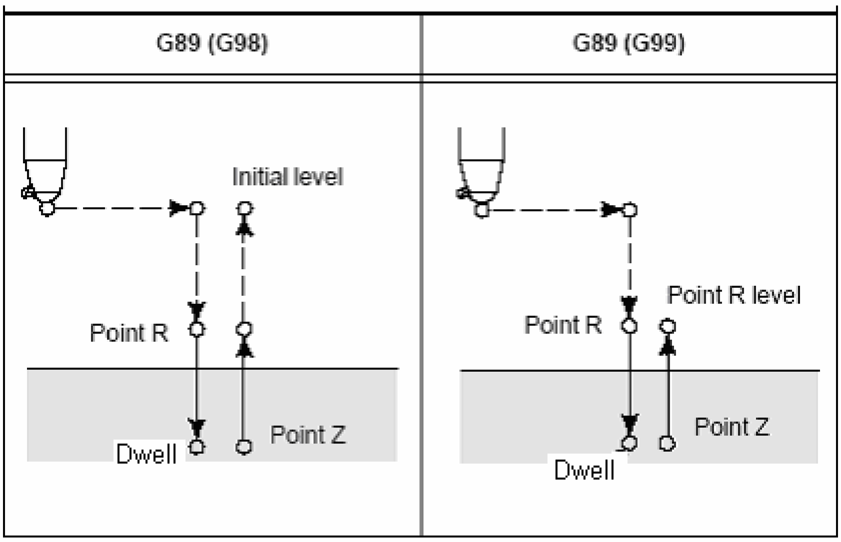
Z_： In incremental programming it specifies the distance from point R level to the bottom of the hole; in absolute programming it specifies the absolute coordinate of the hole bottom.

R_： In incremental programming it specifies the distance from the initial level to point R level; in absolute programming it specifies the absolute coordinate of point R.

P_： Dwell time

F_： Cutting feedrate.

K_： Number of repeats



This cycle is almost the same as G85. The difference is that this cycle perfoms a dwell at the hole bottom.

Before specifying G89, use a miscellaneous function (M code) to rotate the spindle.

If G89 and M code are specified in a same block, M code is executed while the 1st hole positioning operation is being performed, then the system goes on next drilling operation.

If number of repeats K is specified, M code is only executed for the 1st hole.

P is a modal instruction, and the min. value of it is set by number parameter P281, the max. value by P282. If P value is less than the setting by P281, the min. value is used; if P value is more than the setting by P282, the max. value is used. If P is specified in a block containing no drilling, it can't be stored as a modal datum.

If tool length compensation G43, G44 or G49 is specified in canned cycle, the offset value is added while positioning to point R level.

Axis switching: Before the boring axis is changed, the canned cycle must be cancelled.

Boring: In a block that does not contain X, Y, Z, R or any additional axes, boring is not performed.

Example

```
M3 S100                      Spindle running start
G90 G99 G89 X300. Y-250. Z-150. R-120. P1000 F120.
   Positioning, bore hole 1 with 1s dwell at the hole bottom, then return to point R level
Y-550.;                      Positioning, bore hole 2, then return to point R level
Y-750.;                      Positioning, bore hole 3, then return to point R level
X1000.;                      Positioning, bore hole 4, then return to point R level
Y-550.;                      Positioning, bore hole 5, then return to point R level
G98 Y-750.;                  Positioning, bore hole 6, then return to initial level
G80;
G28 G91 X0 Y0 Z0 ;           Return to Reference point
M5;                          Spindle stops
M30;
```

Restriction

Cancellation: Do not specify a G code in 01 group (G00, G01, G02, G03) or G60 in a same block, otherwise alarm occurs.

Tool offset: In canned cycle the tool radius compensation is ignored.

4.4.19 Right-handed rigid tapping G84

Format: G84 X_Y_Z_R_P_F_K_

Function: In rigid tapping, the spindle is controlled by a servo motor that can perform the high-speed and high-precision tapping and it can ensure the tapping initial level without changing point R level. I.e. If a tapping instruction is repeated for many times at the same position, the thread shape will not be damaged.

Explanation:

X_Y_： Hole positioning data

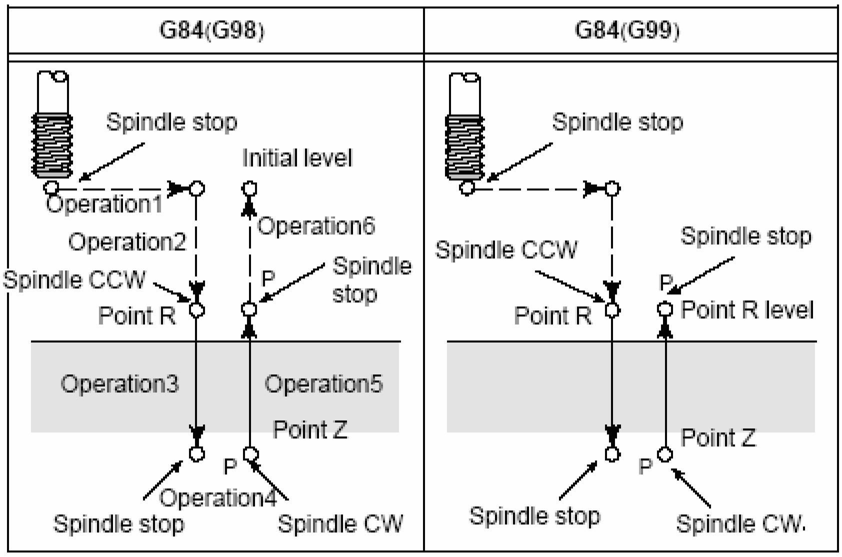
Z_： In incremental programming it specifies the distance from point R level to the bottom of the hole; in absolute programming it specifies the absolute coordinate of the hole bottom.

R_： In incremental programming it specifies the distance from the initial level to point R level; in absolute programming it specifies the absolute coordinate of point R.

P_： Dwell time

F_： Cutting feedrate

K_： Number of repeats



After positioning along X and Y axis, the Z axis rapidly traverses to point R level. The spindle is rotated CCW for tapping from point R level to Z level by G84 instruction. When tapping is finished, the spindle is stopped and a dwell is performed. The spindle is then rotated in the reverse direction, the tool is retracted to point R level, then the spindle is stopped. And traverse to initial level is then performed.

When the tapping is being performed, the feedrate override and the spindle override are assumed to be 100%.

Rigid mode:

Rigid mode can be specified using any of the following methods:

(1) Specify M29 S***** before a tapping instruction

(2) Specify M29 S***** in a block that contains a tapping instruction

If G84 and M code are specified in a same block, M code is executed while the 1st hole positioning operation is being performed, then the system goes on next tapping operation.

If number of repeats K is specified, M code is only executed for the 1st hole.

P is a modal instruction, and the min. value of it is set by number parameter P281, the max. value by P282. If P value is less than the setting by P281, the min. value is used; if P value is more than the setting by P282, the max. value is used. If P is specified in a block containing no drilling, it can't be stored as a modal datum.

If the tool length compensation G43, G44 or G49 is specified in the canned cycle, the offset value is either added or cancelled while positioning to point R level.

Axis switching: Before the tapping axis is changed, the canned cycle must be cancelled. Alarm occurs if the tapping axis is changed in rigid mode.

If S and axis movement instructions are specified between M29 and G84, alarm is issued. If M29 is specified in a tapping cycle, alarm is also issued.

In feed-per-minute mode, the thread lead is obtained from the expression: feedrate/spindle speed.

Feedrate of Z axis=spindle speed×thread lead

Example:

Spindle speed1000r/min

Thread lead1.0mm

then Feedrate of Z axis=1000×1=1000mm/min

G00 X120 Y100； Positioning

M29 S1000 Rigid mode specified

G84 Z-100 R-20 F1000； Rigid tapping

Restriction

F：Alarm is issued if the F value specified exceeds the upper limit of the cutting feedrate.

S：Alarm is issued if the rotation speed exceeds the max. speed of the gear specified which is set by number parameter P294~297.

Cancellation: Do not specify a G code in 01 group (G00, G01, G02, G03) or G60 in a same block, otherwise alarm occurs.

Tool offset: In canned cycle the tool radius compensation is ignored.

Program restart: It is ineffective during the rigid tapping.

4.4.20 Left-handed rigid tapping G74

Format: G74 X_Y_Z_R_P_F_K_

Function: In rigid tapping the spindle is controlled by a servo motor. This instruction can be used for left-hand high-speed and high-precision tapping.

Explanation:

X_Y_： Hole positioning data

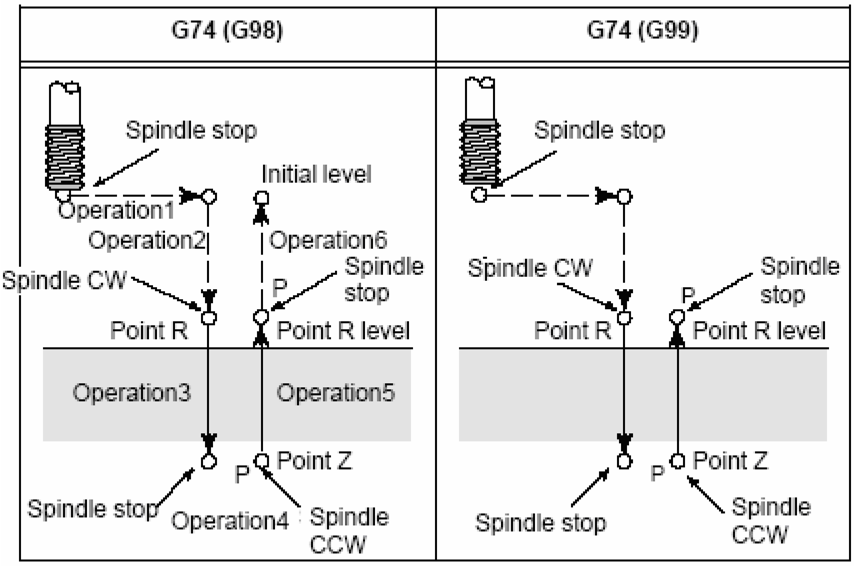
Z_： In incremental programming it specifies the distance from point R level to the bottom of the hole; in absolute programming it specifies the absolute coordinate of the hole bottom.

R_： In incremental programming it specifies the distance from the initial level to point R level; in absolute programming it specifies the absolute coordinate of point R.

P_： Dwell time

F_： Cutting feedrate.

K_： Number of repeats



After positioning along X and Y axis, traverse is performed by Z axis to point R level. The spindle is rotated CW for tapping from point R level to Z level by G74 instruction. When tapping is finished, the spindle is stopped and a dwell is performed. The spindle is then rotated in the reverse direction to retract to point R level and stops. And traverse to initial level is then performed. When the tapping is being performed, the feedrate override and the spindle override are assumed to be 100%.

Rigid mode:

Rigid mode can be specified using any of the following methods:

(1) Specify M29 S***** before a tapping instruction

(2) Specify M29 S***** in a block that contains a tapping instruction

If G74 and M code are specified in a same block, M code is executed while the 1st hole positioning operation is being performed, then the system goes on next tapping operation.

If number of repeats K is specified, M code is only executed for the 1st hole.

P is a modal instruction, and the min. value of it is set by number parameter P281, the max. value by P282. If P value is less than the setting by P281, the min. value is used; if P value is more than the setting by P282, the max. value is used. If P is specified in a block containing no drilling, it can't be stored as a modal datum.

If the tool length compensation G43, G44 or G49 is specified in the canned cycle, the offset value is either added or cancelled while positioning to point R level.

Axis switching: Before the tapping axis is changed, the canned cycle must be cancelled. Alarm occurs if the tapping axis is changed in rigid mode.

If S and axis movement instructions are specified between M29 and G74, alarm is issued. If M29 is specified in a tapping cycle, alarm is also issued.

The thread lead is obtained from the expression: feedrate/spindle speed.

Feedrate of Z axis=spindle speed×thread lead

Example:

Spindle speed1000r/min

Thread lead1.0mm

Then Feedrate of Z axis=1000×1=1000mm/min

G00 X120 Y100；	Positioning
M29 S1000	Rigid mode specified
G74 Z-100 R-20 F1000；	Rigid tapping

Restriction

F： Alarm is issued if the F value specified exceeds the upper limit of the cutting feedrate.

S： Alarm is issued if the rotation speed exceeds the max. speed of the gear used which is set by number parameter P294~297.

Cancellation: Do not specify a G code in 01 group (G00, G01, G02, G03) or G60 in a same block, otherwise alarm occurs.

Tool offset: Before canned cycle the tool radius compensation is cancelled automatically, while it is set up automatically after the canned cycle.

Program restart: It is ineffective during the rigid tapping.

4.4.21 Canned cycle cancel G80

Format: G80

Function: It is used to cancel the canned cycle.

Explanation:

All canned cycles are cancelled for normal operation. Point R and point Z are cancelled too. Other drilling and boring data are also cancelled.

Example:

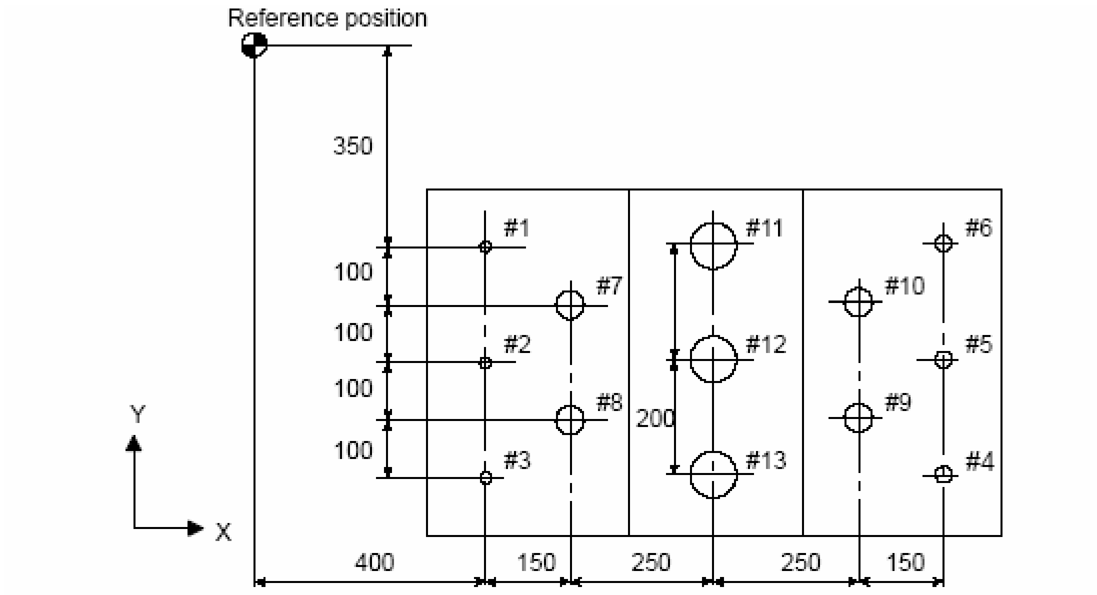
M3 S100	Spindle running start

G90 G99 G88 X300. Y-250. Z-150. R-120. F120.

	Positioning, bore hole 1, then return to point R
Y-550.;	Positioning, bore hole 2, then return to point R
Y-750.;	Positioning, bore hole 3, then return to point R
X1000.;	Positioning, bore hole 4, then return to point R
Y-550.;	Positioning, bore hole 5, then return to point R
G98 Y-750.;	Positioning, bore hole 6, then return to initial level
G80;	
G28 G91 X0 Y0 Z0 ;	Return to Reference point and cancel canned cycle
M5;	Spindle stops

Example:

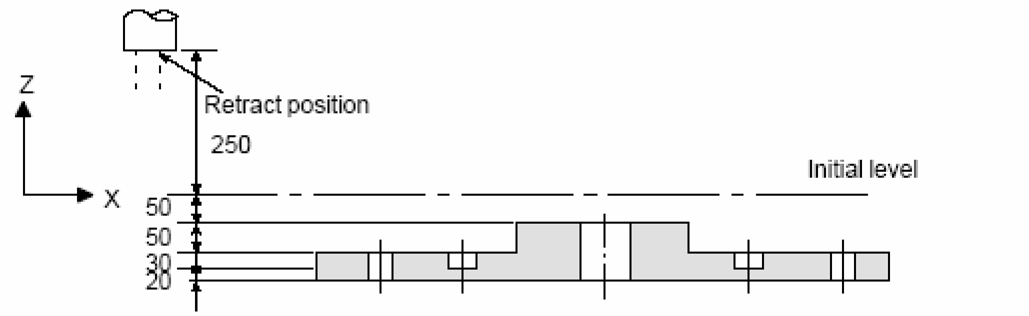
Usage of canned cycle using tool length compensation

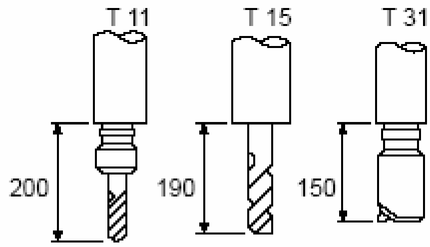


 # 1～ 6... drilling of a Φ10 hole

 # 7～10... drilling of a Φ20 hole

#11～13.. boring of a Φ95 hole





Value 200 is set in offset No.11, 190 is set in offset No.15, 150 is set in offset No.31. The program is as following:

N001 G92 X0 Y0 Z0 ；	Coordinate setting at reference point
N002 G90 G00 Z250 T11 M6 ；	Tool change
N003 G43 Z0 H11 ；	Tool length compensation at initial level
N004 S300 M3 ；	Spindle start
N005 G99 G81 X400 Y-350 ； Z-153 R-97 F120 ；	Positioning, then #1 drilling
N006 Y-550 ；	Positioning, then #2 drilling and point R level return
N007 G98 Y-750 ；	Positioning, then #3 drilling and initial level return
N008 G99 X1200 ；	Positioning, then #4 drilling and point R level return
N009 Y-550 ；	Positioning, then #5 drilling and point R level return
N010 G98 Y-350 ；	Positioning, then #6 drilling and initial level return
N011 G00 X0 Y0 M5 ；	Reference point return, spindle stop
N012 G49 Z250 T15 M6 ；	Tool length compensation cancel, tool change
N013 G43 Z0 H15 ；	Initial level, Tool length compensation
N014 S200 M3 ；	Spindle start
N015 G99 G82 X550 Y-450 ； Z-130 R-97 P30 F70 ；	Positioning, then #7 drilling and point R level return
N016 G98 Y-650 ；	Positioning, then #8 drilling and initial level return
N017 G99 X1050 ；	Positioning, then #9 drilling and point R level return
N018 G98 Y-450 ；	Positioning, then #10 drilling and initial level return
N019 G00 X0 Y0 M5 ；	Reference point return, spindle stop
N020 G49 Z250 T31 M6 ；	Tool length compensation cancel, tool change
N021 G43 Z0 H31 ；	Initial level, Tool length compensation
N022 S100 M3 ；	Spindle start
N023 G85 G99 X800 Y-350 ； Z-153 R47 F50 ；	Positioning, then #11 drilling and point R level return
N024 G91 Y-200 ； Y-200 ；	Positioning, then #12, 13 drilling and point R level return

N025 G00 G90 X0 Y0 M5 ；	Reference point return, spindle stop
N026 G49 Z0 ；	Tool length compensation cancel
N027 M30 ；	Program stop

4.5 Tool compensation G code

4.5.1 Tool length compensation G43, G44, G49

Function:

G43 specifies the positive compensation for tool length.

G44 specifies the negative compensation for tool length.

G49 is used to cancel tool length compensation.

Format:

There are 2 modes A/B for tool length offset which are set by bit parameter No. 39.0 in this system.

Mode A:

$$\left.\begin{matrix}\text{G43}\\\text{G44}\end{matrix}\right\}\text{Z\_ H\_ ;}$$

Mode B:

G17 G43 Z_H;

G17 G44 Z_H;

G18 G43 Y_H;

G18 G44 Y_H;

G19 G43 X_H;

G19 G44 X_H;

Tool length offset mode cancel：G49; or H0;

Explanation:

The instruction above is used to shift an offset value for the end point of specified axis. Due to the difference of the tool length value assumed (usually the 1st tool) and the actual tool length in machining saved in the offset memory, the tool of different lengths can be used for machining only by changing the tool length offset value, but not changing the program.

G43，G44 specify the different offset direction and H code specifies the offset number. For the tool length compensation the effectiveness of the offset value by H code respecified or in next block is set by bit parameter No.39.6.

1 Offset direction

G43： Positive offset (frequently -used)

G44： Negative offset

Either for absolute instruction or incremental instruction, when G43 is specified, the offset value (stored in offset memory) specified with the H code is added to the coordinate of the specified axis moving end point in the program. When G44 is specified, the offset value specified by H code is subtracted from the coordinate of the end position, and the resulting value obtained is taken as the final coordinate of the end position.

G43，G44 are modal G code, which are effective till another G code belonging to the same group is used.

2 Specification of offset value

The length offset number is specified by H code, and the new moving instruction value of Z axis is obtained by plusing or subtracting the value of the offset number from the moving instruction value of Z axis. The offset number can be specified by H00～H128 as required.

The value of the offset number can be stored into the offset memory in advance by LCD/MDI panel.

The range of the offset value is as follows:

	mm input
Offset value H	-999.999～+999.999ｍｍ

The offset value corresponding to offset No.00 (H00) is 0. It can't be set in the system.

The tool length compensation is ineffective before Z instruction.

Note While the offset value is changed due to the offset number changing, the old offset value is replaced by the new one, not the adding of the new offset value and the old one.

For example:

H01.......................... offset value 20

H02.......................... offset value 30

G90 G43 Z100 H01 ； .... Z to 120

G90 G43 Z100 H02 ； .... Z to 130

3 Sequence of the offset value

Once the length offset mode is set up, the current offset number takes effect at once; if the offset number is changed, the old offset value will be immediately replaced by the new one. For example:

O×××××；

H01；

G43 Z10； (1) Offset number H01 takes effect

G44 Z20 H02； (2) Offset number H02 takes effect

H03； (3) Offset number H03 takes effect

G49； (4) Offset cancel，H00 takes effect

M30；

4 Tool length compensation cancel

Specify G49 or H00 to cancel tool length compensation. And the tool length compensation is cancelled immediately after they are specified.

Note After B mode of tool length offset is executed along two or more axes, all the axes offset can be cancelled by G49, while only the axis offset perpendicular to a specified plane can be cancelled by H0.

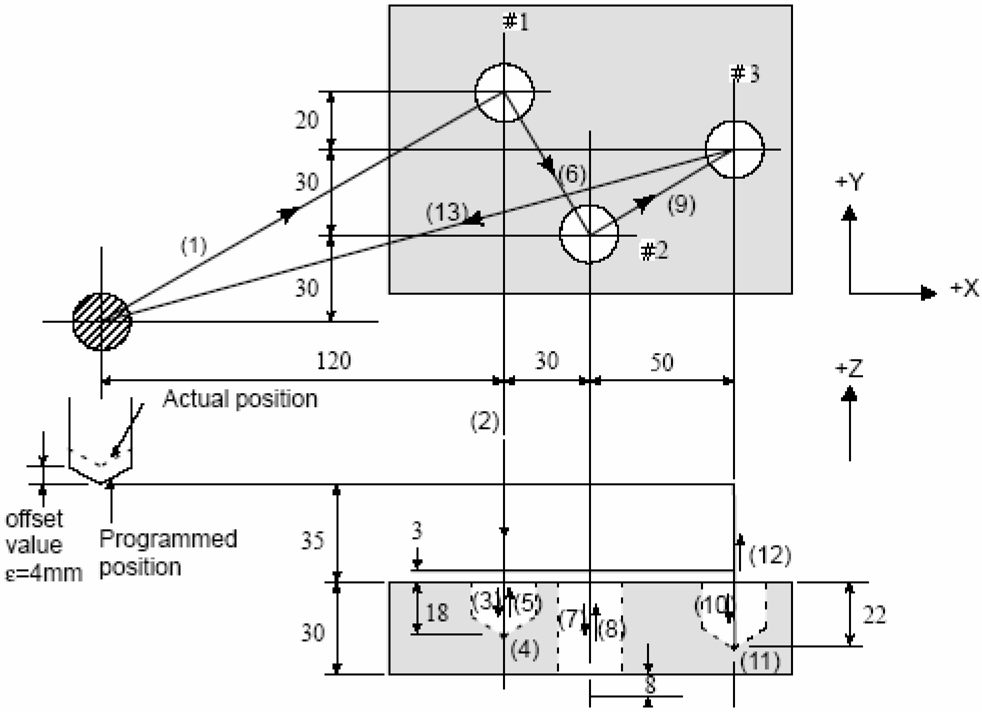
5 G53, G28 or G30 in tool length offset mode

While G53, G28 or G30 is specified in the tool length offset mode, the offset vector of the tool length offset axis is cancelled after it moves to a specified position (G53 cancelled at the specified position; G28, G30 cancelled at the intermediate point), but the modal code is not switched to G49 and the axes except the tool length offset axis are not cancelled. If G53 and G49 are in the same block, all the axis length offsets are cancelled after the axis moves to the specified position; if G28 or G30 is in the same block with G49, all the axes cancel the length offset after they move to the intermediate point. In tool length offset, the offset vector cancelled by G53, G28 or G30 will be restored in the next block in the buffer.

6 Example for tool length compensation

(A) Tool length compensation (in boring hole # 1， #2， #3)

(B) H01= offset value - 4



N1 G91 G00 X120 Y80 ; ……… (1)

N2 G43 Z-32 H01 ; ………………… (2)

N3 G01 Z-21 F200 ; ………………… …(3)

N4 G04 P2000 ; …………………… . (4)

N5 G00 Z21 ; …………………… ……… (5)

N6 X30 Y-50 ; …………… …………… (6)

N7 G01 Z-41 F200 ; ………… ……… (7)

N8 G00 Z41 ; …………………… ……… (8)

N9 X50 Y30 ; …………………… ….. (9)

N10 G01 Z-25 F100 ; ………… ……… (10)

N11 G04 P2000 ; …………………… ….. (11)

N12 G00 Z57 H00 ; ……………… … …(12)

N13 X-200 Y-60 ; ………….. … . ……. (13)

N14 M30 ;

4.5.2 Tool radius compensation G40/G41/G42

Format:

$$\begin{cases} \text{G41 D\_X\_Y\_} \\ \text{G42 D\_ X\_Y\_} \end{cases}$$

G40 X_Y_

Function:

G41 specifies the left offset of the tool moving.

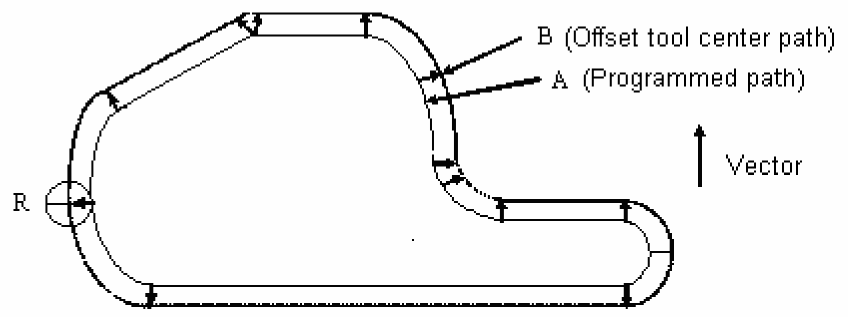
G42 specifies the right offset of the tool moving.

G40 specifies the tool radius compensation cancel.

Explanation:

1 Tool radius compensation

As following figure, to cut workpiece A using the tool with the radius R, the tool center path is shown as B, the distance from B to A is R, the distance that the tool deviates from the workpiece A is called compensation.



Offset and vector

The tool radius compensation is programmed for machining program by programmer. During the machining, the tool diameter is measured and input into the CNC memory. And the tool path turns into a offset path B.

2 Offset value (D value)

The radius offset number is specified by D code, and the new moving instruction value is obtained by the value of the offset number plusing or subtracting the moving value of the program. The offset number can be specified by D00～D127 as required. The diameter or radius value of it can be set by bit parameter No.40.7.

The offset value of the offset number can be saved into the offset memory in advance by LCD/MDI panel. For the tool radius compensation the effectiveness of the offset value by D code respecified or in next block is set by bit parameter No.39.4.

The range of the offset value is as follows:

	mm input
Offset value D	-999.999～+999.999ｍｍ

Note The default offset value of D00 is 0 that can't be set or modified by user.

3 Plane selection and vector

Compensation calculation is carried out in the plane determined by G17，G18，G19. This plane is called the compensation plane. For example, if XY plane is selected, the compensation and vector calculation are carried out by (X，Y) in program. The coordinates of the axis not in compensation plane are not affected by compensation.

In simultaneous 3 axes control, only the tool path projected on the compensation plane is compensated.

The change of the compensation plane can only be performed after the compensation is cancelled.

G code	Compensation plane
G17	X - Y plane
G18	Z - X plane
G19	Y - Z plane

4 G40, G41 and G42

The cancellation and execution of the tool radius compensation vector are specified by G40，G41，G42. They are used to define a mode to determine the value and the direction of the offset vector by combination with G00，G01，G02，G03.

G code	**Function**
G40	Tool radius compensation cancel
G41	Tool radius offset left
G42	Tool radius compensation right

Tool radius compensation cancel (G40)

Use the following instruction to perform the linear motion from the old vector of the start point to the end point in G00, G01 mode:

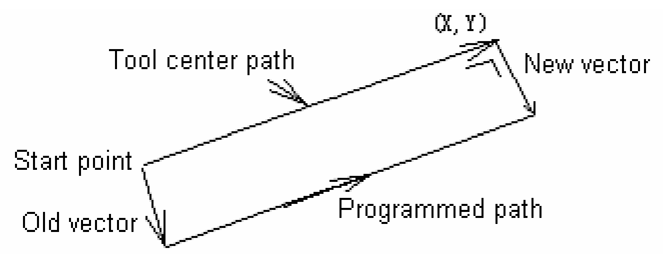
It performs linear movement from the old vector of start point to the end point. In G00 mode, the axes rapidly traverse to the end point. By using this instruction, the system enters into tool radius compensation cancel mode from tool radius compensation mode

If G40 is specified without X__ Y__, no operation is performed by the tool.

Tool radius compensation left (G41)

1 In G00, G01 mode

G41 X__ Y__ D__ ；It specifies a new vector being vertical to the direction of(X，Y) at the block end point. The tool is moved from the tip of the old vector to the tip of the new one at the start point.



When the old vector is zero, by this instuction the tool is switched to tool radius compensation mode from tool offset cancel mode. And the offset value is specified by D code.

2 In G02, G03 mode

G41……；

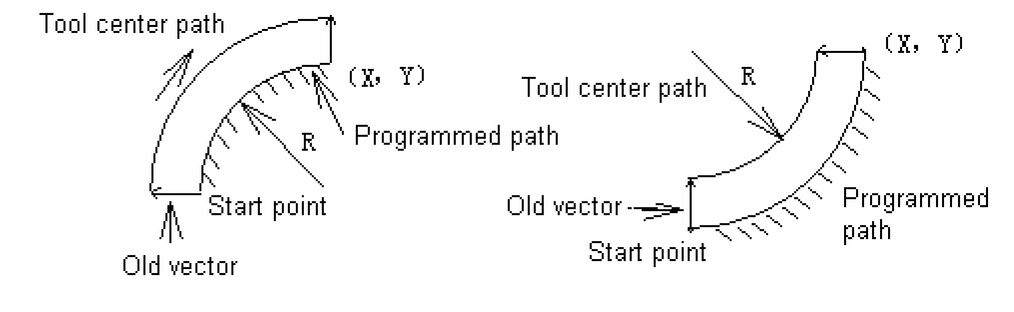
……

……

G02 /G03 X__ Y__ R__ ；

By program above, the new vector that is located on the line between the circle center and the end point can be made out. From the arc advancing direction, it points to the left (right). The tool center moves along an arc from the old vector tip to the new vector tip with the precondition that the old vector is has been made out.

The offset vector points to or is apart from the circle center from the start point or the end point.



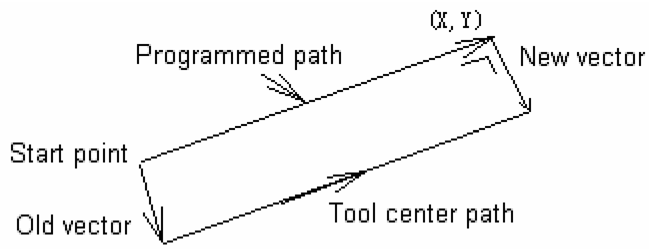
Tool radius compensation right (G42)

By contrast to G41, G42 specifies the tool to deviate at the right side of the workpiece along the tool advancing direction. I.e. the vector direction got in G42 is reverse to the vector direction got in G41. Besides the direction, the deviation of G42 is identical with that of G41.

1 In G00, G01 mode

G42 X__ Y__ D__ ；

G42 X__ Y__ ；



2 In G02, G03 mode

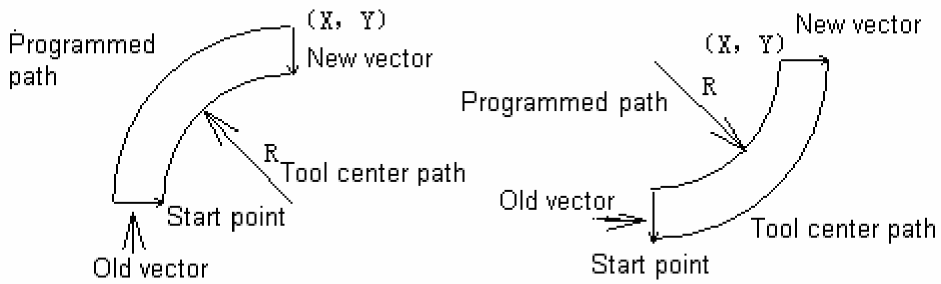


Fig. 4-5-2 (A)

6 Precautions on offset

(A) Specification of offset number

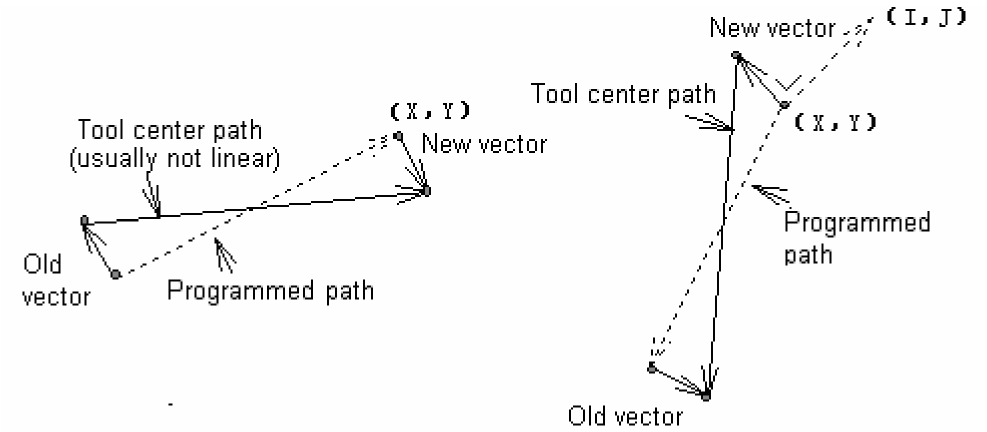
G41，G42 and G40 are modal instructions. The offset number is specified by D code they can be specified at any place from the offset cancel mode to tool radius compensation mode. Alarm is issued if G41, G42 instructions are not followed by moving instructions.

(B) From the offset cancel mode to tool radius compensation mode

The moving instruction must be positioning (G00) or linear interpolation (G01) when the mode is switched from the offset cancel to tool radius compensation. And the circular interpolation(G02，G03) is impermitted.

(C) Switching of tool radius compensation

The offset direction is usually changed from the left to the right or vice versus via offset cancel mode. But the positioning (G00) or linear interpolation (G01) can be changed directly not via offset cancel mode, and the tool path is as follows:



G1G41 D__X__ Y__；

……

G1G42 D__X__ Y__；

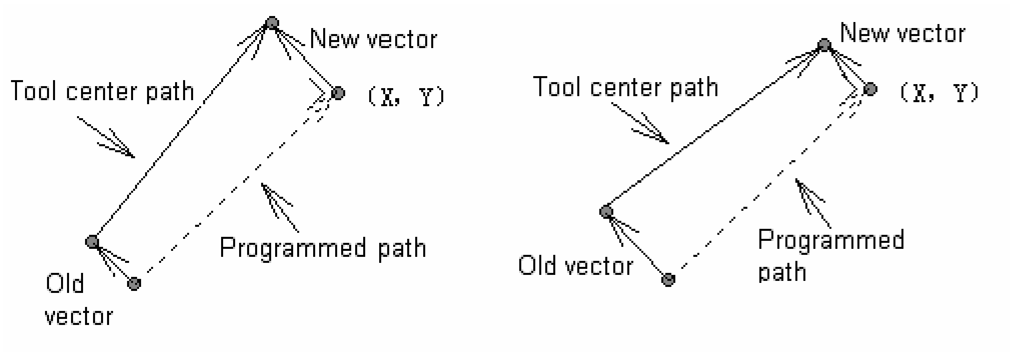
G42 D__X__ Y__；

……

G41 D__ X__ Y__；

(D) The change of offset value

The change of offset value is usually performed at the tool change in offset cancel mode, but for the positioning (G00) or linear interpolation (G01) it can also be performed in offset mode. It is shown as follows:



The change of offset value

(E) The positive and negative offset value and the tool center path

If the offset value is set for negative, it is equivalent to change the G41 and G42 in program that the outer cutting for workpiece turns into inner cutting, and inner cutting for outer cutting.

In the following programming figure, the offset value is assumed for positive:

When a tool path is programmed as(A), and the offset value is set for negative, the tool center moves as in (B); if a tool path is programmed as(B), and the offset value is set for negative, the tool center moves as in (A).

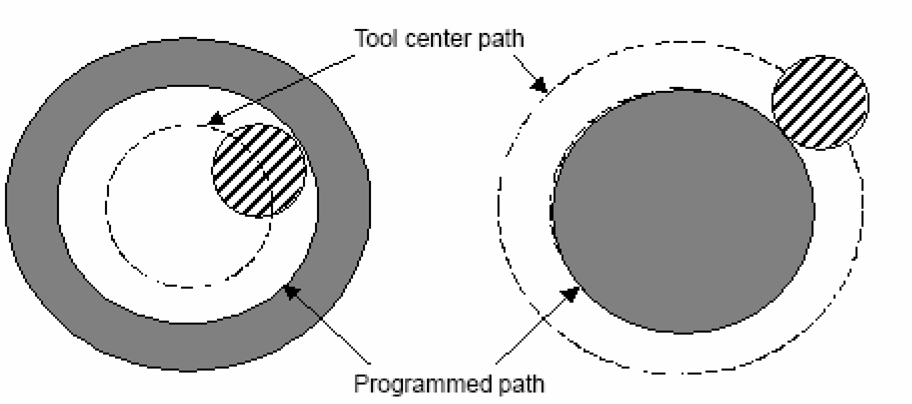


Fig.(A) Fig. (B)

The figure with acute angles is often used (with sharp-angle arc interpolation figure). If the offset value is set for negative, the inner side of the workpiece can't be cut. When cutting the inner sharp angle in a point, interpolate an arc with a proper radius at the point for smooth cutting transition.

The compensation for left or right is judged by the compensation direction (workpiece unmoved) to the direction of the tool movement relative to the workpiece. By G41or G42, the system enters compensation mode, and by G40 the compensation mode is cancelled.

The example for compensation program is as following:

The block 1, in which the compensation cancel mode is changed for compensation mode by G41 instruction, is called start. At the block end, the tool center is compensated by the tool radius that is vertical to the next block (from P1 to P2). The offset value is specified by D07, i.e. the offset number is set for 7. and G41 specifies the tool path compensation left.

During the offset, the workpiece figure is programmed as P1→P2……P9→P10→P11, and the tool path compensation is performed automatically.

Program example for the tool path compensation

G92 X0 Y0 Z0；

(1) N1 G90 G17 G0 G41 D7 X250 Y550 ； (The offset value must be preset by the offset number.)

(2) N2 G1 Y900 F150 ；

(3) N3 X450 ；

(4) N4 G3 X500 Y1150 R650 ；

(5) N5 G2 X900 R-250 ；

(6) N6 G3 X950 Y900 R650 ；

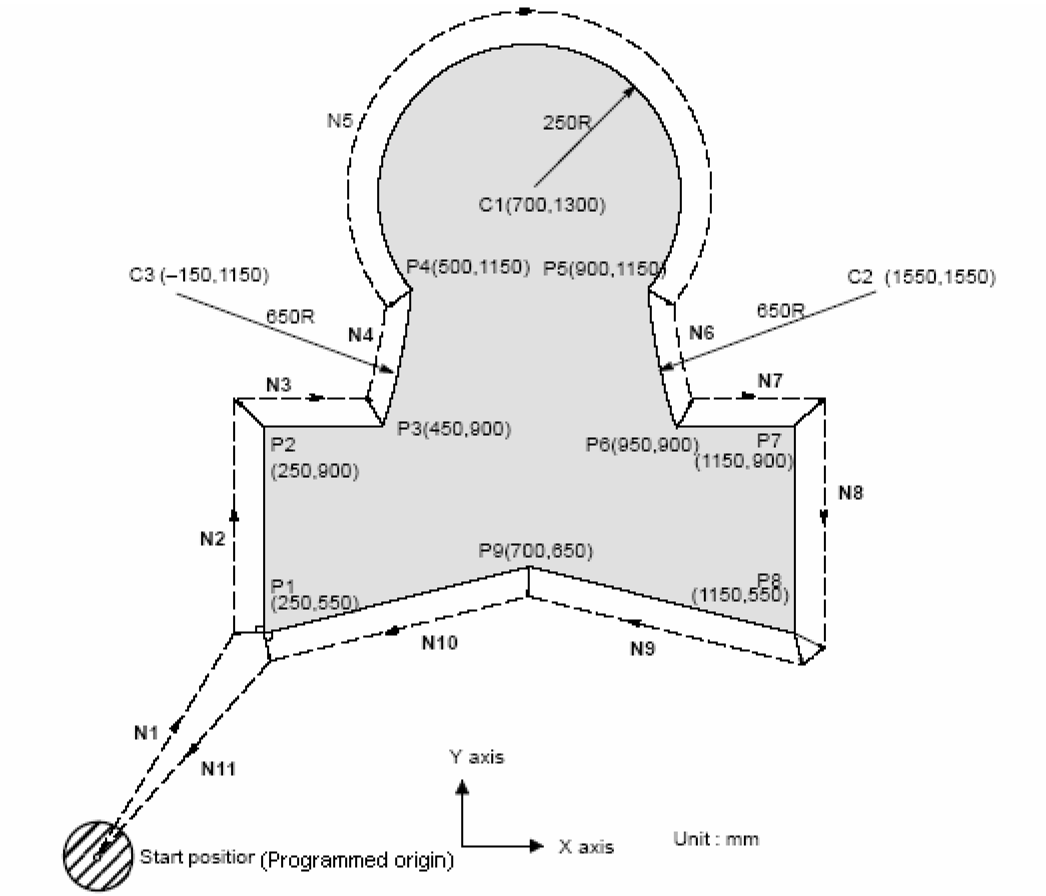
(7) N7 G1 X1150 ；

(8) N8 Y550 ；

(9) N9 X700 Y650 ；

(10) N10 X250 Y550 ；

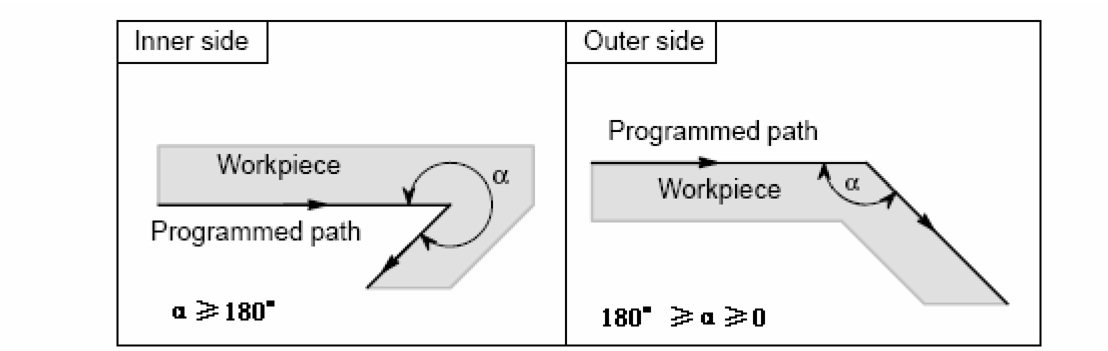
(11) N11 G0 G40 X0 Y0 ；



4.5.3 Explanation of tool radius compensation

Conception:

Inner side and outer side: when an angle of intersection created by tool paths specified with move instructions for two blocks is over 180°, it is called inner side, when the angle is between 0° and 180°, it is called outer side.



Meanings of symbols:

The following symbols are used in following figures:

——S indicates a position at which a single block is executed once.

—— SS indicates a position at which a single block is executed twice.

——SSS indicates a position at which a single block is executed three times

——L indicates that the tool moves along a straight line.

——C indicates that the tool moves along an arc.

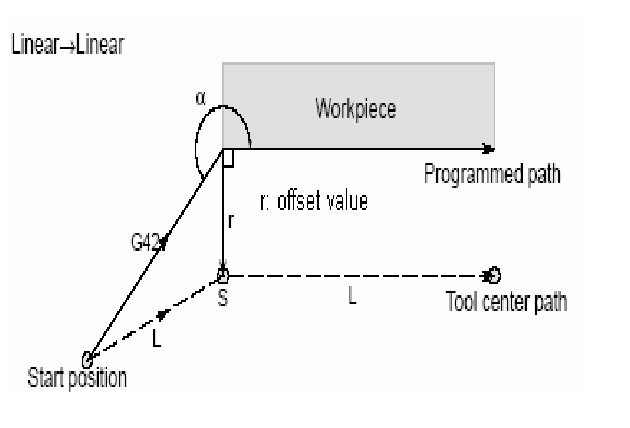
——r indicates the tool radius compensation value.

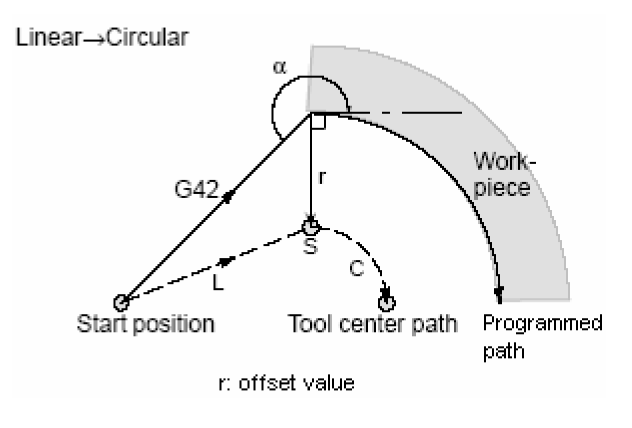
——An intersection is a position at which the programmed paths of two blocks intersect with each other after they are shifted by r

——O indicates the center of the tool

1. Tool movement in start-up When the offset cancel mode is changed to offset mode, the tool moves as illustrated below(start-up):

（a）Tool movement around an inner side of a corner（α≥180°）

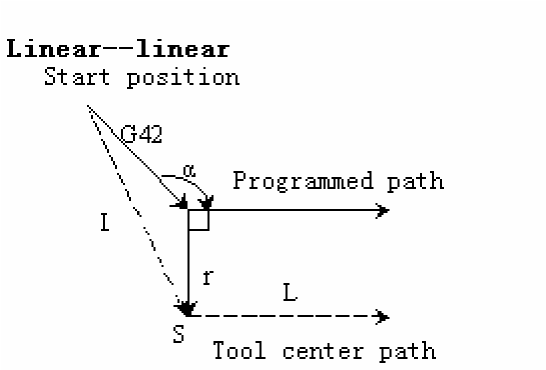


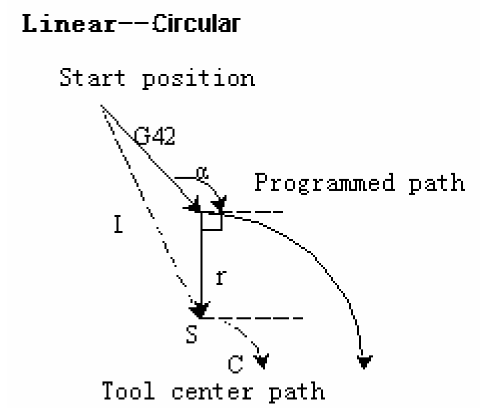


(b) Tool movement around an outer side of a corner at an obtuse angle（180°＞α≥90°）

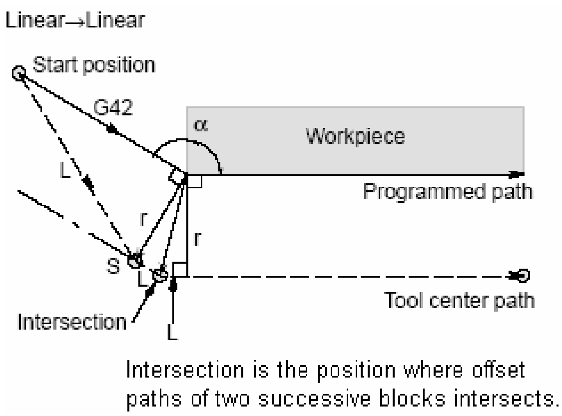
：There are 2 tool path types at offset start or cancel: A and B,which is set by bit parameter No.40.0.

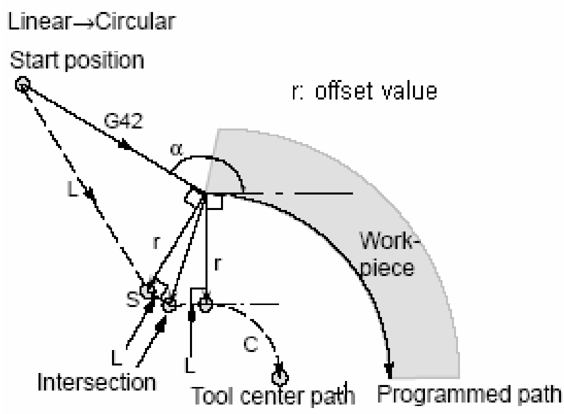
A





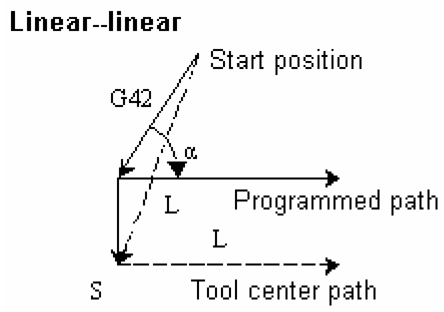
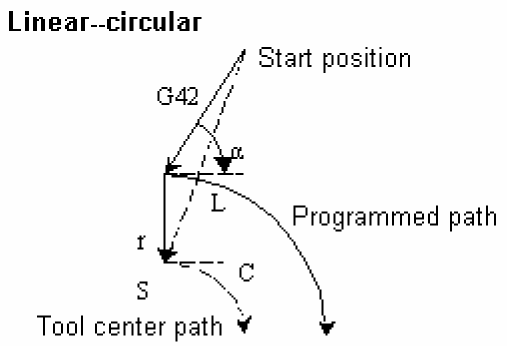
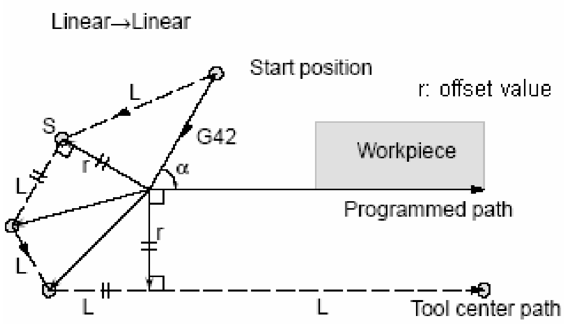
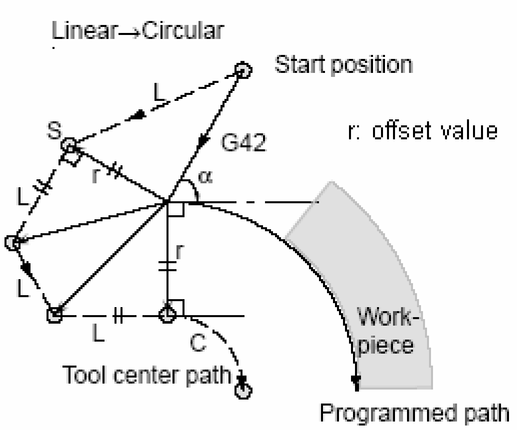
B



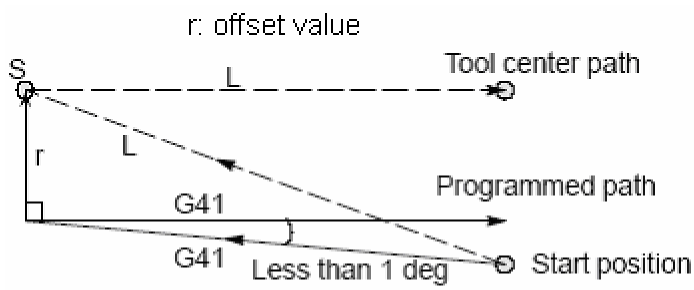


（C）Tool movement around an outer side of a corner at an acute angle（α＜90°）

There are 2 tool path types at offset start or cancel: A and B, which is set by bit parameter No.40.0.

A		
B		

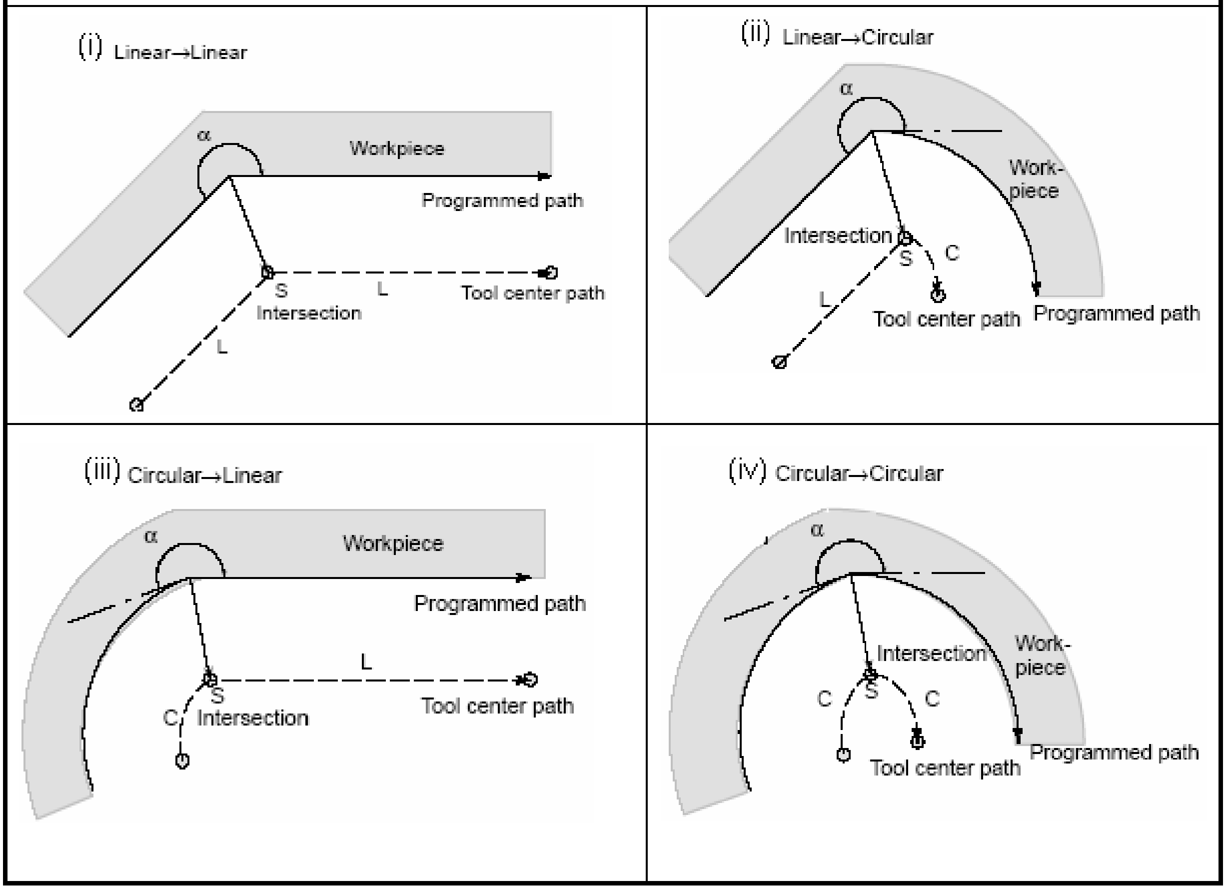
（d） Tool movement around an outer side of a corner at an acute angle less than 1°(α＜1°) linear→linear



2. Tool movement in offset mode

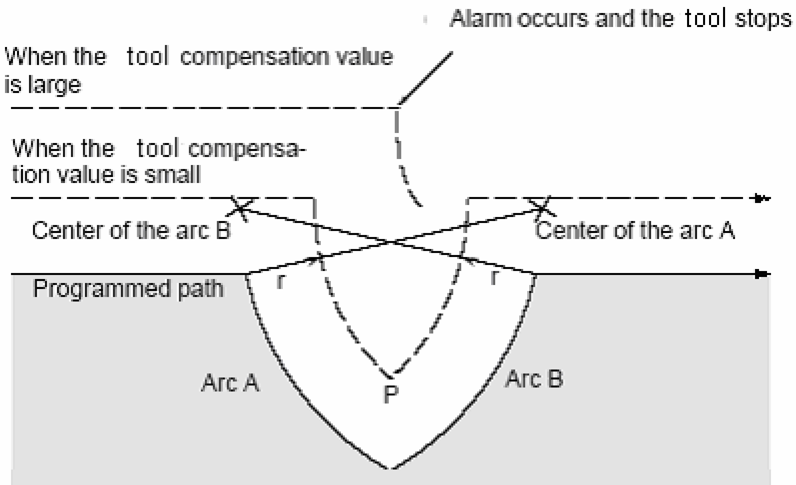
Alarm occurs and tool stops if the offset plane is changed during the offset. The tool movement in offset mode is as following figures:

（a）Tool movement around an inner side of a corner（α≥180°）



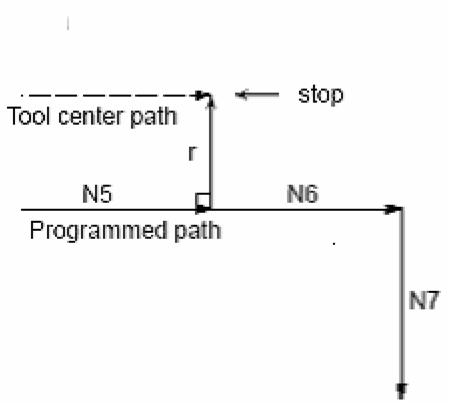
3. Special condition:

(i) No intersection



For the figure above, if the tool radius is small, the offset paths of the arcs has an intersection. If the radius gets larger, the intersection may not exist. The tool stops at the end of the last block and alarm is issued.

(ii) Arc center coincides with the start point or end point



(G41)
N5 G01 X100.0 ;
N6 G02 X100.0 I0 J0 ;
N7 G03 Y–100.0 J–100.0 ;

As the figure shows, alarm is issued and the control stops at the end of last block.

4. Tool movement in offset cancel mode

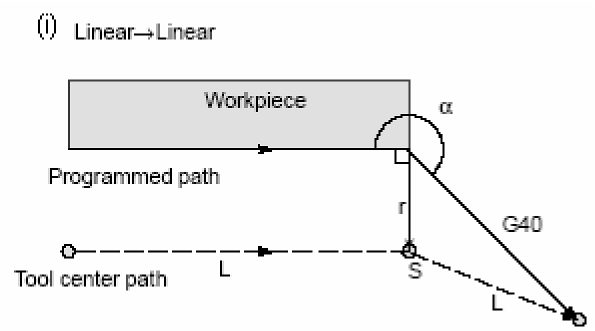
In offset mode, when the block complies to any of the following condition is executed, the system enters offset cancell mode. The operation of this block is called offset cancel.

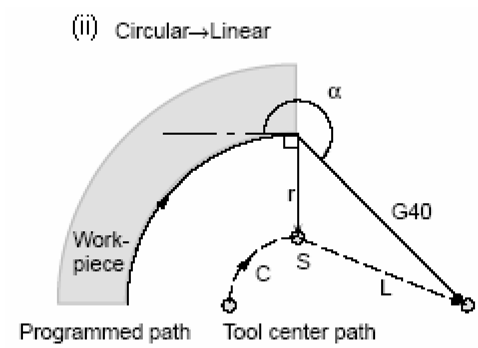
a) Instruction G40

b) When the tool radius compensation number is 0:

Arc instruction （G03 or G02）is unallowed in offset cancel mode. Alarm is issued and tool stops if arc is specified

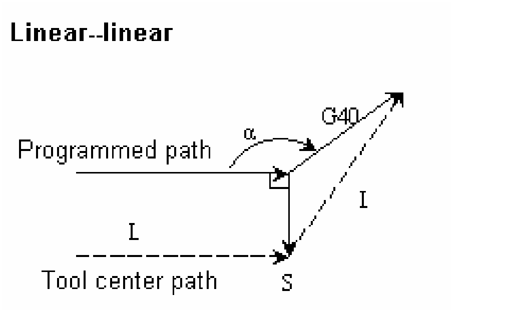
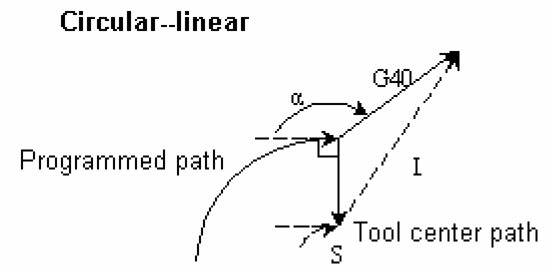
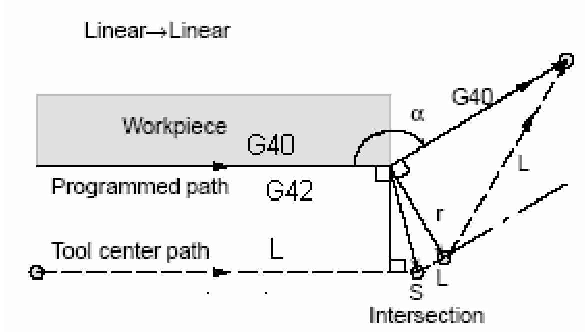
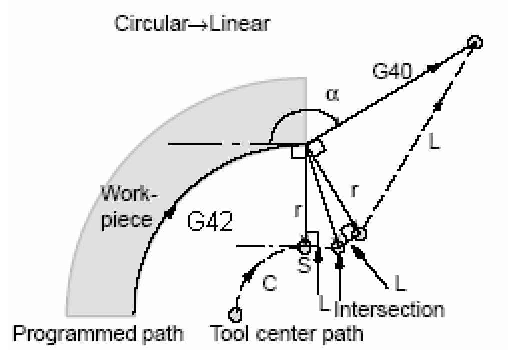
（a）Tool movement around an inner side of a corner（α≥180°）





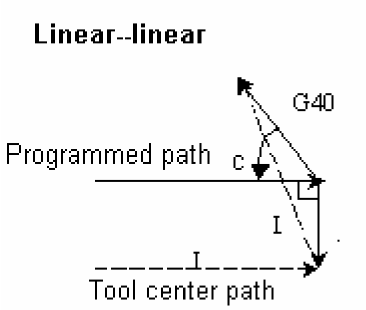
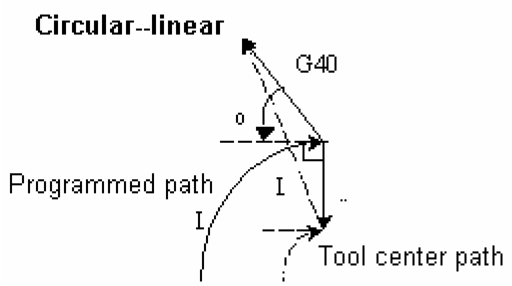
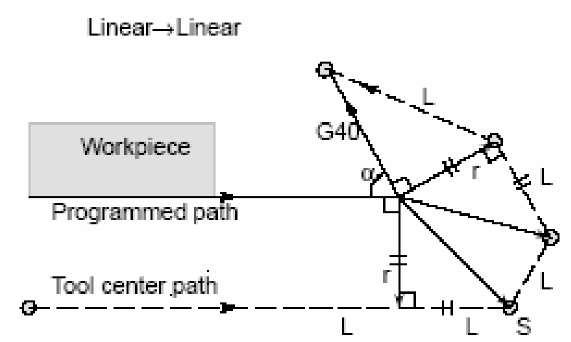
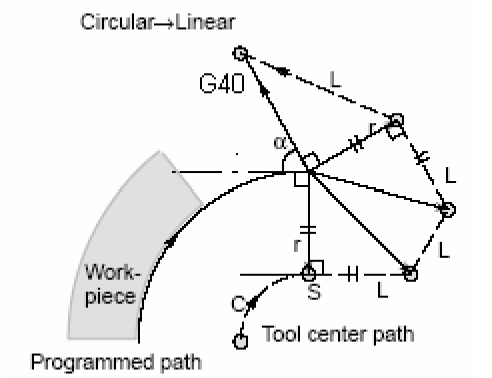
(b) Tool movement around an inner side of a corner (90°≤α<180°)

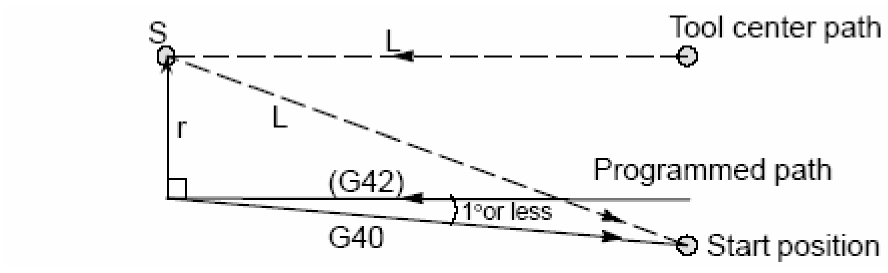
There are 2 tool path types at offset start or cancel: A and B, which is set by bit parameter No.40.0.

A		
B		

(c) Tool movement around an outer side of a corner at an acute angle (α<90°)

There are 2 tool path types at offset start or cancel: A and B, which is set by bit parameter No.40.0.

A		
B		

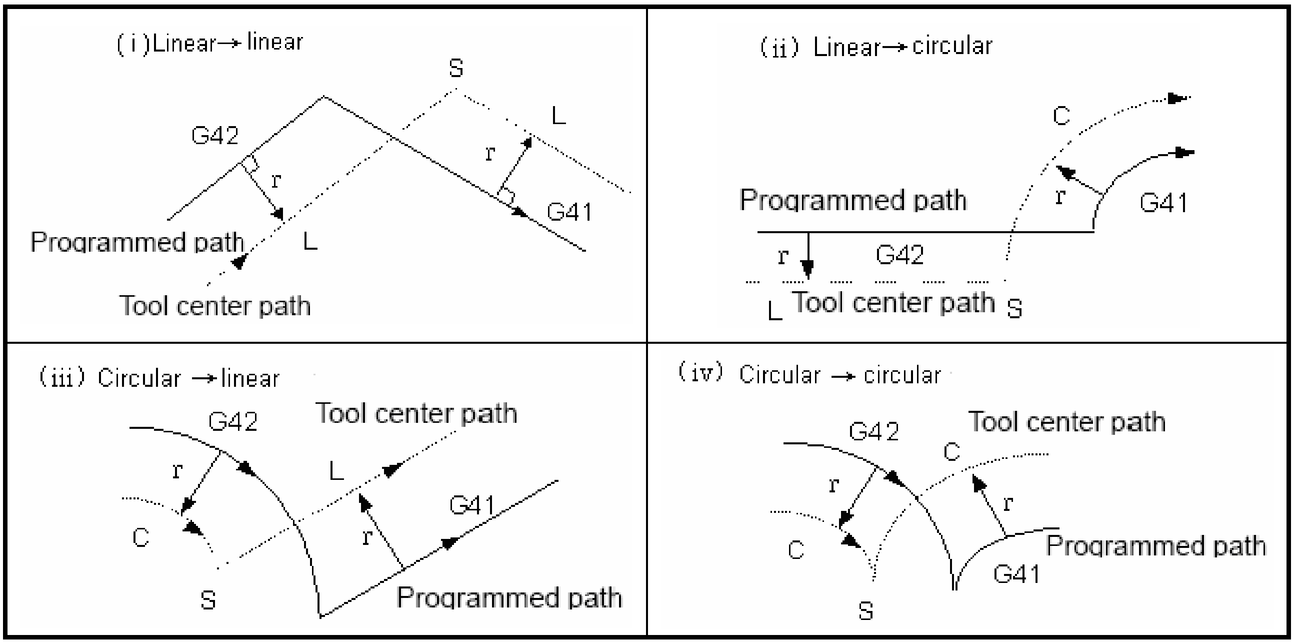
（d） Tool movement around an outer side of a corner at an acute angle less than 1°(α＜1°) Linear→linear


5. Offset direction change in offset mode

The offset direction is defined by tool radius compensation G code. The sign of the offset value is as following:

Sign of offset value G code	+	-
G41	Left offset	Right offset
G42	Right offset	Left offset

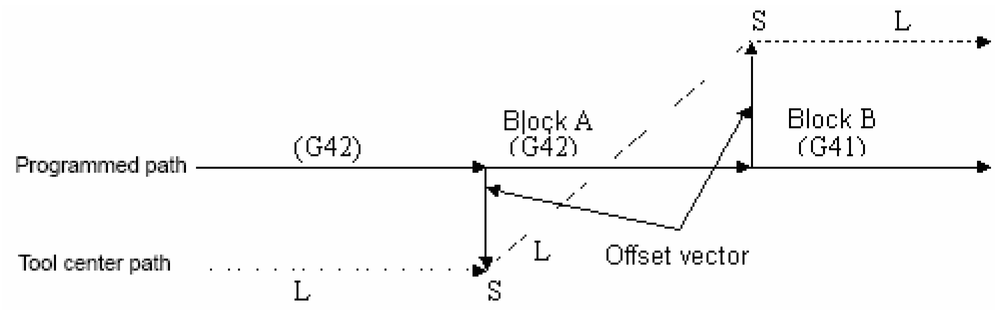
In a special situation, the offset direction can be changed in offset mode, however the direction change is unallowed in the start-up block and the block following it. There is no inner and outer side when the offset direction is changed. The following offset value is assumed to be positive.



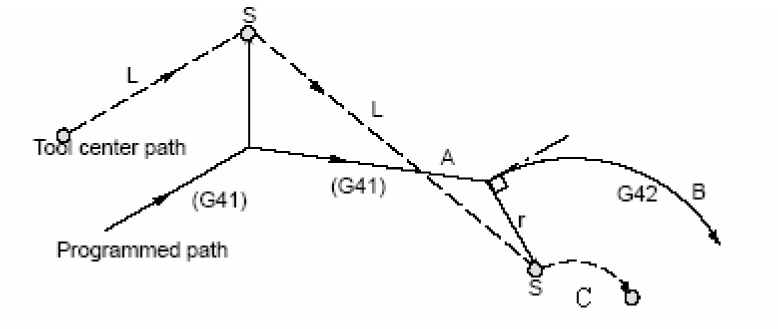
（v）For the offset without an intersection

When changing the offset direction from block A to block B using G41 and G42, if intersection of the offset path is not required, the vector normal to block B is created at the start point.

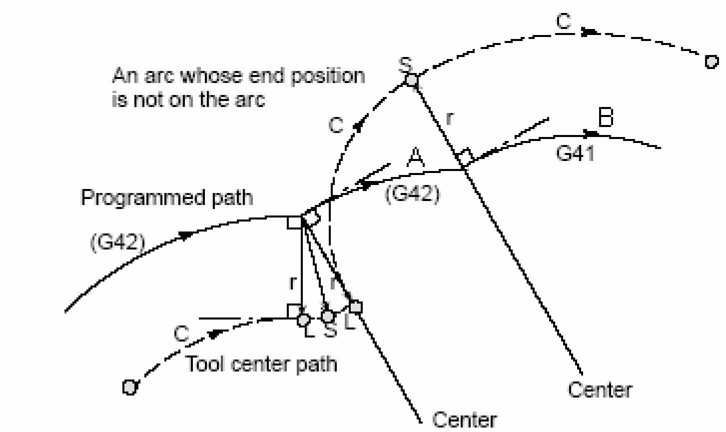
(1) Linear---- linear



(2) Linear----- circular



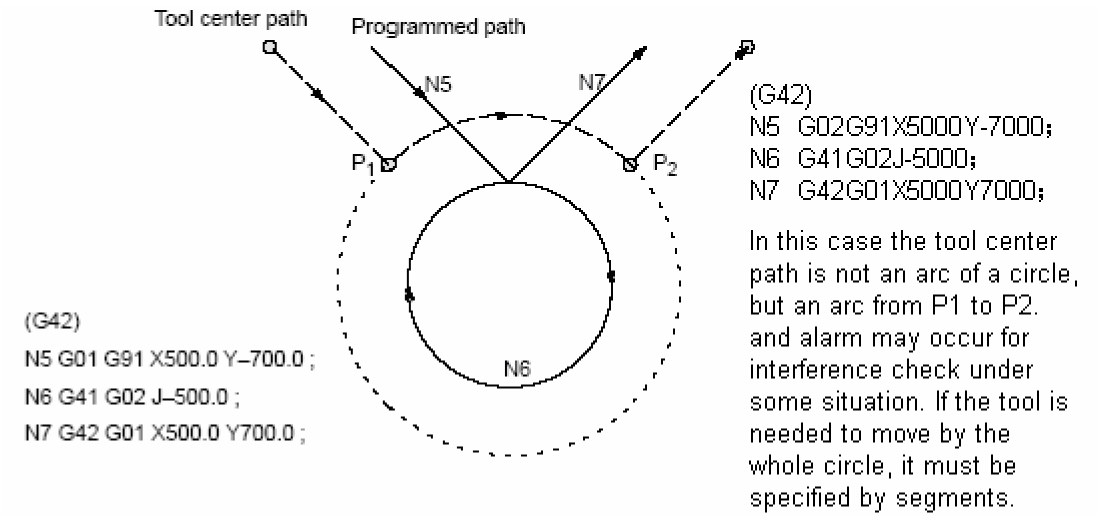
(3) Circular----- circular



（iv）Normally there is almost no possibility of generating the situation that the length of the tool center path is larger than the circumference of a circle. When G41 and G42 are changed, the following situation may occur:

Circular ----- circular (linear-----circular) Alarm occurs if the tool offset direction is changed and alarm that the tool offset can't be cancelled by arc instruction is issued when the tool number is D0.

Linear----- linear The tool offset direction can be changed.



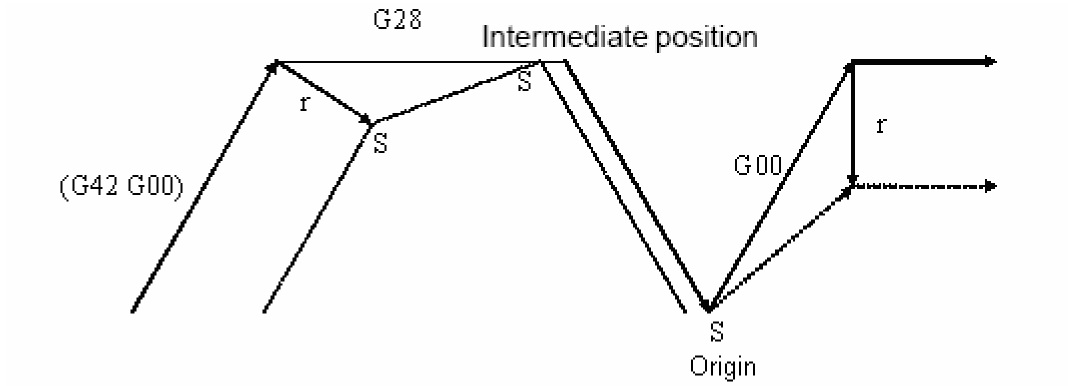
6. Temporary offset cancel

In offset mode, the offset is temporarily cancelled by the following instructions specified by parameter No.40.2.

Refer to offset cancel and offset start for the details of this operation.

a） G28 automatic reference point return

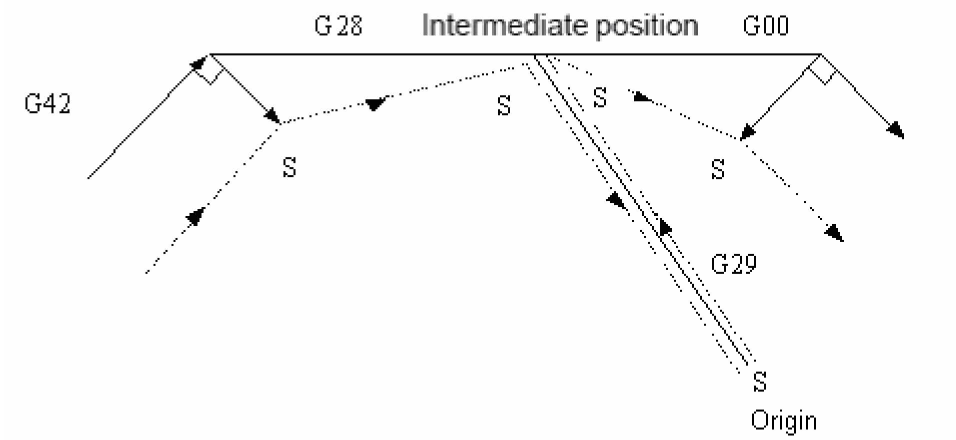
If G28 is specified in offset mode, the offset is cancelled at the intermediate position and automatically restored after reference point return.



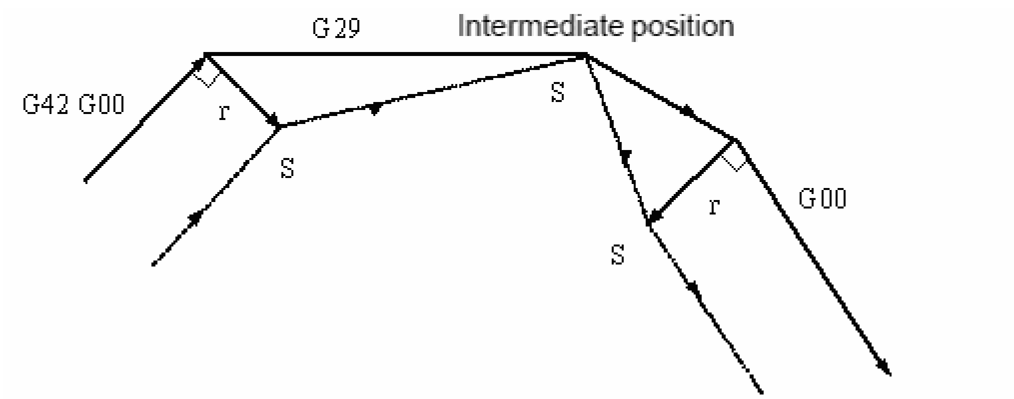
b ） G29 automatic return from reference point

If G29 is specified in offset mode, the offset is cancelled at the intermediate position and automatically restored at the next block.

If it is specified immediately after G28:



If it is not specified immediately after G28:

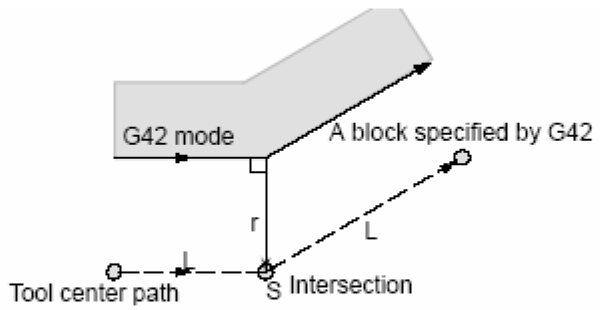


7. Tool radius compensation G code in offset mode

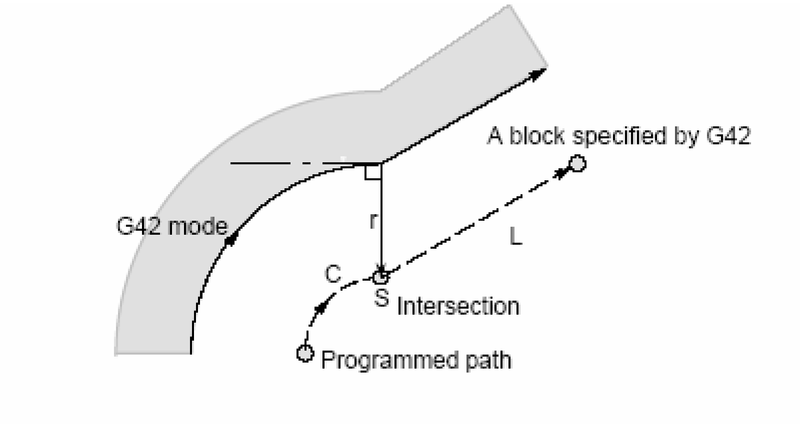
In offset mode, if the tool radius compensation G code (G41, G42) is specified, a vector perpendicular to the previous block will be created, which is irrelative to the machining inner or outer side. If this G code is specified in circular instructions, the arc will not be correctly generated.

If this code is specified in a circular instruction, correct motion will not be obtained.
Refer to (5) for offset direction change by tool radius compensation G (G41，G42)

Linear----- linear



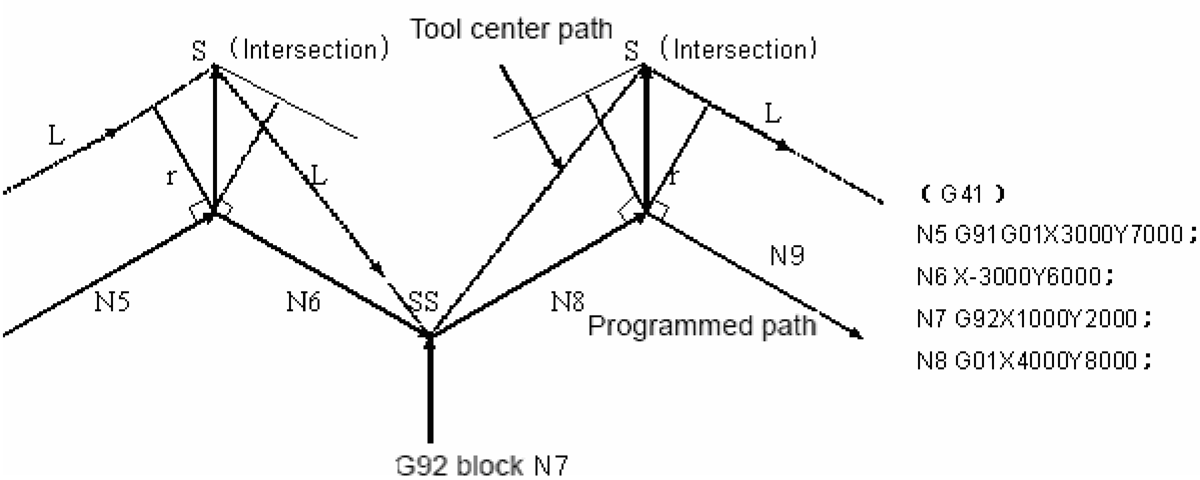
Circular----- linear



8. Instruction for cancelling the offset vector temporarily

In offset mode, if G92 (absolute programming) is specified, the offset vector is temporarily cancelled and then the offset vector is restored automatically.

In this case, different from the offset cancel mode, the tool moves directly from the intersection to the specified point where the offset vector is cancelled. Also when offset mode is restored, the tool moves directly to the intersection.



9. A block without tool movement

The following blocks have no tool movement. In these blocks, the tool will not move even if tool radius compensation is effective.

(1) M05 ; …………………… M code output

(2) S21 ; …………………… S code output

(3) G04 X10000; …………… Dwell

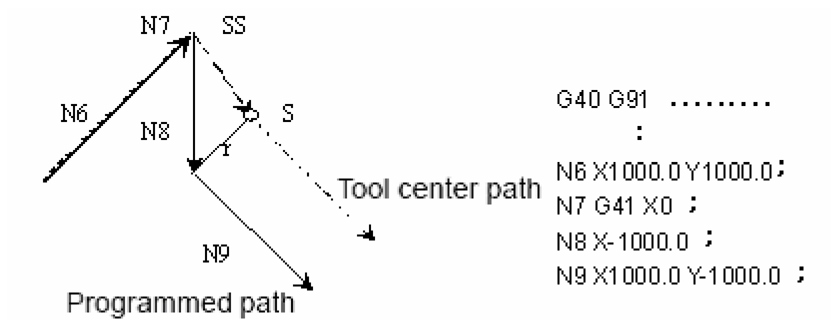
(4) (G17) Z100 ; ……………… Move instruction not included in offset plane

(5) G90 ; ……………………… G code only

(6) G01 G91 X0; ……………Move distance is zero.

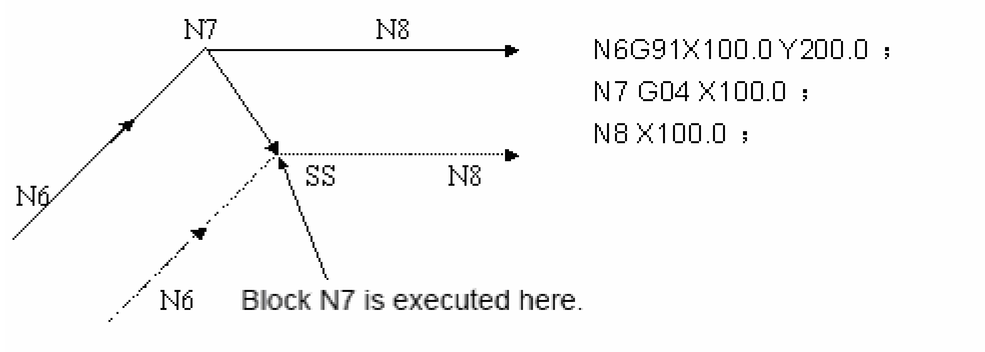
a) Specified at offset start

If the tool movement is not made by the start-up block, it will be done by the next moving instruction block by the system.

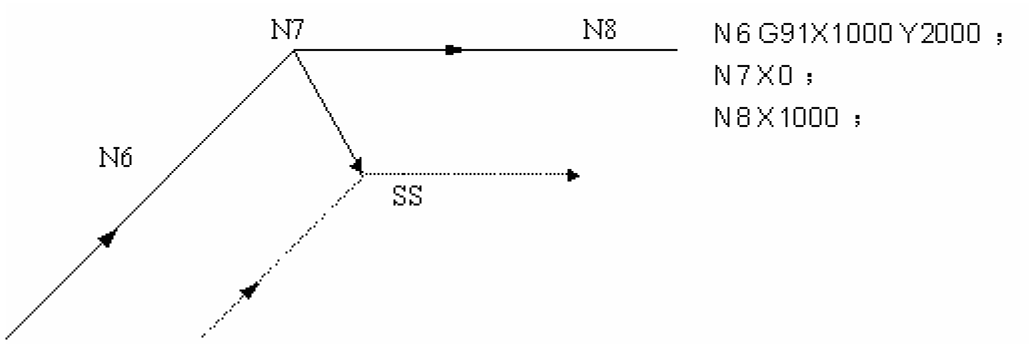


b）Specified at offset mode

If a block with no tool movement is exclusively specified in offset mode, the vector and the tool center path are identical with that the block is not specified. (Refer to item (3)Offset mode). And this block is executed at the single block stop position.



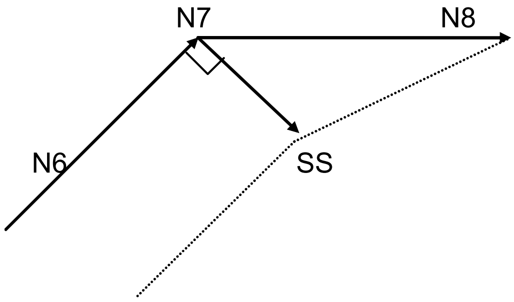
However, when the block moving amount is 0, the tool movement is identical with that of the two or more blocks containing no moving instruction even only one block is specified.



Note The blocks above are executed in G1, G41 mode and the path in G0 doesn' t conforms to the figure.

c）Specified with the offset cancel

A vector with a length offset and the direction perpendicular to the movement direction of the previous block is formed when the block specified together with offset cancel contains no tool movement, and it will be cancelled in next moving instruction.



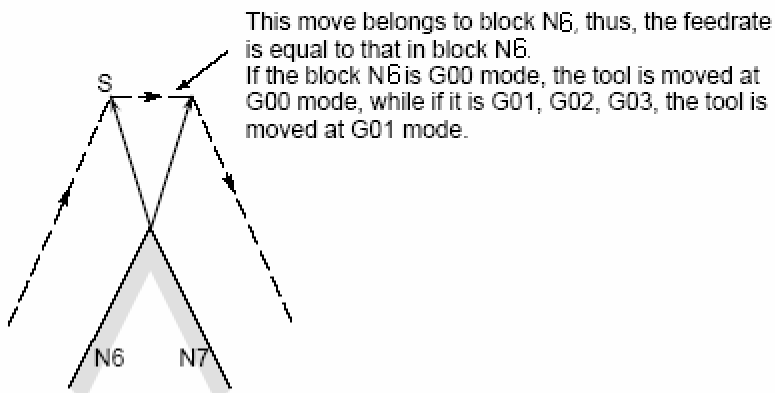
N6 G91X100.0 Y100.0 ;
N7 G40 ;
N8 X100.0 Y0 ;

10. Corner movement

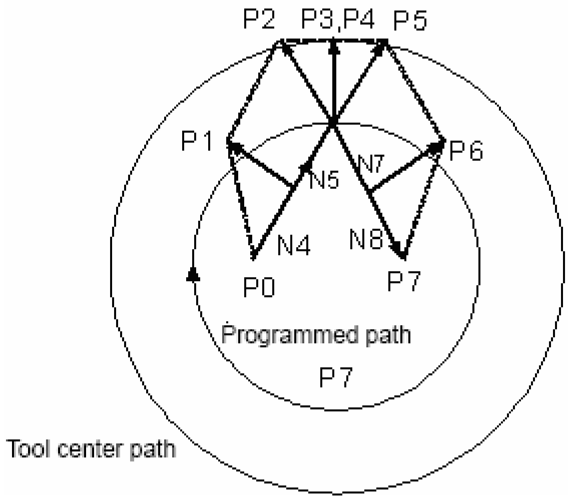
If two or more vector are formed at the end of the block, the tool traverses straightly to another vector from one vector, the movement is called corner movement.

If ΔVX≤ΔV limit and ΔVY≤ΔV limit, the hind vectors are ignored.

If these vectors are not consistent, a movement around the corner is generated, which belongs to the hind block.



But if the path of the next block overpasses the semicircle, the function above is not performed. The reason is that:



N4 G41G91X1500Y2000;
N5 X1500Y2000;
N6 G02J-6000;
N7 G01X1500Y-2000;
N8 G40X1500Y-2000;

If the vector is not ignored, the tool path is as follows:

P0 →P1 →P2 →P3（arc） →P4 →P5 →P6 →P7

If the distance between P2 and P3 is ignored, P3 is ignored, the tool path is as follows:

P0 →P1 →P2 →P4 →P5 →P6 →P7 The arc cutting of the block N6 is ignored.

11. Interference check

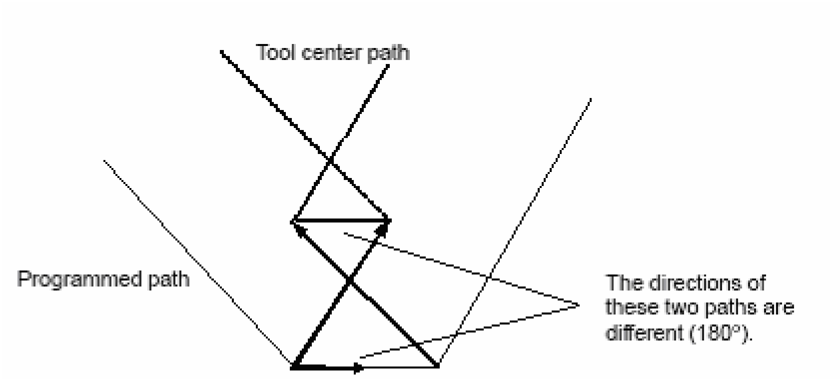
The tool overcutting is called “interference”. The Interference check function checks the tool overcutting in advance. If the interference is dectected by grammar check function after the program is loaded, alarm is issued. The inteference check in tool radius compensation is set by bit parameter No.41.3.

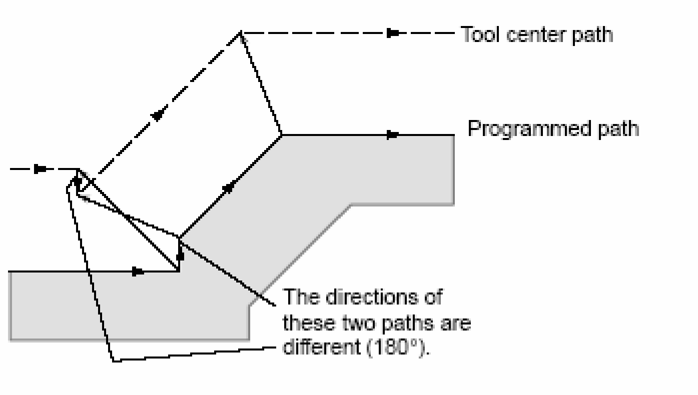
Primary conditions of interference:

（1） The tool path is different from the program path.(The included angle between paths is from 90°to 270°).

（2） Except above conditions, in arc machining, the included angle between the start point and the end point of the tool center path is much different from that of the program path(above 180°).

Example ①





12. Manual operation

See the manual operation in Operation section for the manual tool radius compensation. If the tool length compensation is performed in tool radius compensation, the offset value of the tool radius is regarded to be changed.

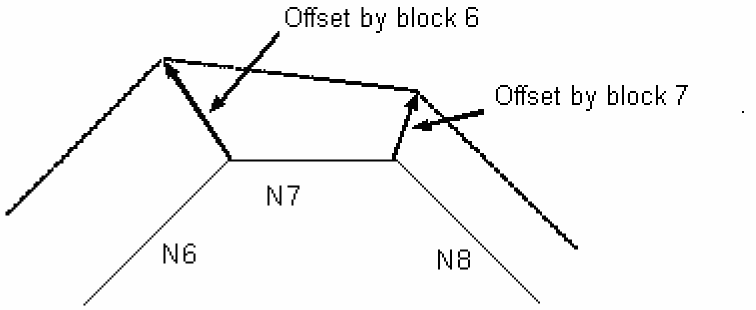
13. Precautions for offset

（a）To specify offset value

The offset value number is specified by D code. Once specified, D code is effective till another one is specified or offset is cancelled. Besides the offset value for the tool radius compensation, it is also used for tool offset value.

（b）To change the offset value

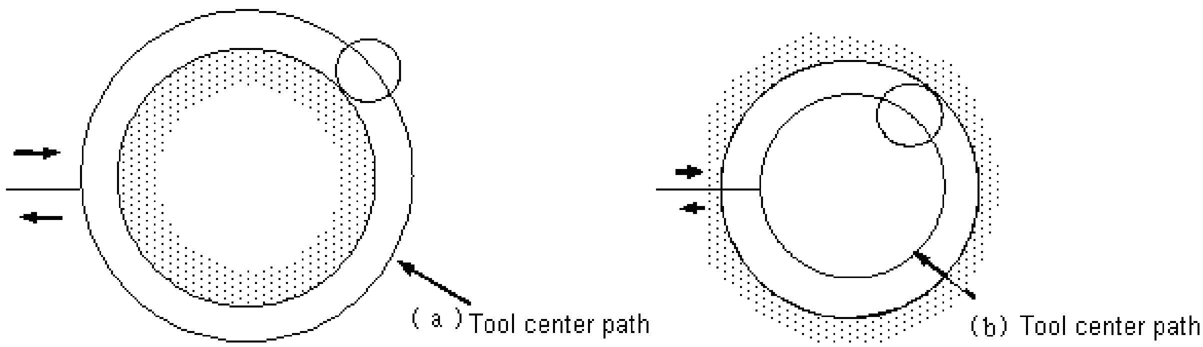
Usually during tool change, the offset value must be changed in offset cancel mode. If it is changed in offset mode, the new offset value is obtained at the block end.



（c）Positive and negative tool offset value and tool center path

If the offset value is negative(－), G41 and G42 is exchanged in program. If the tool center is moving around the outer side of the workpiece, it will pass around the inner side, and vice versa.

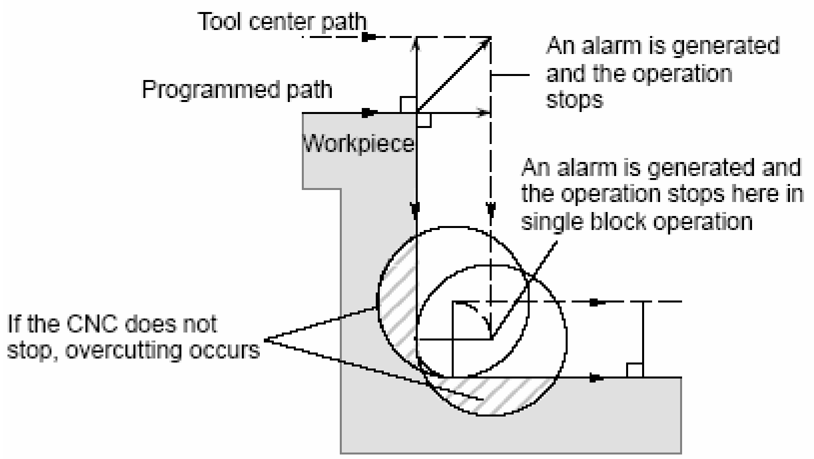
The figure below shows the example. Generally, the offset value is programmed to be positive（＋）. When a tool path is programmed as in figure（a）, if the offset value is made for negative（－）, the tool center moves as in （b）, and vice versa. So the same program permits cutting for male or female shape, and the gap between them can be adjusted by the selection of the offset value.



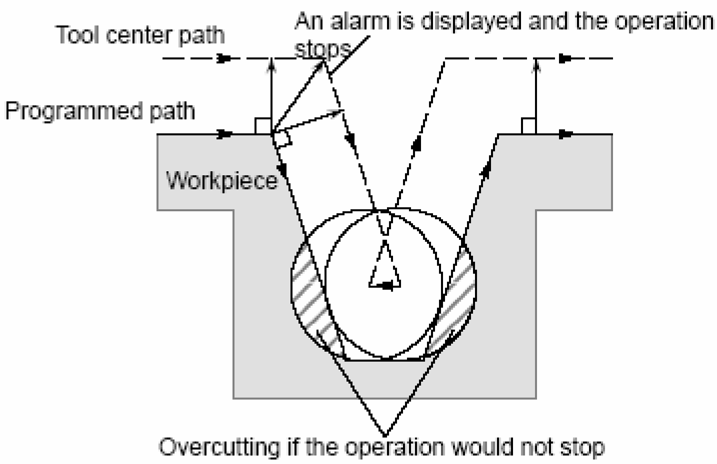
（d） Overcutting by tool radius compensation

（1）Machining an inner side of the corner at a radius smaller than the tool radius

When the radius of a corner is smaller than the tool radius, because the inner offsetting of the tool will result in overcuttings, an alarm will be issued and this is because overcutting is generated when the single block execution is stopped.

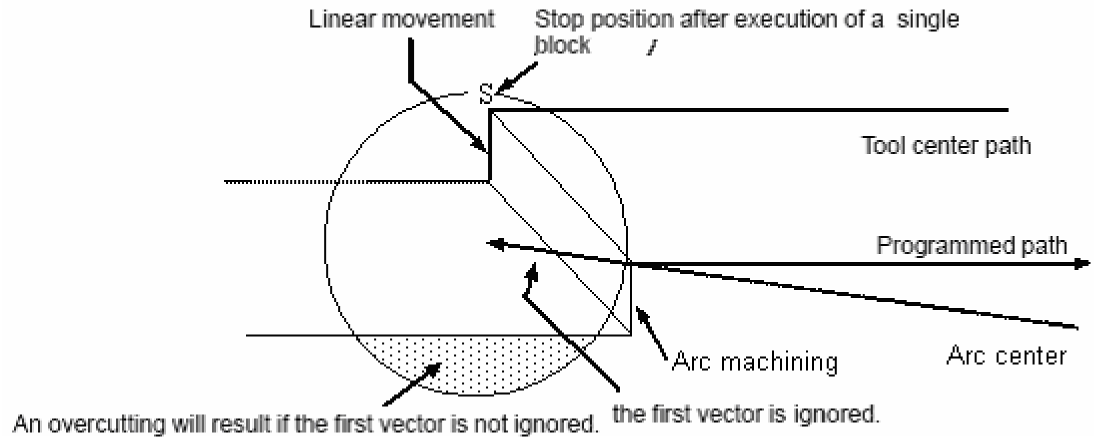


(2) When machining a groove smaller than the tool radius, since the tool redius offset forces the path of the tool center to move in the reverse of the programmed direction, overcutting will result.



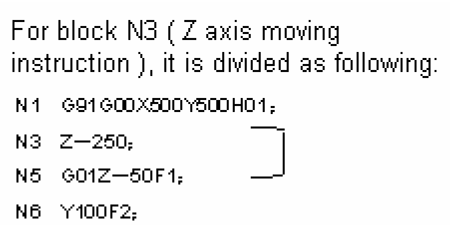
(3) Machining a step smaller than the tool radius

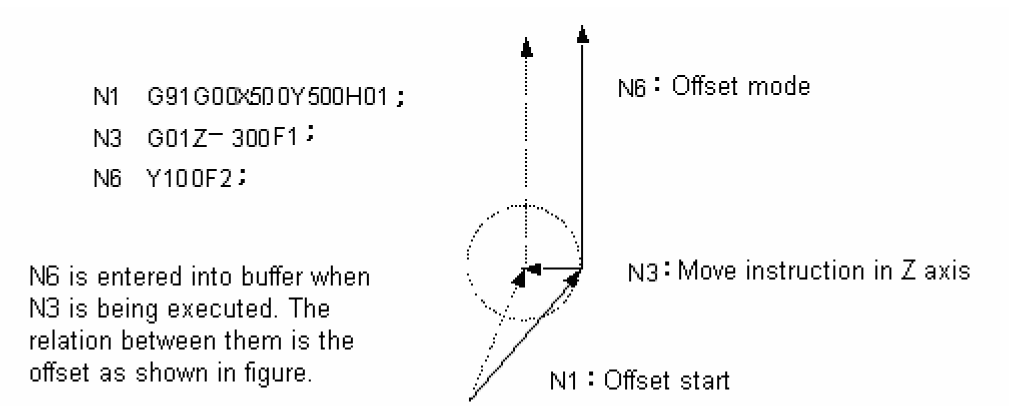
When machining a slot smaller than the tool radius specified by circular machining in the case of a program containing this step, the tool center path with the common offset becomes reverse to the programmed direction. In this case, the first vector is ignored, and the tool moves linearly to the second vector position. This single block operation is stopped at this point. If the machining is not in the single block mode, the auto run is continued. If the step is linear, no alarm will be issued and the tool cuts correctly. But uncut part will remain.



Starting tool radius compensation and cutting along Z axis

It is usually used such a method that the tool is moved along the Z axis after the tool radius compensation is effected at some distance from the workpiece at the start of the machining. In the case above, if it is desired to divide the motion along the Z axis into rapid feed and cutting feed, follow the precedure below:





4.5.4 Corner offset circular interpolation（G39）

Format: G39 or

I_ J_

G39 I_ K_

J_ K_

Function: By specifying G39 in offset mode during tool radius compensation, corner offset circular interpolation can be specified. The radius of the corner offset equals the offset value. And the effectiveness of the corner arc in radius compensation is set by bit parameter No.41.6.

Explanation:

1、 When the instruction above is specified, corner circular interpolation in which the radius equals offset value can be performed.

2、 G41 or G42 preceding this instruction determines whether the arc is CW or CCW. G39 is a non-modal G code.

3、 When G39 (without I, J, K) is programmed, the arc at the corner is formed so that the vector at the end point of the arc is perpendicular to the start point of the next block. It is shown as follows:

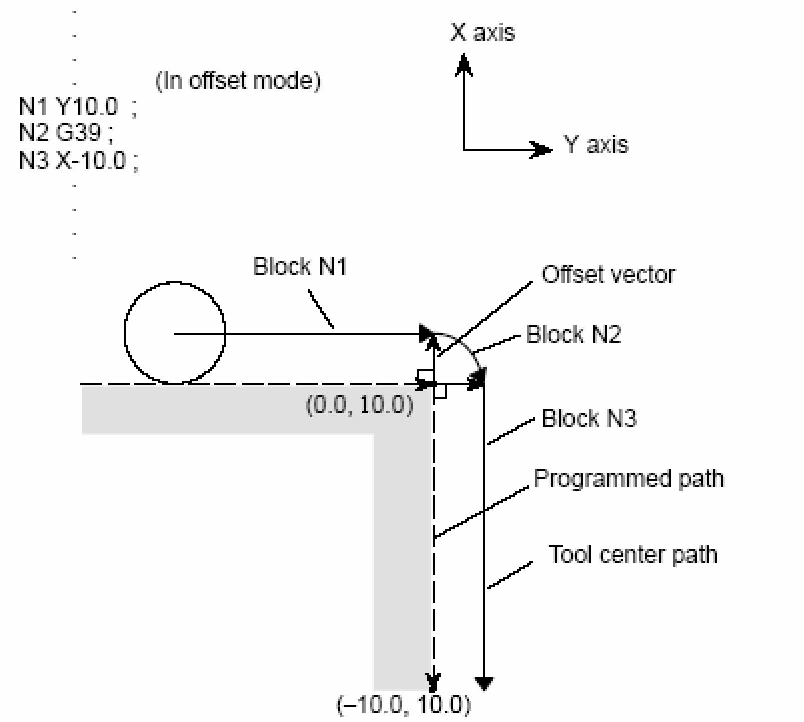


Fig. 4-5-4-1 G39 without I, J, K

4、 When G39 is specified with I, J, K, the arc at the corner is formed so that the vector at the end of the arc is perpendicular to the vector defined by the I, J, K values. It is shown as follows:

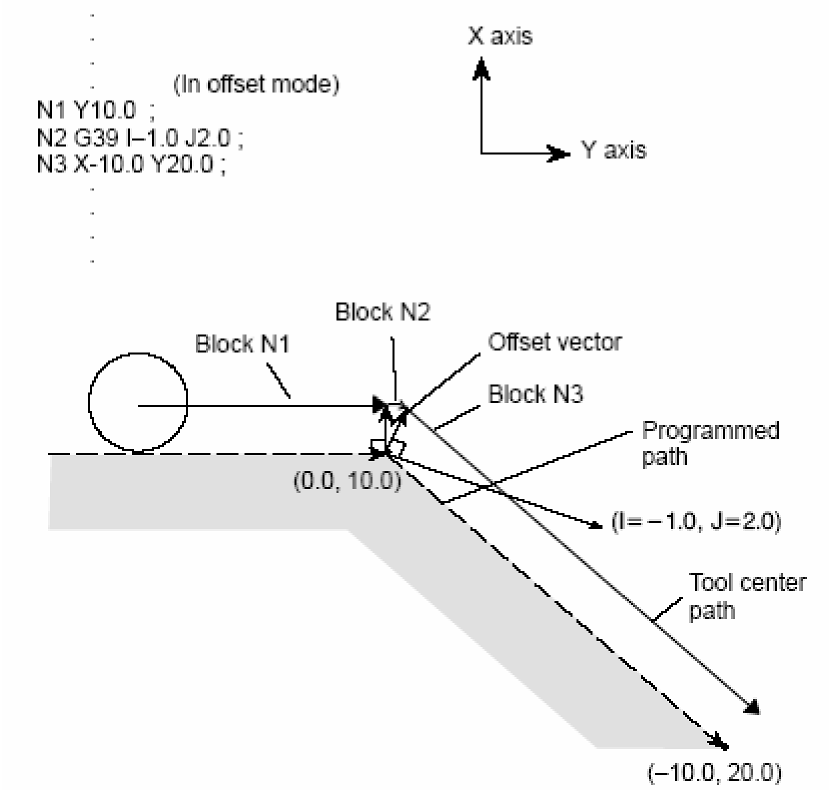


Fig. 4-5-4-2 G39 containing I, J, K

4.5.5 Tool offset value and number input by program（G10）

Format:

G10 L10 P_ R_ ；	Geometric offset value of H code
G10 L12 P_ R_ ；	Geometric offset value of D code
G10 L11 P_ R_ ；	Wear offset value of H code
G10 L13 P_ R_ ；	Wear offset value of D code
P ：	Tool offset number
R ：	Tool offset value in absolute mode（G90）

For the tool offset value in incremental mode（G91）, it is added by the value of the offset number specified (the result is the tool offset value.)

Explanation: The range of tool offset value:

Geometric offset: metric input ±999.999mm；inch input ±99.9999 inch

Wear offset: metric input ±99.999mm；inch input ±9.9999 inch

Note **For inch and metric switch, the tool offset value automatic change is set by bit parameter No.41.0.**

4.6 Feed G code

4.6.1 Feed mode G64/G61/G63

Format:

Dwell (exact stop) mode G61

Tapping mode G63

Cutting mode G64

Function:

Dwell mode G61: Once specified, this function is effective till G62, G63 or G64 is specified. The tool is decelerated for an in-position check at the end point of a block, then next block is executed.

Tapping mode G63: Once specified, this function is effective till G62, G61 or G64 is specified. The tool is not decelerated at the end point of a block, but the next block is executed. When G63 is specified, feedrate override and feed hold are both ineffective.

Cutting mode G64: Once specified, this function is effective till G62, G61 or G63 is specified. The tool is not decelerated at the end point of a block, and the next block is executed.

Explanation:

No parameter format.

G64 is the system default feed mode, no deceleration is performed at the end point of a block and next block is executed directly.

The purpose of in-position check in dwell mode is to check whether the servo motor has reached within a specified range.

In exact stop mode, the tool movement paths in cutting mode and tapping mode are different.

See following Fig. 4-6-1-1:

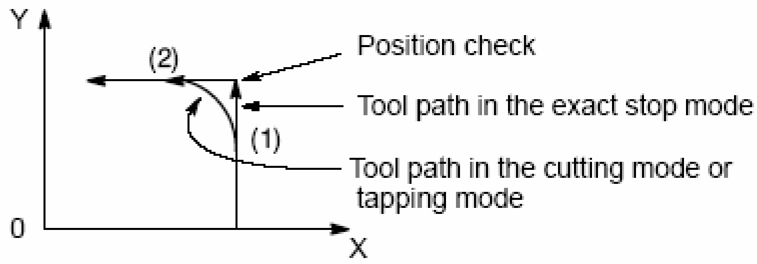


Fig. 4-6-1-1 Tool path from block 1 to block 2

4.6.2 Automatic override for inner corners（G62）

Format: G62

Function: Once specified, this function is effective till G63, G61 or G64 is specified. When the tool moves along an inner corner during tool radius compensation, override is applied to the cutting feedrate to suppress the amount of cutting per unit of time to get a good surface finish.

Explanation:

1、When the tool moves along an inner corner and inner arc area during tool radius compensation, it decelerates automatically to reduce the load of the tool to get a smooth surface.

2、When G62 is specified, and the tool path with tool radius compensation forms an inner corner, the feedrate is automatically overriden at both ends of the corner. There are four types of inner corners as shown in Fig. 4-6-2-1. In figure: 2°≤θ≤θp≤178°; θp is set by number parameter P144.

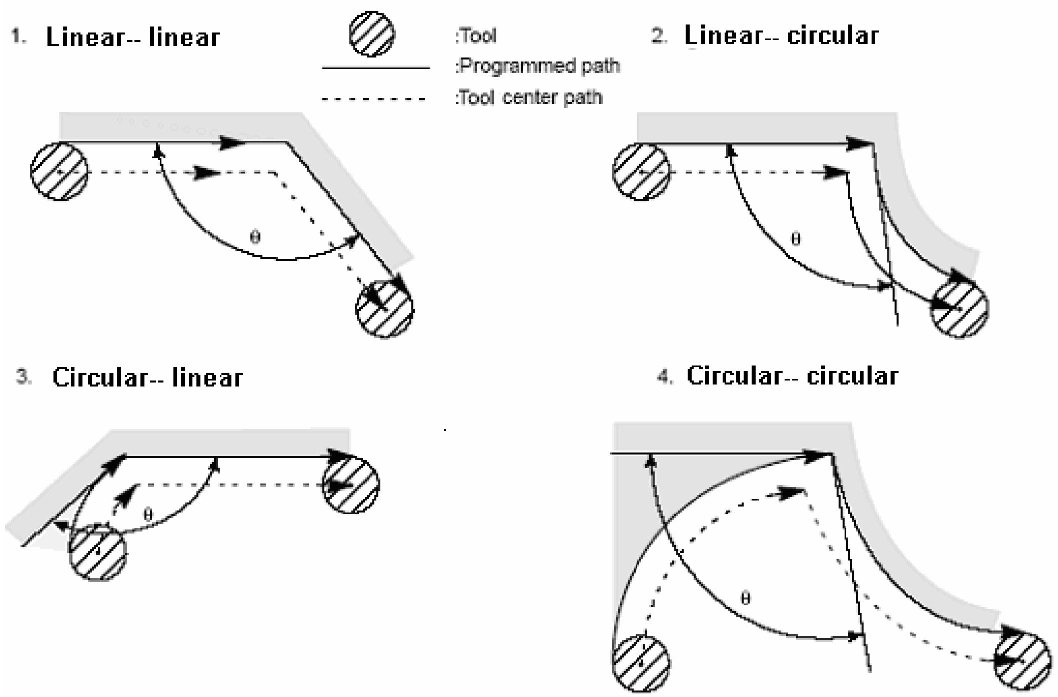


Fig. 4-6-2-1

3、When a corner is determined to be an inner corner, the feedrate is overriden before and after the inner corner. The Ls and Le, where the feedrate is overriden, are distances from points on the tool center path to the corner (Fig. 4-6-2-2), where Ls+Le≤2mm.

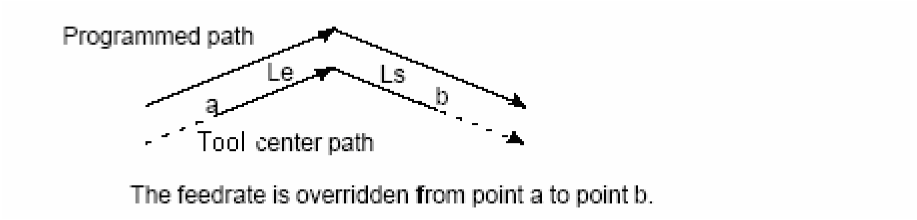


Fig. 4-6-2-2 Straight line to straight line

4 When a programmed path consists of two arcs, the feedrate is overriden if the start and end points are in the same quadrant or in adjacent quadrants. (Fig. 4-6-2-3)

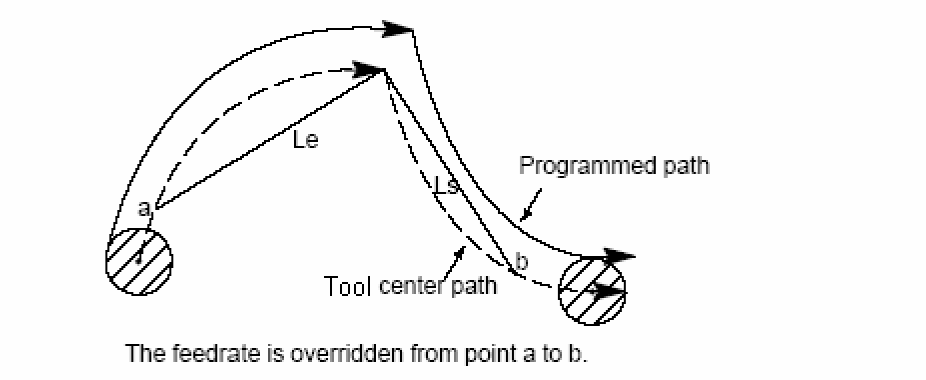


Fig. 4-6-2-3 Arc to arc

5 Regarding a program from straight line to arc or from arc to straight line, the feedrate is overriden from point a to point b and from point c to point d. (Fig. 4-6-2-4)

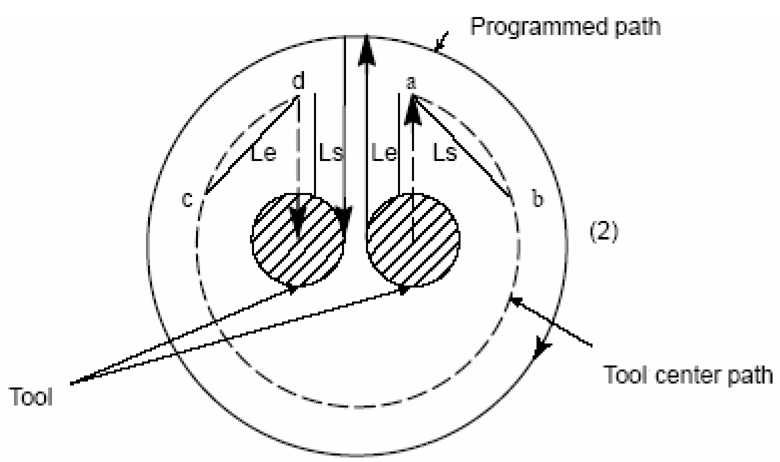


Fig. 4-6-2-4 Straight line to arc, arc to staight line

Restriction

1 Override for inner corners is disabled during acceleration/deceleration before interpolation.

2 Override for inner corners is disabled if the corner is preceded by a start-up

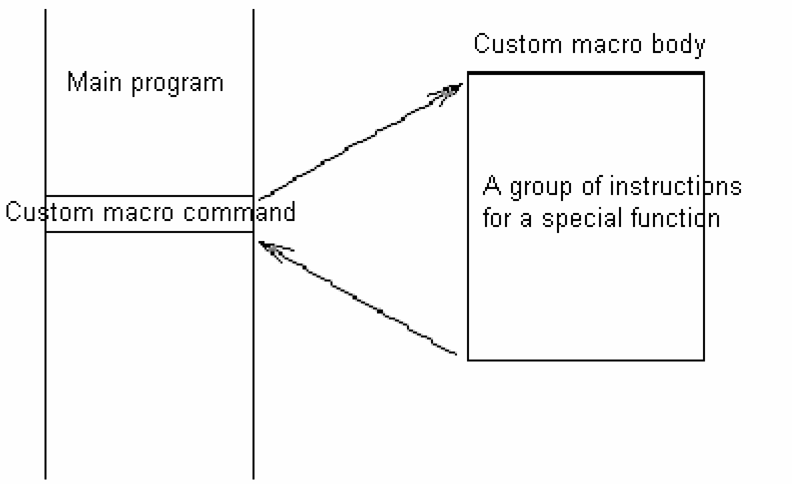
block or followed by a block including G41 or G42.

3 Override for inner corner is not performed if the offset is zero.

4.7 Macro G code

4.7.1 Custom macro

The function by a group of instructions can be saved into memory like a subprogram in advance, and the functions are represented by an instruction. If the instruction is written out in the program, these functions can be used. This group of instructions is called custom macro body and the instruction represented is called “custom macro instruction”. The custom macro body is also abreviated for macro. The custom macro instruction is also called macro calling instruction.



Variables can be used in custom macro body, and they can be operated and assigned by macro instructions.

4.7.2 Macro variables

Both the common CNC instructions and the variables, operation as well as the transfer instructions can be used in the custom macro body. It begins with program number and ends with M99.

O8000 ;	Program number
G65 H01 ... ... ;	Operation instruction
G90 G00 X#101 ... .. ;	CNC instructions using variables
...	
...	
...	
G65 H82 ... ;	Transfer instruction
...	
...	
M99 ;	Custom macro body ends

The composition of custom macro body

1. Variables usage

With a variable, the parameter value in custom macro body can be specified. The variable value can be assigned by the main program or set by LCD/MDI; or be assigned by a computation during custom macro body execution.

Multiple variables can be used in custom macro and they are differentiated by their variable numbers.

(1) Variable representation

The variable is expressed by a sign # followed by a variable number, Format:

#i (i = 1， 2， 3， 4 ……)

(e.g.) #5， #109， #1005

(2) Variable citation

The variable can be used to replace the value of parameter.

(e.g) F#103 When #103 = 15, it is the same as F15.

G#130 When #103 =3, it is the same as G3.

Note 1 Variable cann't be cited by parameter word O and N (program number and sequence number). Such as O#100，N#120 are not permitted in programming.

Note 2 Variable exceeding the max. limit of the parameter can' t be used. When #30 = 120, M#30 exceeds the max. limit of the instruction.

Note 3 Display and setting of variable: It can be displayed on LCD, or be set by MDI.

2. Typies of variables

Variables are classified into null variables, local variables, common variables and system variables with different applications and characteristics.

（1）Null variables: #0 (This variable always be null, no value can be assigned to it.)

（2）Local variables: #1～#50：

They can only be used for data storage in a macro such as the results of operations. When the power is turned off, they are initialized for null. When a macro is called, arguments are assigned to local variables.

（3）Common variables: #100～#199，#500～#999：

They can be shared among the main program and the custom macros called by the main program. Namely the variable #I in a custom macro program is identical with that in other macro program. So the common variable #I of operation result of a macro program can be used in other macro programs.

Common variables usage not specified in this system can be used freely by user.

Variable number	Variable type	Function
＃100～＃199	Common variables	Cleared at power-off, all reset for “null” at power-on
＃500～＃999		Data saved in files and reserved even power-off

(4) System variables:

They are used for reading and writing a variety of CNC data, which are shown as follows:

1) Interface input signal #1000 --- #1047 (read signal input by PLC by bits)

2) Interface output signal #1100 --- #1147 (write signal output to PLC by bits)

3) Tool length offset value #1500～#1755 （readable and writable）

4) Tool length wear offset value #1800～#2055 （readable and writable）

5) Tool radius offset value #2100～#2355 （readable and writable）

6) Tool radius wear offset value #2400～#2655 （readable and writable）

7) Tool magazine data list #2700～#2955 （read-only，unwritable）

8) Alarm #3000

9) User data list #3500～#3755 （read-only，unwritable）

10) Modal message #4000～#4030 （read-only，unwritable）

11) Position message #5001～#5030 （read-only，unwritable）

12) Workpiece zero offset #5201～#5235 （readable and writable）

13) Additional workpiece coordinate system #7001～#7250 （readable and writable）

3. Explanation for system variables

1）Modal message

Variable No.	Function	Group No.
#4000	G10,G11	00
#4001	G00,G01,G02,G03	01
#4002	G17,G18,G19	02
#4003	G90,G91	03
#4004	G94,G95	04
#4005	G54,G55,G56,G57,G58,G59	05
#4006	G20,G21	06
#4007	G40,G41,G42	07
#4008	G43,G44,G49	08
#4009	G73,G74,G76,G80,G81,G82,G83,G84,G85,G86,G87,G88,G89	09
#4010	G98,G99	10
#4011	G15,G16	11
#4012	G50,G51	12
#4013	G68,G69	13
#4014	G61,G62,G63,G64	14
#4015	G96,G97	15
#4016	To be expanded	16
#4017	To be expanded	17
#4018	To be expanded	18
#4019	To be expanded	19
#4020	To be expanded	20
#4021	To be expanded	21
#4022	D	
#4023	H	
#4024	F	
#4025	M	
#4026	S	
#4027	T	
#4028	N	
#4029	O	
#4030	P (current selected additional coordinate system)	

Note 1 P code stands for the current selected additional coordinate system.
Note 2 When G#4002 code is being executed,the value obtained in #4002 is 17, 18 or 19.
Note 3 The modal message can be read, but cannot be written.

2）Current position message

Variable No.	Position message	Relative coordinate system	Read operation as moving	Tool offset value
#5001	Block end position of X axis (ABSIO)	Workpiece coordinate system	Allowed	Tool nose position not involved（program specified position）
#5002	Block end position of Y axis (ABSIO)			
#5003	Block end position of Z axis (ABSIO)			
#5004	Block end position of 4th axis (ABSIO)			
#5005	Block end position of 5th axis (ABSIO)			
#5006	Block end position of X axis (ABSMT)	Machine coordinate system	Unallowed	Tool basic position involved（Machine coordinate）
#5007	Block end position of Y axis (ABSMT)			
#5008	Block end position of Z axis (ABSMT)			
#5009	Block end position of 4th axis (ABSMT)			
#5010	Block end position of 5th axis (ABSMT)			
#5011	Block end position of X axis (ABSOT)	Workpiece coordinate system		
#5012	Block end position of Y axis (ABSOT)			
#5013	Block end position of Z axis (ABSOT)			
#5014	Block end position of 4th axis (ABSOT)			
#5015	Block end position of 5th axis (ABSOT)			
#5016	Block end position of X axis (ABSKP)		Allowed	
#5017	Block end position of Y axis (ABSKP)			
#5018	Block end position of Z axis (ABSKP)			
#5019	Block end position of 4th axis (ABSKP)			
#5020	Block end position of 5th axis (ABSKP)			

#5021	Tool length offset value of X axis			
#5022	Tool length offset value of Y axis		Unallowed	
#5023	Tool length offset value of Z axis			
#5024	Tool length offset value of 4th axis			
#5025	Tool length offset value of 5th axis			
#5026	Servo position offset of X axis			
#5027	Servo position offset of Y axis			
#5028	Servo position offset of Z axis			
#5029	Servo position offset of 4th axis			
#5030	Servo position offset of 5th axis			

Note ABSIO：the last block end point coordinate in workpiece coordinate system
ABSMT：the current machine coordinate system position in machine coordinate system
ABSOT：the current coordinate position in workpiece coordinate system
ABSKP：effective position of G31 block skip signal in workpiece coordinate system

3）Workpiece zero offset value and additional zero offset value

Variable No.	**Function**
#5201 … #5205	External workpiece zero offset value of the 1st axis … External workpiece zero offset value of the 5th axis
#5206 … #5210	G54 workpiece zero offset value of the 1st axis … G54 workpiece zero offset value of the 5th axis
#5211 … #5215	G55 workpiece zero offset value of the 1st axis … G55 workpiece zero offset value of the 5th axis
#5216 … #5220	G56 workpiece zero offset value of the 1st axis … G56 workpiece zero offset value of the 5th axis
#5221 … #5225	G57 workpiece zero offset value of the 1st axis … G57 workpiece zero offset value of the 5th axis
#5226 …	G58 workpiece zero offset value of the 1st axis …

#5230	G58 workpiece zero offset value of the 5th axis
#5231 … #5235	G59 workpiece zero offset value of the 1st axis … G59 workpiece zero offset value of the 5th axis
#7001 … #7005	G54 P1 workpiece zero offset value of the 1st axis … G54 P1 workpiece zero offset value of the 5th axis
#7006 … #7010	G54 P2 workpiece zero offset value of the 1st axis … G54 P2 workpiece zero offset value of the 5th axis
#7246 … #7250	G54 P50 workpiece zero offset value of the 1st axis … G54 P50 workpiece zero offset value of the 5th axis

4. Local variable

The correspondence of address and local variable:

Argument address	Local variable No.	Argument address	Local variable No.
A	#1	Q	#17
B	#2	R	#18
C	#3	S	#19
I	#4	T	#20
J	#5	U	#21
K	#6	V	#22
D	#7	W	#23
E	#8	X	#24
F	#9	Y	#25
M	#13	Z	#26

Note 1 The assignment is done by an English letter followed by a numerical value. Besides the letters G, L, O, N, H and P, the other 20 letters can also be assigned for arguments. Each letter from A-B-C-D… to X－Y－Z can be assigned once and the they need not to be assigned by letter order. The addresses not assigned may be omitted.

Note 2 G65 should be specified prior to argument using.

5. Precautions for custom macro body

1) Input by keys

Press # key behind the parameter words G, X, Y, Z, R, I, J, K, F, H, M, S, T, P, Q for inputting “#”

2) Either operation or transfer instruction can be specified in MDI mode.

   Except G65, other parameter data can be input by keys but can't be displayed.

3) H, P, Q, R of the operation and transfer instructions preceding or behind G65 are all used as parameters for G65.

   H02 G65 P#100 Q#101 R#102 ； Correct

   N100 G65 H01 P#100 Q10 ； Correct

4) Variable range: －99999～99999

5) The result of the variable operation can be a decimal fraction with a precision of 0.0001. All operations, except H11 (OR operation), H12 (AND operation), H13(NOT operation),H23(ROUNDING operation) with decimal portion neglected in operation, are done without the decimal portions abnegated.

Example:

#100 ＝ 35， #101 ＝ 10， #102 ＝ 5

#110 ＝ #100÷#101 (＝3.5)

#111 ＝ #110×#102 (＝17.5)

#120 ＝ #100×#102 (＝175)

#121 ＝ #120÷#101 (＝17.5)

6) The execution time of operation and transfer instruction differs depending on different conditions, usually the average time is 10ms.

7) See the details for common variables operation in *OPERATION* manual input mode section.

4.7.3 Custom macro call

When G65 is specified, the custom macro specified by address P is called, and the data are transferred to custom macro body by arguments.

Format:

G65 P □□□□□ L □□□□ <By arguments> ；

Calling times

Number of the custom macro body called

Behind G65 code, P is used to specify custom macro number, L is used to specify custom macro calling times, and the arguments are used to transfer data to custom macro.

If repetition is needed, specify the repetition times behind L code; if L is omitted, the default time is 1. If it is specified by arguments, the values will be assigned to the corresponding local variables.

Note 1 If the subprogram number specified by address P is not retrieved, an alarm (PS 078) will be issued.

Note 2 No. 90000~99999 subprograms are the system reserved programs, if they are called, they can be executed, but the cursor will rest on at M98 block, and the program interface displays the main program all the time.

4.7.4 Operation and transfer instruction

1. Format:

G65 Hm P#i Q#j R#k ；

m： 01～99 represent the operation or transfer function.

#i： Variable name for saving operation result.

#j： Variable name 1 for operation, Or a constant which is expessed directly without #.

#k： Variable name 2 for operation. Or a constant.

Significance： #i = #j ○ #k

└──── Operation sign, specified by Hm

Example: P#100 Q#101 R#102……#100 = #101 ○ #102 ；

P#100 Q#101 R15 ……#100 = #101 ○ 15 ；

P#100 Q-100 R#102……#100 = -100 ○ #102 ；

H code specified by G65 has no effect to the offset selection.

G code	H code	Function	Definition
G65	H01	Value assignment	#i ＝ #j
G65	H02	Addition	#i ＝ #j ＋ #k
G65	H03	Subtraction	#i ＝ #j － #k
G65	H04	Multiplication	#i ＝ #j × #k
G65	H05	Division	#i ＝ #j ÷ #k
G65	H11	Logic addition(OR)	#i ＝ #j OR #k
G65	H12	Logic multiplication (AND)	#i ＝ #j AND #k
G65	H13	AND-OR	#i ＝ #j XOR #k
G65	H21	Square root	$\#i=\sqrt{\#j}$
G65	H22	Absolute value	#i ＝ ∣#j∣
G65	H23	Compliment	#i＝#j-trunc(#j÷#k)×#k
G65	H24	Algorism to binary	#i ＝ BIN(#J)
G65	H25	Binary to algorism	#i ＝ BCD(#J)
G65	H26	Compound multiplication and division	#i ＝ (#i×#j) ÷ #k
G65	H27	Compound square root	$\#i=\sqrt{\#j^2+\#k^2}$
G65	H31	Sine	#i ＝ #j×SIN(#k)
G65	H32	Cosine	#i ＝ #j×COS(#k)
G65	H33	Tangent	#i ＝ #j×TAN(#k)

G65	H34	Arctangent	#i ＝ ATAN(#j/#k)
G65	H80	Unconditional transfer	Turning N
G65	H81	Conditional transfer 1	IF #j ＝ #k， GOTO N
G65	H82	Conditional transfer 2	IF #j ≠ #k， GOTO N
G65	H83	Conditional transfer 3	IF #j ＞ #k， GOTO N
G65	H84	Conditional transfer 4	IF #j ＜ #k， GOTO N
G65	H85	Conditional transfer 5	IF #j ≥ #k， GOTO N
G65	H86	Conditional transfer 6	IF #j ≤ #k， GOTO N

2. Operation instruction:

1）Variable assignment: # I = # J

G65 H01 P#I Q#J

Example: G65 H01 P# 201 Q1005； （#201 = 1005）

G65 H01 P#201 Q#210； （#201 = #210）

G65 H01 P#201 Q-#202； （#201 = -#202）

2）Addition: # I = # J+# K

G65 H02 P#I Q#J R#K

Example: G65 H02 P#201 Q#202 R15； （#201 = #202+15）

3）Subtraction: # I = # J－# K

G65 H03 P#I Q#J R# K；

Example: G65 H03 P#201 Q#202 R#203； （#201 = #202－#203）

4）Multiplication: # I = # J×# K

G65 H04 P#I Q#J R#K；

Example: G65 H04 P#201 Q#202 R#203； （#201 = #202×#203）

5）Division: # I = # J÷# K

G65 H05 P#I Q#J R#K

Example: G65 H05 P#201 Q#202 R#203； （#201 = #202÷#203）

6）Logic addition (OR): # I = # J.OR. # K

G65 H11 P#I Q#J R#K；

Example: G65 H11 P#201 Q#202 R#203； （#201 = #202.OR. #203）

7）Logic multiplication (AND): # I = # J.AND. # K

G65 H12 P#I Q#J R#K；

Example: G65 H12 P# 201 Q#202 R#203； （#201 = #202.AND.#203）

8）AND-OR: # I = # J.XOR. # K

G65 H13 P#I Q#J R#K

Example: G65 H13 P#201 Q#202 R#203； （#201 = #202.XOR. #203）

9）Square root：# I = $\sqrt{\#J}$

G65 H21 P#I Q#J ；

Example: G65 H21 P#201 Q#202 ； （#201 = $\sqrt{\#202}$ ）

10）Absolute value: # I = | # J |

G65 H22 P#I Q#J ；

Example: G65 H22 P#201 Q#202 ； （#201 = | #202 |）

11）Rounding: # I = # J－TRUNC(#J/#K)×# K，TRUNC：reserving or abnegating decimal portion

G65 H23 P#I Q#J R#K

Example: G65 H23 P#201 Q#202 R#203； （#201 = #202- TRUNC (#202/#203)×#203

12）Algorism to binary: # I = BIN (# J)

G65 H24 P#I Q#J ；

Example: G65 H24 P#201 Q#202 ； （#201 = BIN（#202））

13）Binary to algorism：# I = BCD (# J)

G65 H25 P#I Q#J ；

Example: G65 H25 P#201 Q#202 ； （#201 = BCD（#202））

14）Compound multiplication and division: # I =（# I×# J）÷# K

G65 H26 P#I Q#J R# k；

Example: G65 H26 P#201 Q#202 R#203； （#201 =（# 201×# 202）÷# 203）

15）Compound square root: # I = $\sqrt{\#J^2+\#K^2}$

G65 H27 P#I Q#J R#K

Example: G65 H27 P#201 Q#202 R#203； （#201 = $\sqrt{\#202^2+\#203^2}$ ）

16）Sine: # I = # J•SIN（# K）（Unit：‰ degree）

G65 H31 P#I Q#J R#K；

Example: G65 H31 P#201 Q#202 R#203； （#201 = #202•SIN（#203））

17）Cosine: # I = # J•COS（# K）（Unit：‰ degree）

G65 H32 P#I Q#J R# k；

Example: G65 H32 P#201 Q#202 R#203； （#201 =#202•COS（#203））

18）Tangent: # I = # J•TAM（# K）（Unit：‰ degree）

G65 H33 P#I Q#J R# K；

Example: G65 H33 P#201 Q#202 R#203； （#201 = #202•TAM（#203））

19）Arctangent: # I = ATAN（# J /# K）（Unit：‰ degree）

G65 H34 P#I Q#J R# k；

Example: G65 H34 P#201 Q#202 R#203； （#201 =ATAN（#202/#203））

Note 1 The unit of （P）～（S）are 1‰ degree.

Note 2 If Q, R required are not specified in operations above, they are defaulted for zero.

Note 3 TRUNC: rounding operation, the decimal portion is abandoned.

3. Transfer command

1）Unconditional transfer

G65 H80 Pn； n： Sequence number

(Example) G65 H80 P120； (To N120 block)

2）Conditional transfer 1 #J.EQ.# K (=)

G65 H81 Pn Q#J R# K； n： Sequence number

(Example) G65 H81 P1000 Q#201 R#202；

When # 201 = #202, it goes to N1000 block; when # 201 ≠ #202, the execution proceeds by sequence.

3）Conditional transfer 2 #J.NE.# K (≠)

G65 H82 Pn Q#J R# K； n： Sequence number

(Example) G65 H82 P1000 Q#201 R#202；

When # 201 ≠ #202, it goes to N1000 block; when # 201 = #202, the execution proceeds by sequence.

4）Conditional transfer 3 #J.GT.# K (＞)

G65 H83 Pn Q#J R# K； n：Sequence number

(Example) G65 H83 P1000 Q#201 R#202；

When # 201 ＞ #202, it goes to N1000 block; when # 201 ≤ #202, the execution proceeds by sequence.

5）Conditional transfer 4 #J.LT.# K (＜ ＝)

G65 H84 Pn Q#J R# K； n： Sequence number

(Example) G65 H84 P1000 Q#201 R#202；

When # 201 ＜ #202, it goes to N1000 block; when # 201 ≥#202, the execution proceeds by sequence.

6）Conditional transfer 5 #J.GE.# K (≥)

G65 H85 Pn Q#J R# K； n： Sequence number

(Example) G65 H85 P1000 Q#201 R#202；

When # 201 ≤#202, it goes to N1000 block; when # 201 ＜ #202, the execution proceeds by sequence.

7）Conditional transfer 6 #J.LE.# K (≤)

G65 H86 Pn Q#J R# K； n： Sequence number

(Example) G65 H86 P1000 Q#201 R#202；

When # 201 ≤#202, it goes to N1000 block; when # 201 ＞ #202, the execution proceeds by sequence.

Note **The sequence number can be specified by variables. Such as G65 H81 P#200 Q#201 R#202；if the conditions are met, it goes to the block whose number is specified by #200.**

4. Logic AND, logic OR and logic NOT instructions

Example:

G65 H01 P#100 Q0;

G65 H01 P#101 Q3;

G65 H01 P#102 Q5;

G65 H11 P#100 Q#101 Q#102;

The binary expression for $5 is 101, 3 for 011, and the operation result is #100=7;

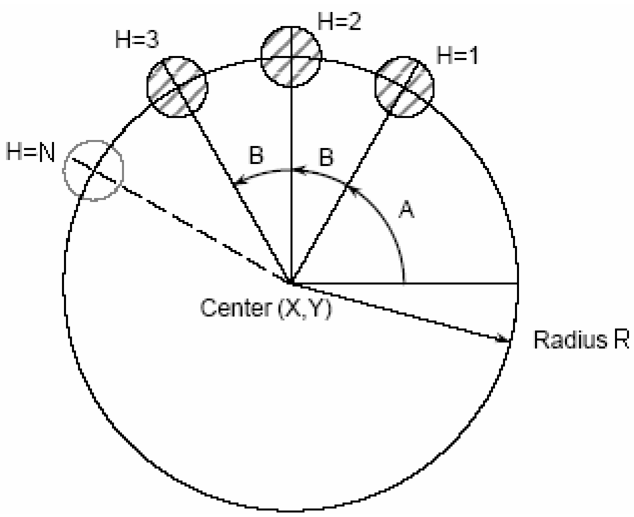
G65 H12 P#100 Q#101 Q#102;

The binary expression for $5 is 101, 3 for 011, and the operation result is #100=1.

4.7.5 Examples for custom macro

1 Bolt hole cycle

To drill N equal-spaced holes on the circumference of the circle with the center regarded as the basic point (X0，Y0) , radius R and the initial angle (A).



X0，Y0 is the coordinate of the basic point in bolt hole cycle.

R：radius, A： initial angle, N： Number. Parameters above using the following variables:

#500： X coordinate value of basic point (X_0)

#501： Y coordinate value of basic point (Y_0)

#502： Radius (R)

#503： Initial angle (A)

#504： Number

If N＞0, for N holes in CCW direction;

If N＜0, for N holes in CW direction.

The following variables are used for the operation in macro.

#100： For the counting of the I hole machining (i)

#101： The final value of the counting(＝｜N｜) (IE)

#102： The angle of the i-thhole (θi)

#103： X coordinate of the i-th hole (Xi)

#104： Y coordinate of the i-th hole (Yi)

The custom macro body can be programmed as following:

```
O9010；
N100 G65 H01 P#100 Q#0；                 i＝0
G65 H22 P#101 Q#504；                    IE＝｜N｜
N200 G65 H04 P#102 Q#100 R3600    00；
G65 H05 P#102 Q#102 R#504；              θi＝A＋360°×I/N
G65 H02 P#102 Q#503 R#102；
G65 H32 P#103 Q#502 R#102；              X i＝X i＋R·COS(θi)
G65 H02 P#103 Q#500 R#103；
G65 H31 P#104 Q#502 R#102；              Y i＝Y i＋R·SIN(θi)
G65 H02 P#104 Q#501 R#104；
G90 G00 X#103 Y#104；                    The i-th hole positioning
M10；                                    Hole drilling M code output.
G65 H02 P#100 Q#100 R1；                 i＝i＋1
G65 H84 P200 Q#100 R#101；           When i＜IE, it goes to N200 to drill IE holes.
M99；
```

Program examples for calling custom macro body is as following:

```
O0010；
G65 H01 P#500 Q100000；                  X0=100MM
G65 H01 P#501 Q-200000；                 Y0=-200MM
G65 H01 P#502 Q100000；                  R=100MM
G65 H01 P#503 Q20000；                   A=20°
G65 H01 P#504 Q12；                      N=12  in CCW direction
G92 X0 Y0 Z0；
M98 P9010；                              Calling custom macro
X0 Y0；
M30；
```

5 Miscellaneous Function M code

The M codes available in this system are listed as following:

	M code	Function
M codes used by program	M30	Program ends and returns to program begining，workpieces added by 1
	M02	Program ends and returns to program begining，workpieces added by 1
	M98	Calling subprogram
	M99	Subprogram ends and returns / execution repeated
M codes controlled by PLC	M00	Program dwell
	M01	Program optional dwell
	M03	Spindle CCW
	M04	Spindle CW
	M05	Spindle stop
	M06	Tool change
	M08	Cooling on
	M09	Cooling off
	M10	A axis release
	M11	A axis clamp
	M16	Tool release
	M17	Tool clamping
	M19	Spindle orientation
	M21	Tool search instruction in retraction
	M22	Tool search instruction for a new tool
	M23	Magazine to spindle instruction
	M24	Magazine retraction instruction
	M29	Rigid tapping
	M32	Lubricating on
	M33	Lubricating off
	M35	Helical chip remover on
	M36	Helical chip remover off
	M40	X axis mirror image

	M41	Y axis mirror image
	M42	Z axis mirror image
	M43	Mirror image cancel
	M44	Spindle blowing on
	M45	Spindle blowing off
	M50	Auto tool change start
	M51	Auto tool change finish
	M53	Tool judging after tool change

When move instruction and miscellaneous function are specified in the same block, the instructions are executed in one of the following two ways:

（1） Simultaneous execution of the move instruction and miscellaneous function instruction.

（2） Executing miscellaneous function instructions on completion of the move instruction execution.

The selection of either sequence depends on the machine tool builder's specification. Refer to the manual by the machine builder for details.

When a numerial value is specified following address M, code sigal and strobe signal are sent to the machine. The machine uses these signals to turn on or off these functions. Usually only one M code can be specified in a block. In some cases, up to three M codes can be specified in a block by bit parameter No.33#7. Some M codes can't be specified simultaneously for the restrictions of the mechanical operation. See the machine manual by the builder for the restrictions to specify multiple M codes for the same block in mechanical operation.

5.1 M codes controlled by PLC

If an M code controlled by PLC is in a same block with a move instruction, they are executed simultaneously.

5.1.1 Forward and reverse rotation instructions（M03, M04）

Instruction: M3（M4） Sx x x；

Explanation: Viewed from the positive Z axis to negative, the spindle counterclockwise (CCW) rotation is defined as forward rotation, clockwise (CW) as reverse rotation.

The instruction of Sx x x specifies the spindle speed, it is the gear in gear mode.

Unit: r/min

When it is controlled by frequency converter, Sx x x specifies the actual speed. e.g. S1000 specifies the spindle to rotate by a speed of 1000r/min.

5.1.2 Spindle stop (M05)

Instruction: M5 When M5 is executed in auto mode, the spindle stops but the speed specified by S instruction is reserved. The deceleration at spindle stop is set by the machine builder. It is usually by energy consumption brake.

5.1.3 Cooling on and off （M08, M09）

Instruction: M8（M9） It is used to control the the cooling pump. If the miscellaneous functions are locked in auto mode, this instruction is not executed.

5.1.4 A axis release and clamping（M10, M11）

Instruction: M10（M11） It is used for A axis release and clamping.

5.1.5 Tool release and clamping（M16, M17）

Instruction: M16（M17） It is used for tool release and clamping.

5.1.6 Spindle orientation（M19）

Instruction: M19 It is specified for spindle orientation which is used for tool change and positioning.

5.1.7 Tool search instruction（M21, M22）

Instruction: M21 It is used to search tool in retraction; M22，it is used to search a new tool for clamping.

5.1.8 Magazine rotation instruction（M23, M24）

Instruction: M23 It is used to rotate the tool magazine to the spindle; M24 It is used to rotate the tool magazine back.

5.1.9 Rigid tapping（M29）

Instruction: M29 It is used for rigid tapping.

5.1.10 Lubricating on and off（M32, M33）

Instruction: M32（M33） It is used to control the lubricating pump. If the miscellaneous functions are locked in auto mode, this instruction is not executed.

5.1.11 Helical chip remover on and off（M35, M36）

Instruction: M35（M36） It is used to control the helical chip remover.

5.1.12 Mirror image instructions（M40, M41, M42, M43）

Instructions: M40, M41, M42, M43 M40 is used to specify X axis mirror image; M41 is used to specify Y axis mirror image; M42 is used to specify Z axis mirror image; M43 is used to cancel mirror image.

5.1.13 Spindle blowing on and off（M44, M45）

Instruction: M44（M45） It is used to control the spindle blowing.

5.1.14 Auto tool change start and end（M50, M51）

Instruction: M50（M51） It is used to control the start and end of auto tool change.

5.1.15 Tool judging after tool change（M53）

Instruction: M53 It is used to judge the tool after the tool change.

5.2 M codes used by program

M codes used by program are classified for main program type and macro type. If the M code for program and the move instruction are in a same block, the move instruction will be executed before M code.

5.2.1 Program end and return （M30, M02）

When M30（M02）in the program is executed in auto mode, the auto mode is cancelled. The blocks following them are not executed and the spindle and cooling stops. The control returns to the beginning of the program while the numbers of the workpiece machined added by 1. Under any situations M30（M02）is regarded as the end of the program execution. M30 can be set by bit parameter NO.33#4 to return to program beginning; M02 can be set by bit parameter NO.33#2 to return to program beginning.

Note 1 M codes such as M00, M01, M02, M30, M98, M99 can't be specified together with other M codes, they must be specified in single blocks, or alarm is issued by system.

Note 2 These M codes include the M codes sent to machine by CNC, and the CNC inner operation codes such as the M code to disable the block pre-reading function. In addition, the M code sent to machine by CNC without inner operation can be specified in a same block.

5.2.2 Program dwell（M00）

In Auto running, automatic operation pauses after a block containing M00 is executed. And the previous modal information will be saved. The automatic operation can be continued by pressing cycle start key, which is equivalent to pressing down feed hold key.

5.2.3 Program optional stop（M01）

Automatic operation is stopped optionally after a block containing M01 is executed. If the "optional stop" on-off is set for ON, M01 is equivalent to M00; if the "optional stop" on-off is set for OFF, M01 is ineffective. See *OPERATION MANUAL* for its operation.

5.2.4 Subprogram calling （M98）

This code is used to call a subprogram in the main program. See the *PROGRAMMING* Section 2.2 *General structure of a program* for details.

5.2.5 Program end and return（M99）

1. In auto mode, if M99 is used at the end of the main program, the control returns to the program beginning to continue automatic operation after the block containing M99 is executed. The blocks followed are not to be executed, and the number of the workpieces machined is not accumulated.
2. If M99 is used at the end of a subprogram, the control returns to the the main program block containing M98 after the block containing M99 is executed.

6 S codes for Spindle Function

The code signal converted to analog signal by S code and the numerical value followed is sent to the machine to control the machine spindle.

S is a modal value.

6.1 Spindle analog control

When the bit parameter NO.1#2 SPT=0, the spindle speed is controlled by analog voltage specified by address S and the numerical value followed. See details about it in *OPERATION* manual.

Format: S_

Explanation:

1 A block can contain only one S code.

2 The spindle speed can be specified directly by address S and a numerical value followed. The unit of it is r/min. e.g. For M3 S300, it means the spindle runs at a speed of 300 r/min.

3 If the move instruction and S code are specified in a same block, they are executed simultaneously.

4 The spindle speed is controlled by S code followed by a numerical value.

6.2 Spindle switch volume control

When the bit parameter NO.1#2 SPT=1, the spindle speed is controlled by switch volume specified by address S and two digits number followed.

Four gears are available in this system as spindle speed is controlled by switch volume. See details on the correspondence of S code and the spindle speed as well as the gears in the manual by machine builder.

Format: S01（S1）；

S02（S2）；

S03（S3）；

S04（S4）；

Explanation:

1 Alarm is issued and the execution stops if S code beyond the codes above is specified in program.

2 For a two-digit S code, the latter two digits are effective if S code is specified with

a four-digit number.

6.3 Constant surface speed control (G96/G97)

Format:

Constant surface speed control instruction

G96 S_ Surface speed（m/min or feet/min）

Constant surface speed control cancel instruction

G97 S_ Spindle speed (rmp)

Constant surface speed controlled axis instruction

G96 Pn P1 X axis；P2 Y axis；P3 Z axis；P4 4th axis

Max. spindle speed clamping

G92 S_ S specifies the max. spindle speed（rmp）

Function： The number following S is used to specify the surface speed (relative speed of tool and workpiece). The spindle is rotated so that the surface speed is constant regardless of the tool position.

Explanation:

1 G96 is a modal instruction. After it is specified, the program enters the constant surface speed control mode and the S value specified is assumed as a surface speed.

2 A G96 instruction must specify the axis along which constant surface speed control is applied. It can be cancelled by G97 instruction.

3 To execute the constant surface speed control, it is necessary to set the workpiece coordinate system, and the coordinate value at the center of the rotary axis becomes zero.

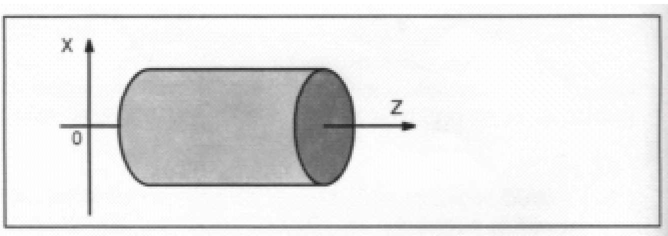


Fig. 6-1-1 Workpiece coordinate system for constant surface speed control

4 When constant surface speed control is applied, if a spindle speed higher than the value specified in G 92 S_, it is clamped at the maximum spindle speed. When the power is switched on, the maximum spindle speed is not yet set, the S in G96 is regarded as zero till M3 or M4 appears in program.

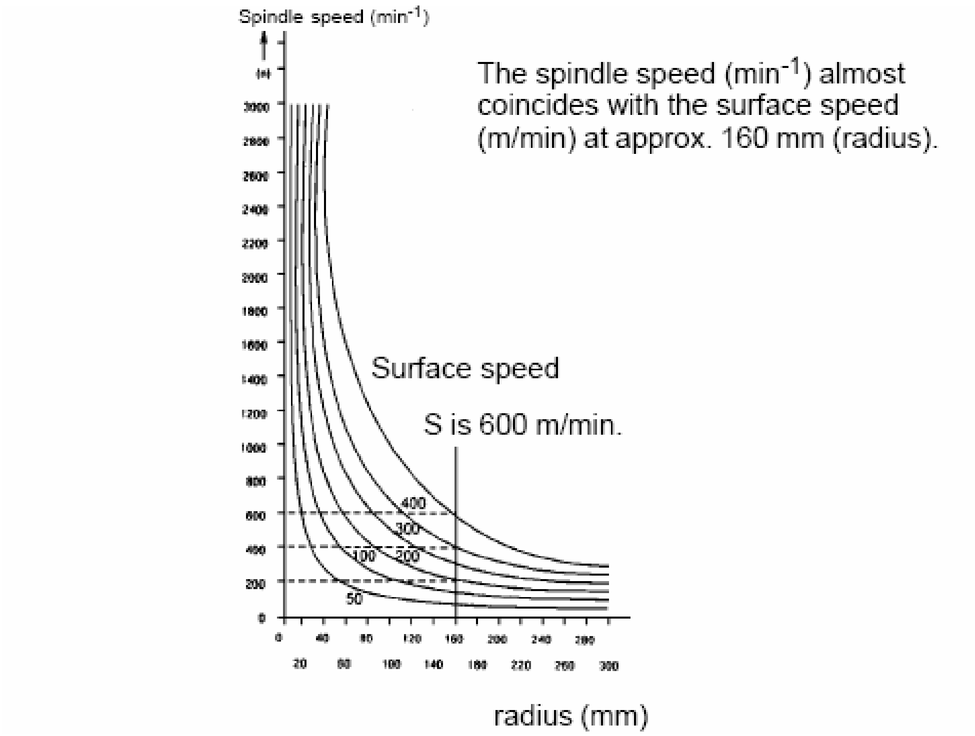
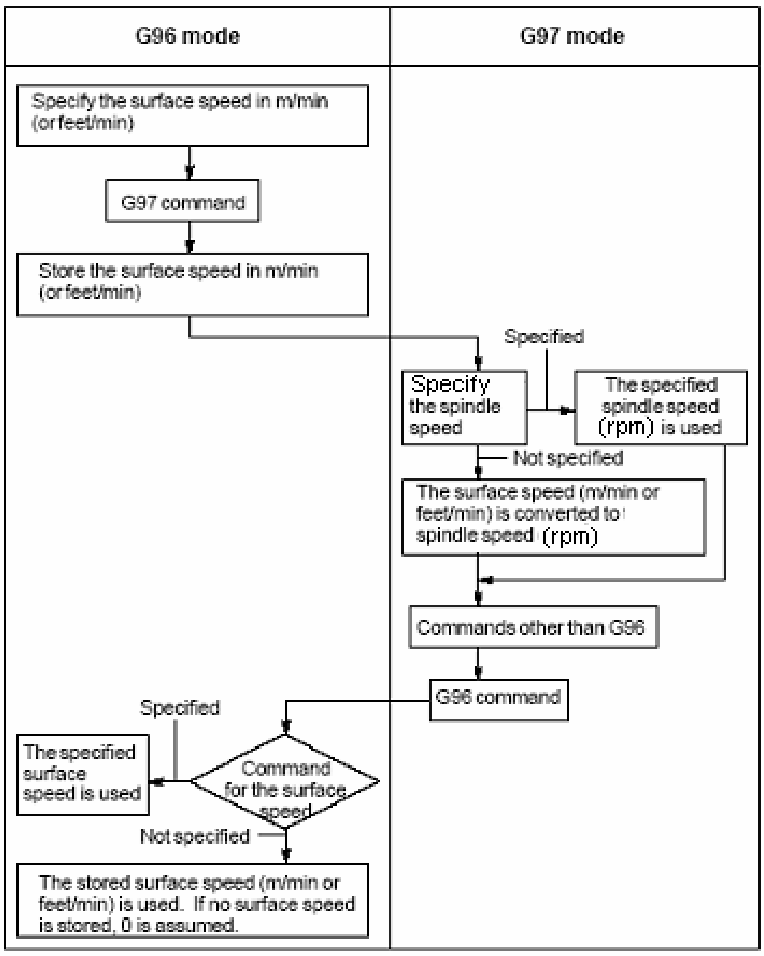


Fig. 6-6-2 Relations of workpiece radius, spindle speed and surface speed

5 Surface speed is specified in G96 mode



Restriction

1 Because the response problem in the servo system may not be considered when the spindle speed changes, while the constant surface speed is still effective during threading, so it is recommended to cancel the constant surface speed by G97 before threading.

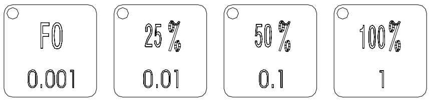
2 In a traverse block specified by G00, the constant surface speed control is not made by calculating the surface speed by a transient change of the tool position, but is made by calculating the surface speed based on the position at the end point of the traverse block, on the condition that cutting is not performed during traverse. Therefore, the constant surface cutting speed is not needed.

7 Feed Functions F code

The feed functions are used to control the feedrate of the tool. They are used as following:

7.1 Traverse

G00 instruction is used for rapid positioning. And the traverse speed can be set by number parameter P88~P92. Override can be applied to a traverse speed by the OVERRIDE adjusting keys on the operator panel as follows:



In which, F0 is set by number parameter P93.

The acceleration of rapid positioning （G0）can be set by number parameter P105~124. It can be properly set depending on the machine and the motor response.

Note　A feedrate F instruction is ineffective even it is specified in a block containing G00 and the system performs positioning at the speed specified by G0.

7.2 Cutting feedrate

Feedrate of linear interpolation(G01), circular interpolation(G02，G03) are specified with the numbers after F code. The unit of it is mm/min. The tool moves by the feedrate programmed. Override can be applied to feedrate using the override key on the operator panel. （Override range：0％～150%）In order to prevent mechanical vibration, acceleration/deceleration can be automatically applied at the beginning and the end of the tool movement respectively. The acceleration can be set by the number parameter P125~P128.

In non-forecast mode, the maximum cutting feedrate is set by number parameter P94 and in forecast mode, it is set by number parameter P96. If the feedrate is more than that, use the feedrate set by that parameter.

In non-forecast mode, the minimum cutting feedrate is set by number parameter P95 and in forecast mode, it is set by parameter P97. If the feedrate is less than that, use the feedrate set by that parameter.

The cutting feedrate in auto mode at power-on is set by number parameter **P87.**

The cutting feedrate can be specified by the following two types:

1、Feed per minute（G94）：it is used to specify the feed amount per minute after F code.

2、Feed per revolution（G95）：it is used to specify the feed amount per revolution after

F code.

7.2.1 Feed per minute（G94）

Format: G94 F_

Function: It specifies the tool feed amount in a minute. Unit：mm/min or inch/min。

Explanation:

1、After specifying G94 (in feed per minute mode), the feed amount of the tool per minute is directly specified by a number after F.

2、G94 is a modal code. Once specified, it is effective till G95 is specified. The default at power-on is feed per minute mode.

3、An override from 0% to 150% can be applied to feed per minute with the override key on the operator panel.

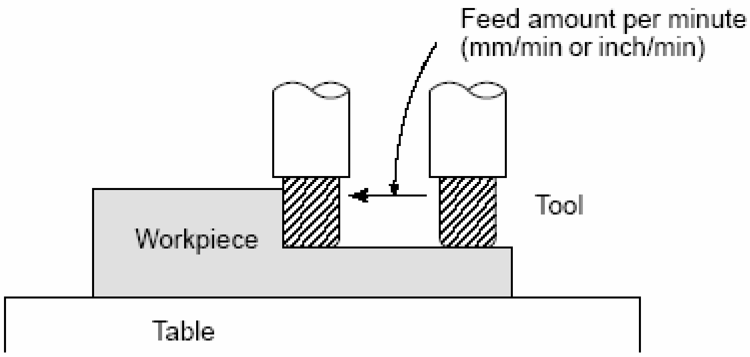


Fig. 7-2-1-1 Feed per minute

Restriction

Feed per minute mode can't be applied for some instructions such as threading.

7.2.2 Feed per revolution（G95）

Format: G95 F_

Function: It specifies the tool feed amount in a revolution. Unit：mm/rev or inch/rev

Explanation:

1 After specifying G95 (feed per revolution mode), the feed amount of the tool per revolution is directly specified by a number after F.

2 G95 is a modal code. Once specified, it is effective till G94 is specified.

3 An override from 0% to 150% can be applied to feed per revolution with the override key on the operator panel.

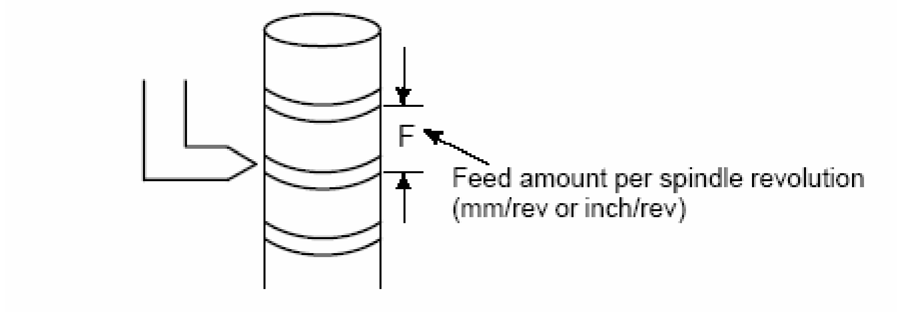
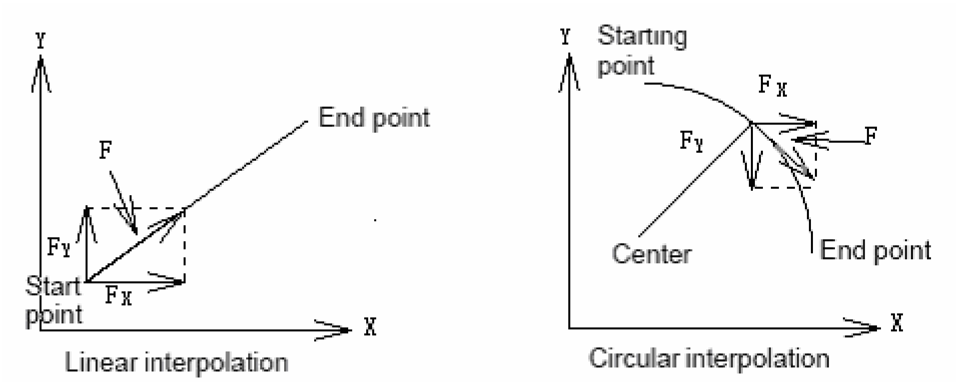


Fig. 7-2-2-1 Feed per revolution

Note **Feedrate fluctuation may occur if the spindle speed is low. The slower the spindle rotates, the more frequently the feedrate fluctuation occurs.**

7.3 Tangential speed control

Usually the cutting feed of the tool is made by controlling the speed along the tangent of the contour path to reach a value specified.



F: The speed along the tangent : $F = \sqrt{F_x^2 + F_y^2 + F_z^2}$

F_X: The speed along X axis

F_Y: The speed along Y axis

F_Z: The speed along Z axis

7.4 Feedrate override keys

The feedrate in JOG mode and AUTO mode can be overriden by the override keys on the operator panel. The override range from 0～150%(16 gears with 10% per gear). In AUTO mode, if the feedrate override is adjusted for zero, the feeding is stopped by the system with 0 cutting override displayed. The execution is continued if the override is readjusted.

7.5 Auto acceleration/deceleration

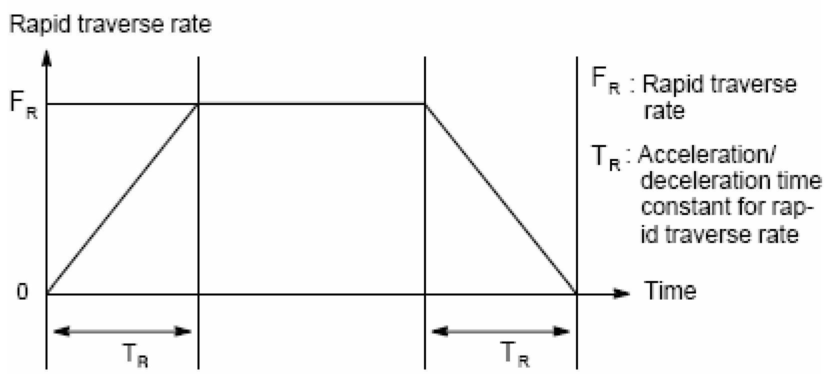
The stable start and stop can be done by auto acceleration/deceleration at the beginning and the end of the moving controlled by the system motor. And the auto acceleration/deceleration can also be done when the moving speed is changed, so the speed can be changed steadily. Therefore the acceleration/deceleration needn't to be considered for programming.

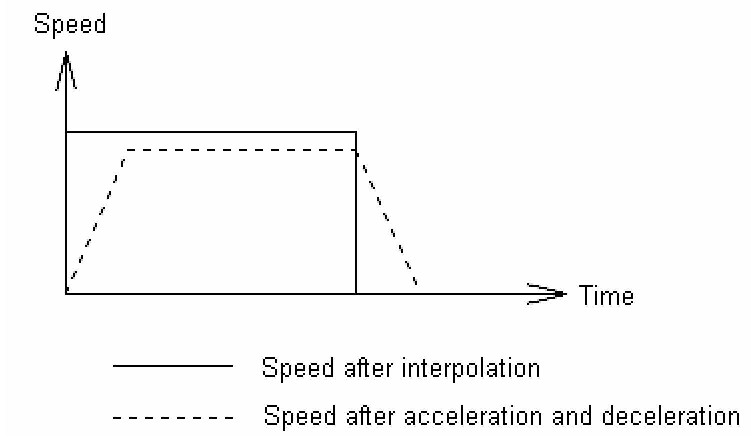
Rapid traverse: Pre-acceleration/deceleration（0：linear type；1：S type）hind acceleration/deceleration（0：linear type；1：exponential type）

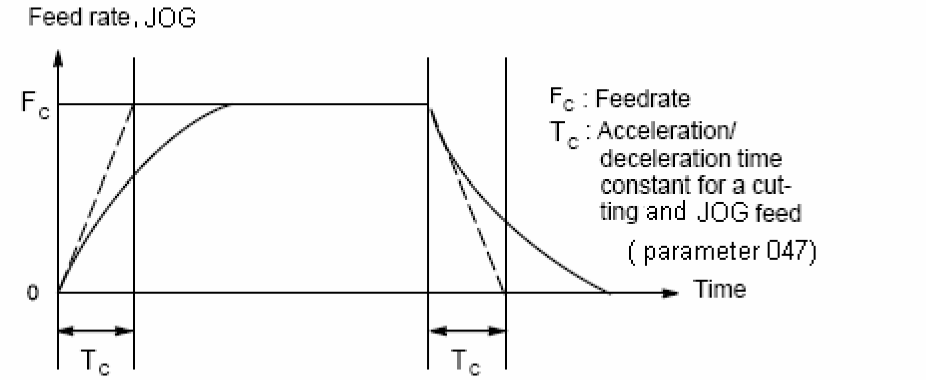
Cutting feed: Pre-acceleration/deceleration（0：linear type；1：S type）hind acceleration/deceleration（0：linear type；1：exponential type）

JOG feed: Hind acceleration/deceleration（0：linear type；1：exponential type）

(Set the universal time constant for each axis by parameters)

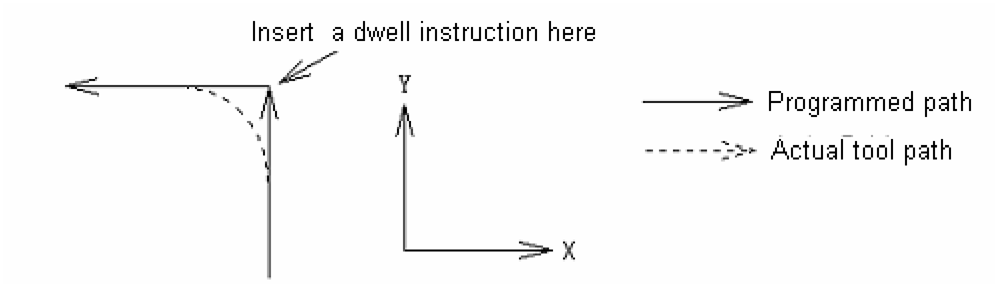






7.6 Acceleration/deceleration for corner of a block

For example: if Y axis moves in a block, and X axis moves in the block following, the tool path is as following during the Y axis deceleration and the X axis acceleration:



If the dwell (exact stop) instruction is inserted, the tool moves along the real line as in above figure by the program. Otherwise the bigger the cutting feedrate is, or the longer the time constant of the acceleration/deceleration, the bigger the arc at the corner is. For circular instruction, the actual arc radius of the tool path is smaller than the radius given by the program. Under the condition allowed by mechanical system, reduce the time constant of the acceleration/deceleration as far as possible to decrease the error at the corner.

8 Tool Function

8.1 Tool function

By specifying a numerical value (up to128) following address T, tools can be selected on the machine.

Only one T code can be specified in a block. Refer to the machine builder’s manual for the number with address T and the corresponding machine operation of T code.

When a move instruction and a T code are specified in a same block, the instructions are executed in the following two ways:

1 Simultaneous execution of the move and T instructions.

2 Executing T instruction upon completion of the move instruction.

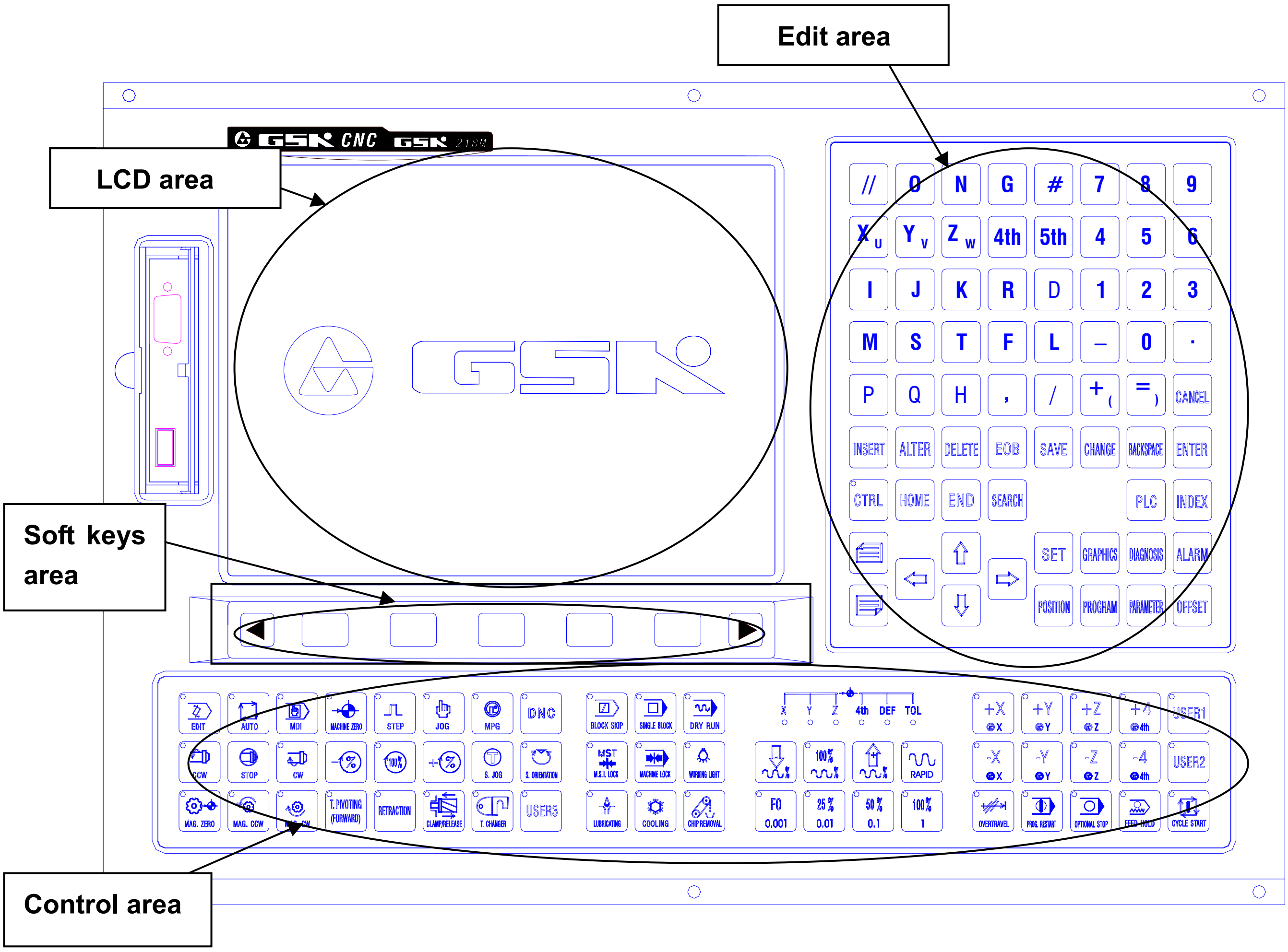
The selection of either 1 or 2 depends on the machine builder’s specifications. Refer to the machine builder’s manual for details.

Ⅲ OPERATION

1 Operator Panel

1.1 Panel layout

The 218M machine center has an integrated operator panel, which is comprised by LCD area, edit area, interface display area and machine control area. The layout of it is shown as following:

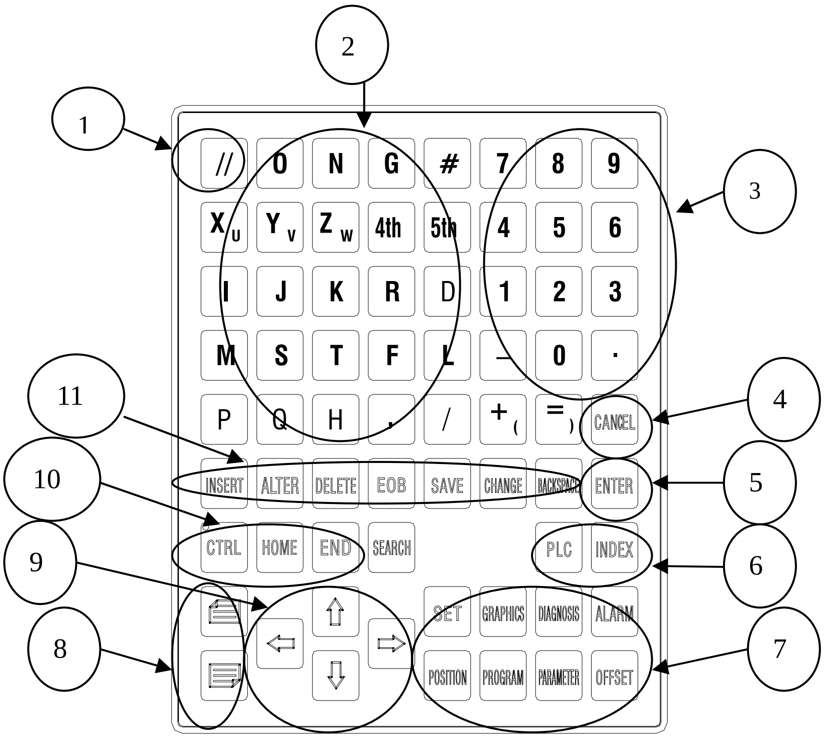


1.2 Explanation of the panel function

1.2.1 LCD area

The display area of this system is applied with a 10.4 inch chromatic LCD that has 640×480 resolution.

1.2.2 Edit area

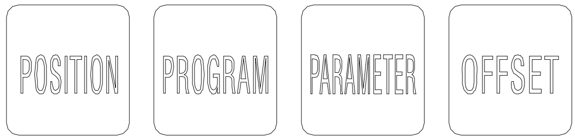


The edit keyboard divided for 11 small areas is explained as follows:

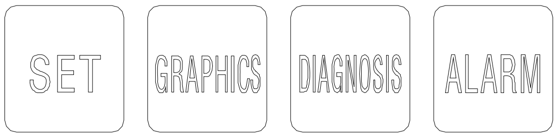
Number	Name	Explanation
1	Reset key	Press this key to reset the system，feed and stop output
2	Address key	Press these keys to input address
3	Numerical keys	Press these keys to input numerical numbers
4	Cancel key	Press this key to delete the characters entered（not saved in buffer）
5	Enter key	Press this key to save the data into the buffer when numerical keys or address keys are pressed
6	Help key	Press this key to display the help directory and PMC ladder
7	Screen operation keys	Press any key of them to enter the corresponding interface (introduced below)
8	Page keys	Press these keys to change the page on the screen in the same display mode
9	Cursor keys	Press these keys to move the cursor in four directions
10	Edit keys	Press these keys to move the cursor to the beginning or the end of the lines and the programs
11	Edit keys	Press these keys to insert, modify and delete the programs, words etc. in program edit

1.2.3 Screen operation keys

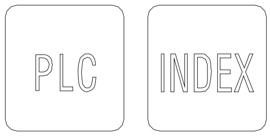
9 page display keys and 1 help display key are laid out in this operator panel, which are as following:



Position page Program page Parameter page Offset page



Setting page Graphic page Diagnosis page Alarm page



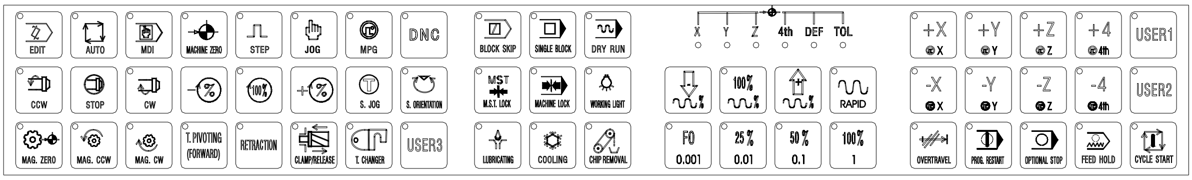
PLC page Index page

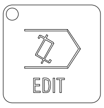
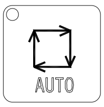
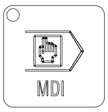
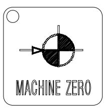
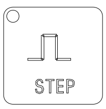
Name	Explanation	Remark
Position page	Press this key to enter position page	Relative coordinate, absolute coordinate of current point, comprehensive display, program monitoring display by changing soft keys
Program page	Press this key to enter program page	Program,MDI, current/mode, current/time, program directory display by changing soft keys, and program directory is switched over by page keys
Parameter page	Press this key to enter parameter page	Bit parameter, number parameter, and macro variable page display by changing soft keys to view and modify the parameter
Offset page	Press this key to enter offset page	2 pages, used to set the tool length and radius and the pitch error compensation of each axis by changing soft keys
Setting page	Press this key to enter program page	2 pages, setting, parameter on-off, coordinate, panel, servo system, data and password setting display by changing soft keys
Graphic page	Press this key to enter setting page	For graphic parameter, graphic display page and the graphic center, dimension, ratio and display page setting by changing soft keys
Diagnosis page	Press this key to enter diagnosis page	To view the I/O signals in the system by changing soft keys
Alarm page	Press this key to enter alarm page	To view alarm display pages by changing soft keys

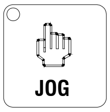
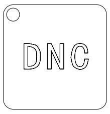
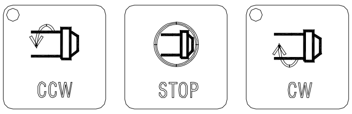
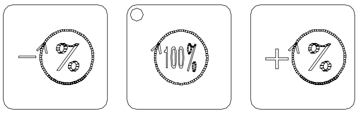
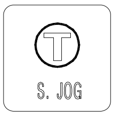
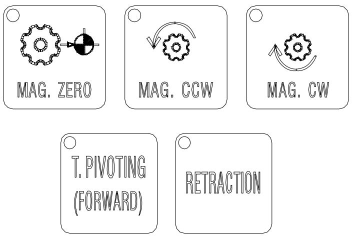
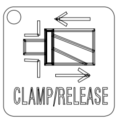
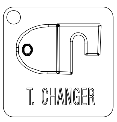
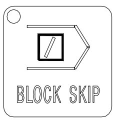
PMC page	Press this key to enter PMC page	To view the corresponding system version, the I/O configuration and modify PLC ladder in Edit mode by changing soft keys
Index page	Press this key to enter index page	To view the corresponding system message by changing soft keys

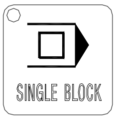
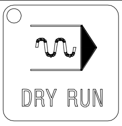
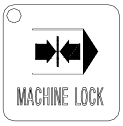
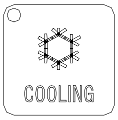
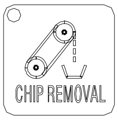
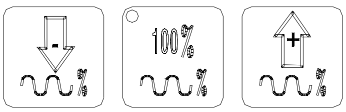
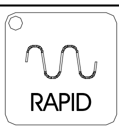
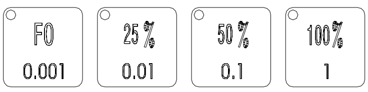
Note **The pages by soft keys above can also be displayed by continuously pressing the corresponding functional keys. Refer to Chapter 3 in this manual for the details of these pages.**

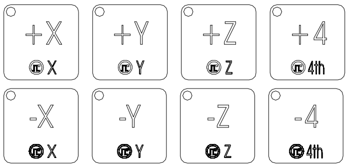
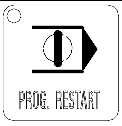
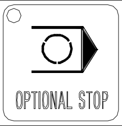
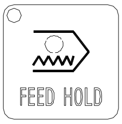
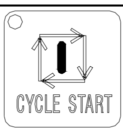
1.2.4 Control area



Keys	Name	Function explanation	Remarks and operation explanation
	Edit mode key	To enter Edit mode	Switching to Edit mode in Auto mode, system slowing down to stop if current block is executed
	Auto mode key	To enter Auto mode	In this mode, internal memory program is selected
	MDI mode key	To enter MDI mode	Switching to MDI mode in Auto mode, system slowing down to stop if current block is executed
	Machine zero mode key	To enter Machine zero mode	Switching to Machine zero mode in Auto mode, system immediately slowing down to stop
	Step mode key	To enter Step mode	Switching to Step mode in Auto mode, system immediately slowing down to stop

	JOG mode key	To enter JOG mode	Switch to JOG mode in Auto mode, system immediately slowing down to stop
	MPG mode key	To enter MPG mode	Switching to MPG mode in Auto mode, system immediately slowing down to stop
	DNC mode key	To enter DNC mode	Switching to DNC mode in Auto mode, system slowing down to stop if current block is executed
	Spindle control keys	Spindle CCW Spindle stop Spindle CW	MPG mode, Step mode, JOG mode
	Spindle override keys	Spindle speed adjusting(spindle speed analog control effective)	Any mode
	Spindle JOG key	Spindle JOG on/off	JOG mode
	Spindle Orientation key	Spindle orientation on/off	JOG mode
	Tool magazine operation keys	Tool magazine operation on/off	JOG mode
	Manual tool clamp/ release key	For manual tool clamping / release	JOG mode
	Manual tool change key	For manual tool change	JOG mode
	Block Skip key	For block preceding with“/”sign skipping, if it is set for on, the indicator lights up	Auto mode, MDI mode, DNC mode

	Single Block key	For switching of program block/blocks, if it is on, the indicator lights up	Auto mode, MDI mode, DNC mode
	Dry Run key	If it is effective, the indicator lights up	Auto mode, MDI mode, DNC mode
	M.S.T. Lock key	If the miscellaneous function is on, its indicator lights up and M,S,T functions are ineffective	Auto mode, MDI mode, DNC mode
	Machine Lock key	If it is on, its indicator lights up, and the axis output is ineffective.	Auto mode, MDI mode, Machine zero, MPG mode, Step mode, JOG mode, DNC mode
	Working Light key	For machine working light on/off	Any mode
	Lubricating key	For machine lubrication on/off	Any mode
	Cooling key	For coolant on/off	Any mode
	Chip Removal key	Chip removal switch	Any mode
	Feedrate Override keys	For adjustment of the feedrate	Auto mode, MDI mode, Edit mode, Machine zero, MPG mode, Step mode, JOG mode, DNC mode
	Rapid traverse key	For rapid traverse on/off	Any mode
	Rapid, Step, and MPG override keys	For rapid override, manual single step, MPG override selection	Auto mode, MDI mode, Machine zero, MPG mode, Step mode, JOG mode, DNC mode

	Manual feeding keys	For positive/negative moving of X, Y, Z 4th axis in JOG, Step mode, and the positive direction of the axis is by MPG	Machine zero, MPG mode, Step mode, JOG mode
	Overtravel release key	Alarm occuring if machine reaches the hard limit, pressing this key with indicator lighting up to move reversely till the indicator gone out	JOG mode
	Program Restart key	Cursor moving to the beginning of the starting block to restart the machine, also for rapid program check	Auto mode, MDI mode, DNC mode
	Optional Stop key	For stop of the program with“M01”	Auto mode, MDI mode, DNC mode
	Feed Hold key	Auto running stops by pressing this key	Auto mode, MDI mode, DNC mode
	Cycle Start key	Auto running begins by pressing this key	Auto mode, MDI mode, DNC mode

2 System Power On/Off and Safety Operations

2.1 System power on

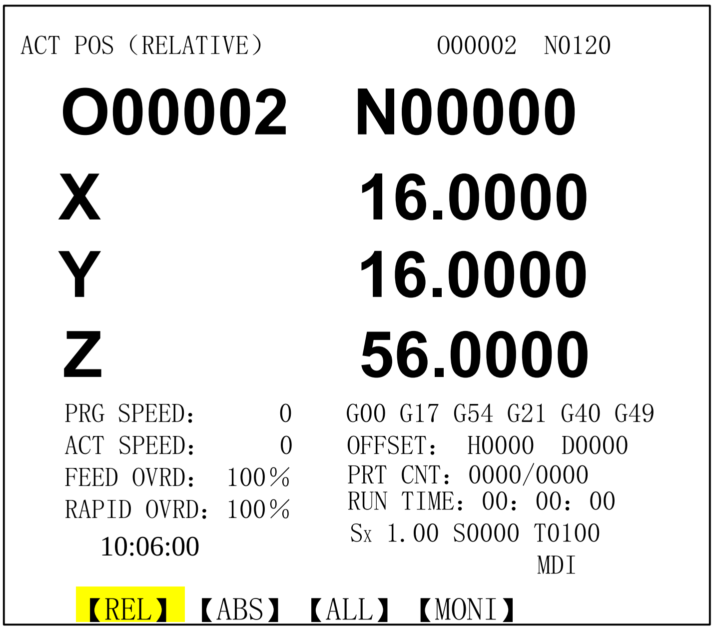
Before this GSK218M is powered on, ensure that:

1 The machine is normal.

2 The voltage of the power supply conforms to the requirement of the machine.

3 The wiring is correct and reliable.

The current position (relative) is displayed after system normalty check and initiation are finished.



2.2 System power off

Before power is off, ensure that:

1 The X, Y, Z axis of the CNC is at halt;

2 Miscellaneous functions(spindle, pump etc.) are off.

3 CNC power is cut off prior to cutting off machine power.

Check the following items before power-off:

1 The LED indicating the cycle start on the operator panel is off.

2 All the movable parts of the CNC machine tool is at halt.

3 Press POWER OFF button to turn off the power.

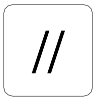
Cut off power at emergency

The power should be cut off immediately to prevent from incident in emergency situation during the machine running. And the zero return and tool setting should be performed again because of the error between the system coordinate and actual coordinate of the position after power-off.

Note Refer to the machine builder's manual for cutting off the machine power.

2.3 Safety operations

2.3.1 Reset operation

The system is in reset mode after pressing  key:

1、All axes movement stops;

2、The M,S functions are ineffective;

3、Whether to save G codes or not after modifying bit parameters N35.1～N35.7 and N36.0～N36.7 and resetting;

4、Whether to clear F, H, D codes or not after modifying bit parameters N34.7 and resetting;

5、Whether to delete the program or not after modifying bit parameters N28.7 and resetting in MDI mode;

6、Whether to clear the executing DNC program display or not after modifying bit parameters N23.2 and resetting;

7、Whether to cancel local coordinate or not after modifying bit parameters N10.3 and resetting;

8、Whether to clear the macro common variables #0-#99 or not after modifying bit parameters N52.6 and resetting;

It can be used in system abnormal output and coordinate axis action.

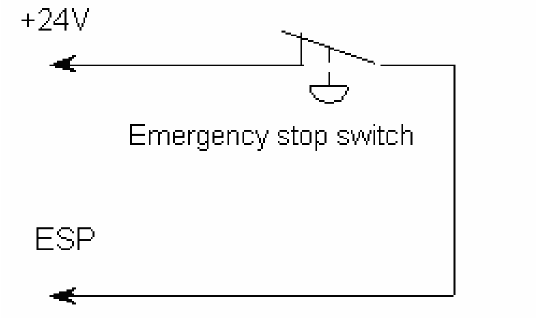
2.3.2 Emergency stop

If the emergency button is pressed during machine running, the system enters into emergency status and the maching movement is stopped immediately. All the outputs such as the spindle running, coolant are also cut off. If the emergency button is released (varying by machine builders, usually the button bumps up by rotating it left-handedly), the emergency is cancelled.

Note 1 Ensure the cause of the fault is eliminated before the emergency is cancelled.

Note 2 Perform the reference point return operation to ensure the position coordinate after the emergency is cancelled.

The common emergency is a normal-close signal. When the trigger point is broken off, the system enters into emergency status and the machine stops immediately. The wiring of the emergency signal circuit is as following:



2.3.3 Feed hold

key can be pressed during the machine running to make the running to dwell. But in rigid tapping, cycle running, the machine dwells till current instruction is executed.

2.4 Cycle start and feed hold

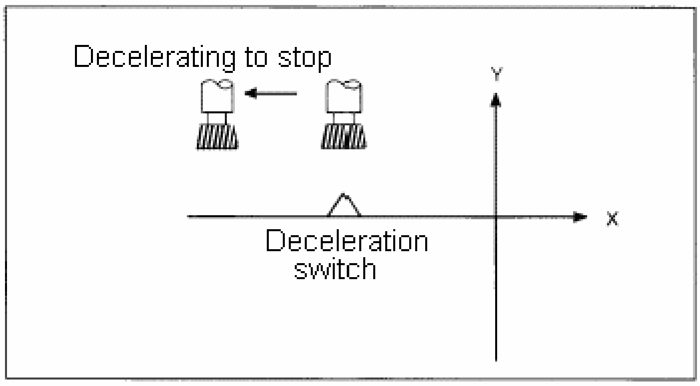
The Cycle Start and Feed Hold keys are used for the program start and dwell operations in Auto mode, MDI mode and DNC modes. The external start and dwell are set by bit parameter No.59.7.7, and they can also be set by modifying the address K5.1 of the PLC. These two methods are equivalent.

2.5 Overtravel protection

Overtravel protection must be employed to prevent the damage to the machine due to the overtravel of the X, Y, or Z axis.

2.5.1 Hardware overtravel protection

The overtravel limit switches are fixed at the positive and negative maximum stroke of the machine X, Y, Z axis respectively. If the overtravel occurs, the moving axis slows down and stops after it touches the limit switch. And the overtravel alarm is issued.



Explanation:

Overtravel in Auto mode

In Auto mode, if the tool contacts the stroke limit switch during moving along an axis, all the axes movement are slowed down to stop with the overtravel alarm being issued. The program execution is stopped at the block where the overtravel occurs.

Overtravel in JOG mode

In JOG mode, any axis contacts the stroke limit switch, all axes will slow down immediately and stop.

2.5.2 Software overtravel protection

The software strokes of the machine are set by the number parameters NO.66～NO.75 (appendix 1), referring to machine coordinate values.

Overtravel alarm occurs if the machine position (coordinate) exceeds the setting software stroke. The alarm issued before or after overtravel for software limit overtravel is set by bit parameter No.11.7. During the overtravel alarm, move the axis reversely in JOG mode, the alarm will be cancelled after the axis is moved out of the overtravel range.

2.5.3 Release of the overtravel alarm

The method for overtravel alarm is: in JOG mode, press the OVERTRAVEL key on the panel, then move the axis out reversely (for positive overtravel, move negatively; for negative overtravel, move positively).

2.6 Storage stroke detection

By storage stroke detection 1 and 2, the system can specify 3 areas that the tool can’ t enter.

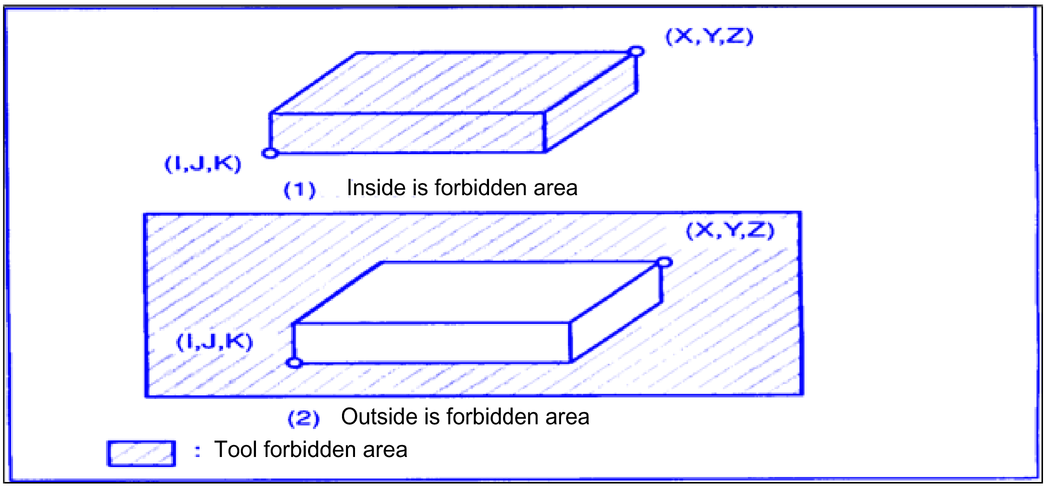


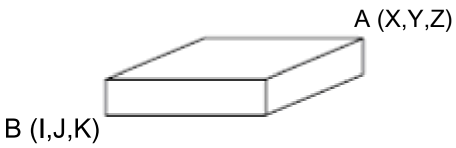
Fig. 2-6-1 Stroke detection

When the tool goes beyond the stroke, alarm is issued and the machine decelerates and stops.

When the tool enters the forbidden area with alarm issued, the tool may move in the reverse direction that the tool enters.

Explanation:

1. Storage stroke detection 1: its boundary is set by number parameter P66~P75, the outside of this area is forbidden area, which is usually set for the machine maximum travel by the machine builder.
2. Storage stroke detection 2: its boundary is set by number parameter P76~P85 or program instructions, the inside or outside of this area can be set for a forbidden area, which is set by bit parameter No.11#0.
   a) When the forbidden area is set by parameters: the A and B points in the following figure must be set.



X > I, Y > J, Z > K

X-I > ζ(Min. command incremental unit)

Y-J > ζ(Min. command incremental unit)

Z-K > ζ(Min. command incremental unit)

ζ(mm)= F / 7500

F= rapid traverse speed(mm/min)

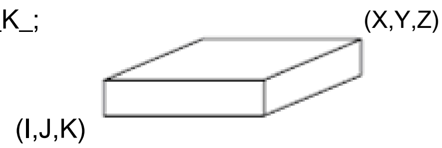
Fig. 2-6-2 To use parameters set up or change forbidden area

Even the coordinate sequence of the 2 points is wrongly given in storage stroke detection 2, a rectangular forbidden area can also be formed by these 2 points taken as the vertex.

When the forbidden area is set by number parameter P76~P85, the distance (output increment) in the machine coordinate system must be given by a min. command incremental unit via data.

b) When the forbidden area is specified by program instructions: by G12 it forbids the tool to enter forbidden area; by G13 it allows the tool to enter the forbidden area. Each G12 or G13 code must be specified by a single block in program. The following commands are used for setting up or changing the forbidden area.

G12 X_Y_Z_I_J_K_;



X > I, Y > J, Z > K

X-I > ζ(**Min. command incremental unit)**

Y-J > ζ(**Min. command incremental unit)**

Z-K > ζ(**Min. command incremental unit)**

ζ(mm)= F / 7500

F= rapid traverse speed(mm/min)

Fig.2-6-3 To set up or change forbidden area by program

If the distance (input increment) in the machine coordinate system is specified by min. input incremental unit via G12, the programming data will be converted to the value of min. command unit by min. incremental unit.And the value will be used by the parameter.

3. Detection point in the forbidden area: prior to the programming for forbidden area, please confirm the detection point(the top of the tool nose or tool collet). As is shown in Fig.2-6-4,if the detection point is A(tool nose), the distance “a” should be used as the data for storage detection; if the detection point is B(tool collet), the distance “b” should be used as the data for storage detection. When the detection point is A (tool nose), and the tool length varies with the tool, the forbidden area should be set up according to the longest tool, as such may ensure the safe running.

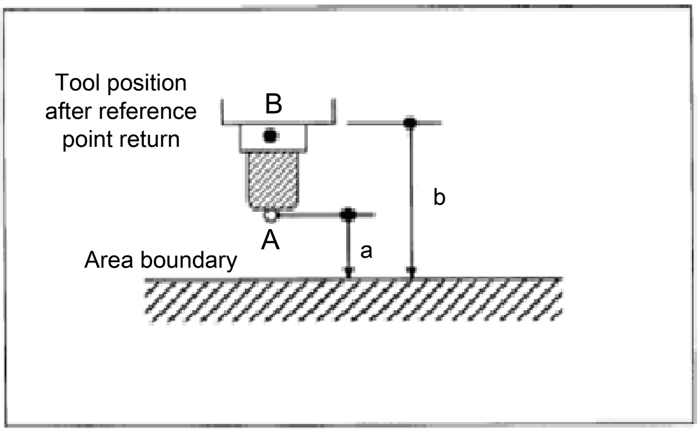


Fig. 2-6-4 To setup forbidden area

4. The overlapping of tool forbidden areas: The forbidden areas can be set by overlapping, as is shown in following figure:

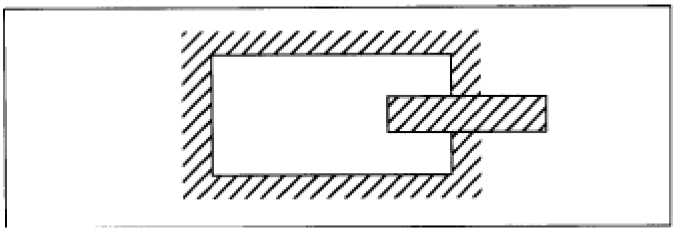


Fig. 2-6-5 The overlapped forbidden areas

Unnecessary limit should be set beyond the machine stroke.

5. Forbidden area to be effective: the forbidden area boundary takes effective immediately after the manual reference point return or auto reference point return by G28 instruction at power on.

   If the reference point is within the forbidden area at power on, an alarm will be issued (effective only for G12 of storage stroke 2).

6. Alarm cancellation: if the tool enters the forbidden area with an alarm being issued, it can only move reversely.

   To cancel alarm, move the tool reversely till it exits the forbidden area and the system resets. If the alarm is cancelled, the tool can move forward or backword freely.

7. G13 changed by G12 in forbidden area: the following results may occur if G13 changed by G12 in forbidden area:

   1） If the forbidden area is interior one, an alarm will be issued for next move.

   2） If the forbidden area is exterior one, an alarm will be issued immediately.

3 Interface Display as well as Data Modification and Setting

3.1 Position display

3.1.1 Four types of position display

Press 

key to enter the position page that includes four types:【REL】,【ABS】,【All】,【MONI】. They can be viewed by the corresponding soft keys, as is shown in the following:

1) Relative mode: It displays the coordinate of the current tool in relative coordinate system by pressing【REL】soft key, called "relative" in following (see Fig.3-1-1):

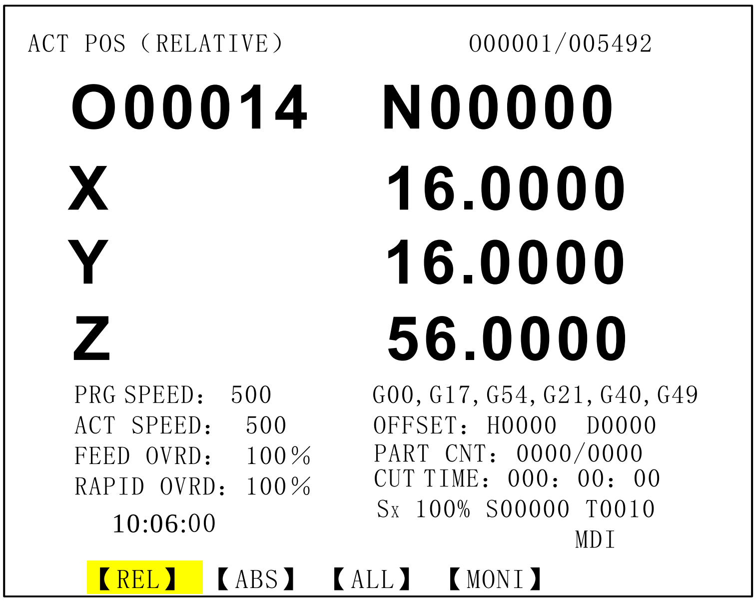


Fig. 3-1-1

Clearing steps of relative coordinate system: press <X> key, as X axis blinks, press <CANCEL> key to clear the coordinate system, and the same for Y and Z axes.

Setup steps of relative coordinate system: press <X> key, as X axis blinks, input the setting data then press <ENTER> key to enter the data into coordinate system.

2) Absolute mode: It displays the absolute coordinate of the current tool by pressing 【ABS】 soft key, which is called "absolute" in following(see Fig.3-1-2):

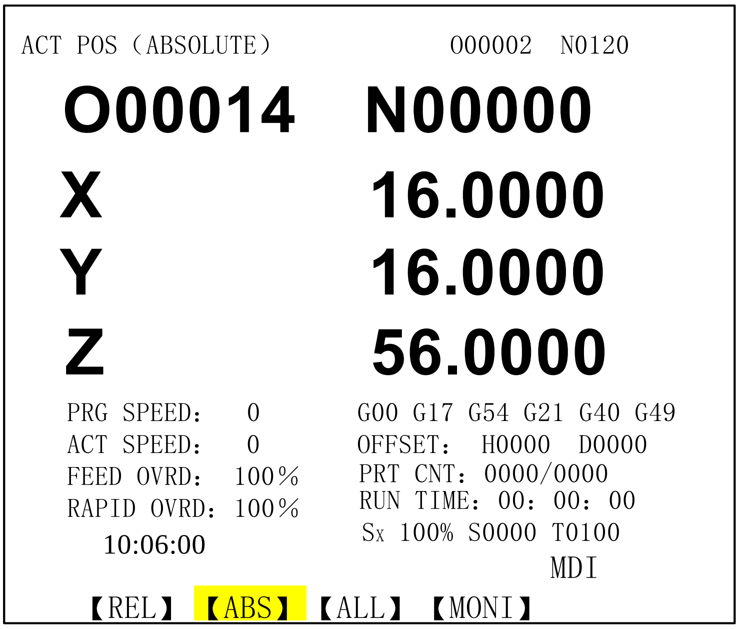


Fig. 3-1-2

3) All mode

It enters【ALL】mode by pressing【ALL】soft key, the coordinates in the following coordinate system can be displayed together:

(A) The position in relative coordinate system;

(B) The position in absolute coordinate system;

(C) The position in machine coordinate system;

(D) The offset amount in Handle interruption (displacement);

(E) Subspeed;

(F) Remaining distance (only displayed in Auto, MDI, DNC mode)

The display is as follows（Fig.3-1-3）:

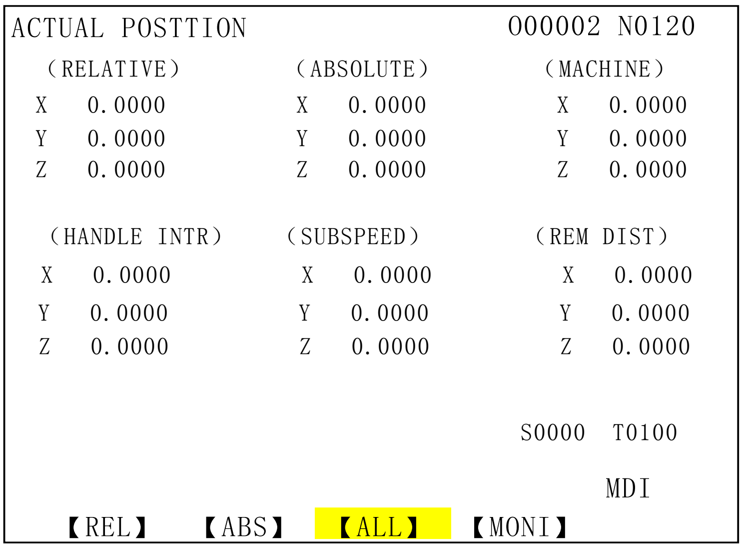


Fig. 3-1-3

4）Monitoring mode

It enters 【MONI】 mode by pressing 【MONI】soft key, in this mode the absolute coordinate, relative coordinate of the current position as well as the current running program modal message and blocks can be displayed together: (See Fig. 3-1-4)

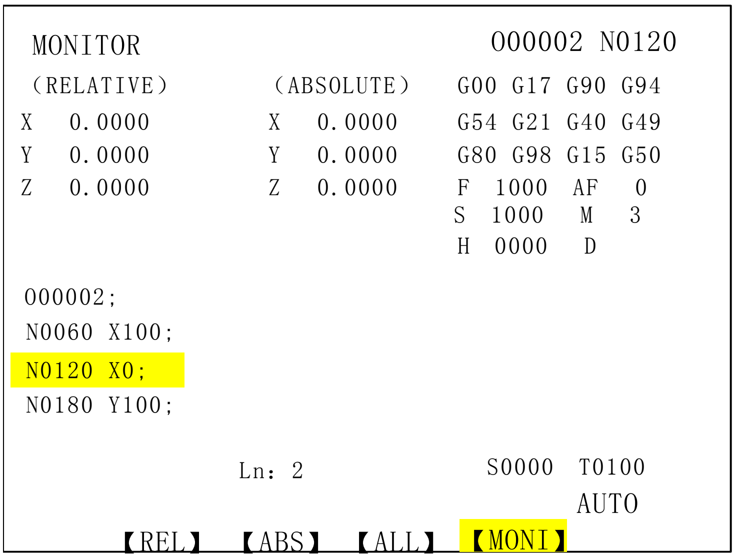


Fig. 3-1-4

Note 1 The display in 【MONI】mode can be set by BIT6 of the parameter NO.023. when BIT6=0, the machine coordinate but the modal instruction is displayed at the original position.

Note 2 In <MACHINE ZERO>, <STEP>,<JOG>,<MPG>modes, the intermediate coordinate system is a relative one; while in <AUTO>,<MDI>,<DNC> modes, it is a remaining distance.

3.1.2 The display of the run time, part count, programming speed and override, actual speed etc.

The programming speed, the actual speed, feedrate and rapid override, G codes, tool offset, run time, part number, spindle override, spindle speed, tools etc. can be displayed in <POSITION> absolute or relative mode(see Fig.3-1-5).

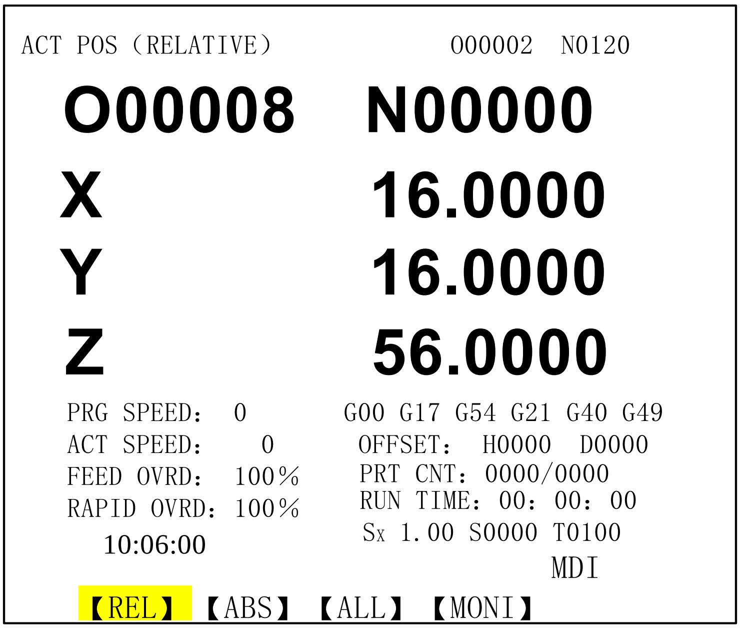


Fig. 3-1-5

The meaning of them is as following:

Programming speed: Speed specified by F code

Actual speed: The actual cutting rate overriden

Feedrate override: Feed override selected by feedrate override keys

Rapid override: Rapid override selected by rapid override keys

G codes: The value of the G code in block being executed

Tool offset: H0000, tool length compensation for current program;D0000, tool radius compensation for current program

Parts count: Plusing 1 when M30 is executed

Run time: Time counting start if Auto run starts, time units are hour, minute and second

S_x : Spindle override for spindle speed

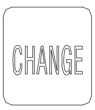
S00000: Actual feedback speed of spindle encoder

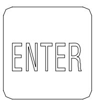
T0000: Tool number specified by T code in program

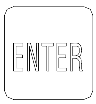
Note The parts counting is reserved after the power-down.

The clearing ways:

1）Switchover to POSITION mode.

2）Press 

key, the cursor locates to the PRT CNT item, input data and press 

key to confirm; if 

key is pressed, the parts counting will be cleared.

3）Shift UP or DOWN key to RUN TIME.

4）Press ENTER key to clear the RUN TIME.

Note 1 To display the actual spindle speed, the encoder must be applied to the spindle.

Note 2 The actual speed= the programming speed F × override; in G00 mode the axes speeds are set by number parameter No.088～093 and they can be overriden by rapid override; the dry run speed is set by number parameter No.086.

Note 3 The programming speed for feed per revolution is displayed when the block involving feed per revolution is being executed.

Note 4 The total workpieces machined is set by number parameter P355, total workpieces to be machined is set by number parameter P356.

3.1.3 Relative coordinate clearing and mediating

The steps of relative coordinate position clearing are as follows:

1）Enter a mode that displays the relative coordinate;(Fig. 3-1-2)

2）**Clearing operation**: Press and hold “X” key till X in the display flickers, then press CANCEL key, the relative coordinate in X axis will be cleared; (Fig. 3-1-5)

3）**Mediating operation**: Press and hold “X” key till “X” in the display flickers, then press “ENTER” key and the relative coordinate in X axis will be mediated. (The relative coordinate of the axis divided by 2)

4）**Coordinate setting**: Press and hold “X” key till “X” in the display flickers, input the setting data and press ENTER key, the data will be entered into the coordinate system.

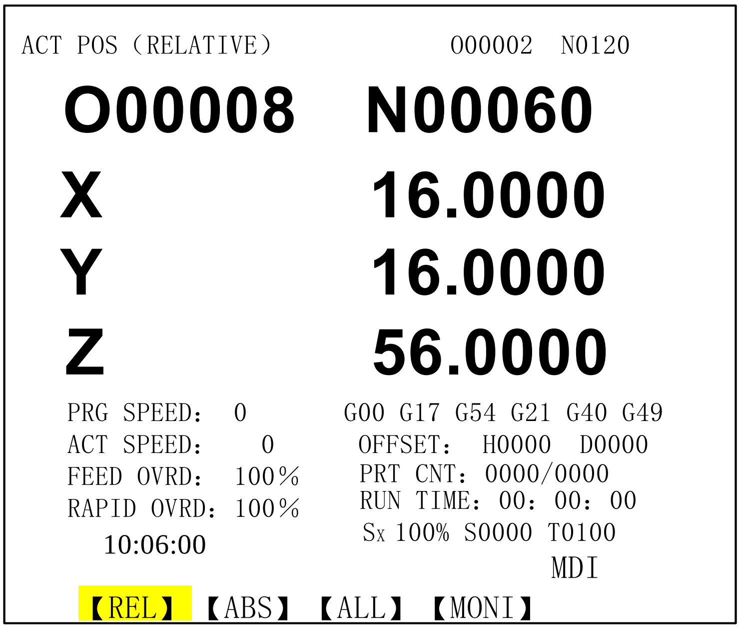


Fig. 3-1-6

5) The clearing of Y and Z axes are the same as above.

3.2 Program display

Press  key to enter program display that have 5 modes: 【◆PRG】,【MDI】,【CUR/MOD】,【CUR/NXT】 and 【DIR】. They can be viewed and modified by corresponding soft keys. See Fig.3-2-1 as following:

1) Program display

Press 【◆PRG】 soft key to enter program page, in this mode, a page of the blocks being executed in the memory can be displayed(See Fig. 3-2-1).

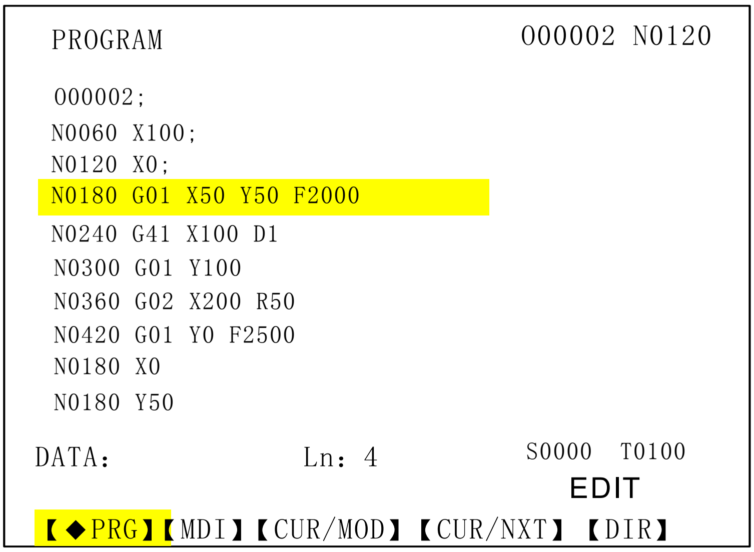


Fig. 3-2-1

Press soft key 【◆PRG】again, it enters the program EDIT and modification page (see Fig.3-2-2)

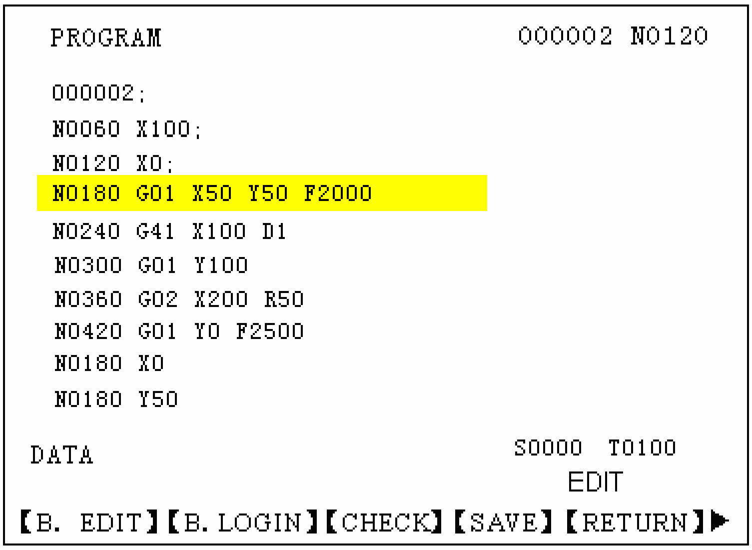


Fig. 3-2-2

Press 【▶】key to enter next page

◀【REPLACE】【CUT】【COPY】【PASTE】【RETURN】▶

Press 【▶】key to enter next page

◀ 【RSTR】 【RETURN】

Press 【◀】key to return to last page

Note The 【CHECK】 function can only be performed in Auto mode.

【B.EDIT】and【B.LOGIN】are used only in AUTO and DNC mode(background edit function). Functions of 【B.EDIT】are the same as the <EDIT> mode that is described in CHAPTER 10 "Program Edit". Save the editing by【B.LOGIN】or exit the background EDIT page by【RETURN】 after editing.

2）MDI display

Press 【MDI】soft key to enter MDI page, in this mode, multiple blocks can be edited and executed. The program format is the same as the editing program. MDI mode is applicable to simple program testing operation. (See Fig. 3-2-3)

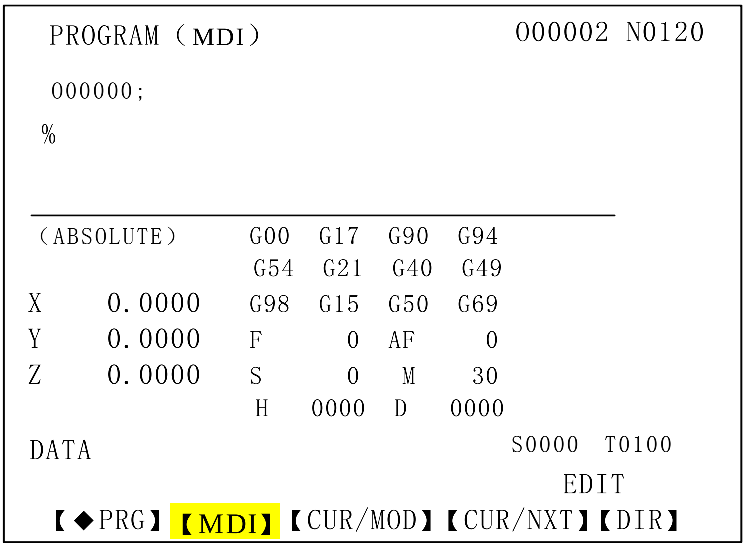


Fig. 3-2-3

3）Program (CUR/MOD) display

Press 【CUR/MOD】soft key to enter current/mode interface, it displays the instructions of the blocks being executed and the current mode. MDI data input and running are available in MDI mode. (See Fig. 3-2-4**).**

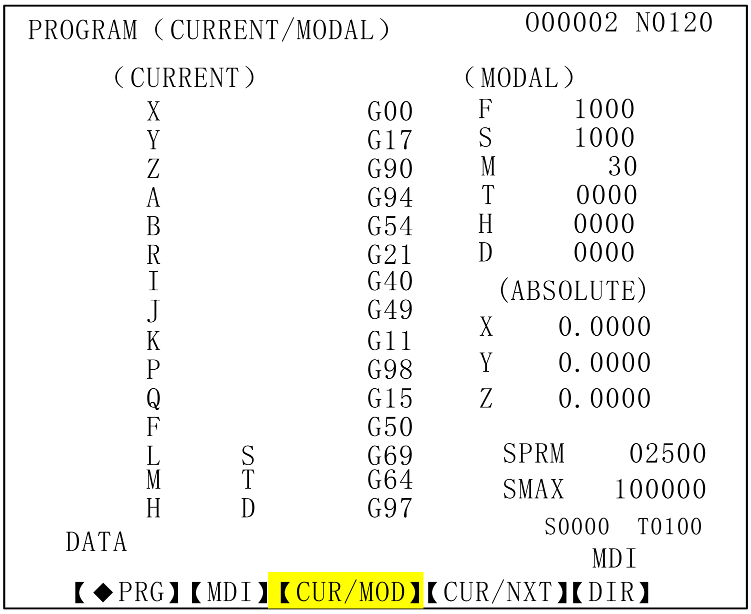


Fig. 3-2-4

4) Program (CUR/NXT) display

Press 【CUR/NXT】soft key to enter current/next interface, It displays the instructions of the blocks being executed and the blocks to be executed. (See Fig. 3-2-5**).**

```
PROGRAM (NEXT/MODAL)                    O00002 N0120

          (CURRENT)                (NEXT)
            X                         X
            Y                         Y
            Z                         Z
            *                         *
            *                         *
            R                         R
            I                         I
            J                         J
            K                         K
            P                         P
            Q                         Q
            F                         F
            L     S                   L     S
            M     T                   M     T
            H     D                   H     D
                                         S0000  T0100
                                               MDI
   【◆PRG】【MDI】【CUR/MOD】【CUR/NXT】【DIR】
```

Fig. 3-2-5

5) Program (DIR) display

Press 【DIR】soft key to enter directory interface, it displays(Fig.3-2-6):

(a) SYS VERSION: hardware and software

(b) PRG USED: The programs saved (including subprogram)

FREE: number of the programs that can be saved.

(c) MEM USED: the capacity occupied by the programs saved (expressed by characters)

FREE: the capacity available for program storage.

(d) PROGRAM DIR: number of the program saved displayed by sequence. Press 

key, the program display changes by the name sequence and time sequence.

```
PROGRAM (DIR)                            O00002 N0120
  SYS VERSION: 1.1(HARD)0.03__07.02.05(SOFT)
  PRG  USED:   16    FREE:    184
  MEM  USED:  832(K) FREE:    7328(K)
  PROGRAM DIR:
 O00001     61     2006-11-12    14: 25
 O00028    252     2006-11-12    14: 25
 O00041   2588     2006-11-12    14: 25
 O00051  14261     2006-11-12    14: 25
 O00151  14261     2006-11-12    14: 25
 O00084    299     2006-11-12    14: 25
 O00073    259     2006-11-12    14: 25
 O00083      9     2006-11-12    14: 25
 O00099     12     2006-11-12    14: 25
                                       S0000  T0100
NO.                                          MDI
  【◆PRG】【MDI】【CUR/MOD】【CUR/NXT】【DIR】
```

Fig. 3-2-6

Explanation: The program numbers in memory can be displayed by the page keys.

3.3 The display, modification and setting of the parameters

3.3.1 Parameter display

Press PARAMETER key to enter parameter page. There are【BITPAR】,【NUMPAR】,【MACRO1】 and【MACRO2】modes in this page. And they can be viewed and modified by corresponding soft keys, the steps are as following:

1）Bit parameter page　Press 【BITPAR】soft key to enter this page (see Fig. 3-3-1):

```
BIT PARAMETER                          000002 N0120
      NO.                  DATA
      0000    1    0    1    0    1    0    1    0
             * *  * *  SEQ  * *  * *  INI  ISO  * *
      0001    1    1    1    1    1    1    0    1
             SJZ  * *  * *  * *  * *  * *  * *  MIRx
      0002    1    1    1    1    1    1    0    0
             ND3  IOP  * *  * *  ASI1 SB1  ASI0 SB0
      0003    1    1    1    1    1    0    0    0
             * *  * *  DIR5 DIR4 DIRZ DIRY DIRX INM
      0004    1    1    1    1    0    0    0    0
             IDG  * *  * *  XIK  AZR  SFD  DLZ  JAX
      0005    1    1    0    0    0    0    0    0
             IPR  * *  * *  * *  * *  * *  ISC  * *
   DATA                                 S0000 T0100
                                              MDI
   【BITPAR】【NUMPAR】【MACRO1】【MACRO2】
```

Fig. 3-3-1

See details about this parameter in Appendix 1.

2）Number parameter page　Press 【NUMPAR】soft key to enter this page (see Fig. 3-3-2):

```
NUM PARAMETER                                      000002 N0120
NO.              MEANING                                   DATA
0000  I/O channel,select input/output device                  9
0001  communication channel 0 baud rate(unit:100)           384
0002  communication channel 1 baud rate                       0
0003  screen protection wait time(minute)                     0
0004  system interpolation period(millisecond)                1
0005  5TH of the sec ref pnt in machine crd  system           3
0006  program axis name of each axis                          0
0007  set axis name in basic coordinate system                0
0008  servo axis number of each axis                          0
0009  STANDBY                                                 0
0010  external workpiece origin point X offset           0.0000
0011  external workpiece origin point Y offset           0.0000
DATA                                               S0000 T0100
                                                          EDIT
     【BITPAR】【NUMPAR】【MACRO1】【MACRO2】
```

Fig. 3-3-2

See details about this parameter in Appendix 1.

3） Macro variable 1 page Press 【MACRO 1】soft key to enter this page (see Fig. 3-3-3):

COMMON VARIABLES 000002 N0120

NO.	DATA	NO.	DATA
0000		0012	
0001		0013	
0002		0014	
0003		0015	
0004		0016	
0005		0017	
0006		0018	
0007		0019	
0008		0020	
0009		0021	
0010		0022	
0011		0023	

ALWAYS NULL

DATA S0000 T0100

EDIT

【BITPAR】【NUMPAR】【MACRO1】【MACRO2】

Fig. 3-3-3

4）Macro variable 2 page Press 【MACRO 2】soft key to enter this page (see Fig. 3-3-4):

SYSTEM VARIABLES 000002 N0120

NO.	DATA	NO.	DATA
1000	0	1012	0
1001	0	1013	0
1002	0	1014	0
1003	0	1015	0
1004	0	1016	0
1005	0	1017	0
1006	0	1018	0
1007	0	1019	0
1008	0	1020	0
1009	0	1021	0
1010	0	1022	0
1011	0	1023	0

INPUT INTERFACE SIGNAL

NO. S0000 T0100

EDIT

【BITPAR】【NUMPAR】【MACRO1】【MACRO2】

Fig. 3-3-4

3.3.2 Modification and setting of the parameter values

1) Select MDI mode;

2) Enter <SET> mode, input the corresponding password in the 2nd page【PSW】mode of the "SET" , and <RETURN> to 【ON-OFF】mode to set the parameter on-off for ON.

3) Press PARAMETER key to enter the parameter page;

4) Move the cursor to the parameter number to be changed:

Method 1: Press page keys to display the parameter to be set; then move the cursor to the place to be modified;

Method 2: Press SEARCH key, input parameter number and press ENTER key for location (step 4 can be ignored).

5) Input the new parameter values by numerical keys;

6) Press ENTER key to enter and display the parameter value;

7) After all parameters are set and entered, set the parameter on-off for OFF.

3.4 Offset display, modification and setting

3.4.1 Offset display

Press OFFSET key to enter offset page, there are 【◆OFFSET】,【PITCH1】,【PITCH2】,【PITCH3】, 【PITCH4】, 【PITCH5】 sub-modes in this page. They can be viewed or modified by corresponding soft keys, which is shown as following:

1）Offset page Press 【◆OFFSET】soft key to enter this page（Fig. 3-4-1）:

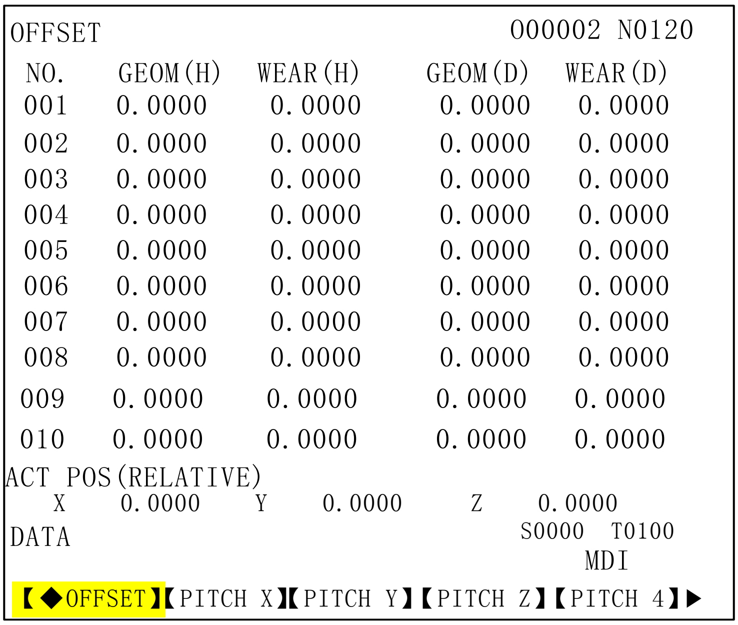


Fig. 3-4-1

Press 【▶】soft key to enter the 2nd page of OFFSET

【 ◀ 】 【PITCH 5】

Press 【◆OFFSET】soft key again to enter OFFSET operation page, as is shown in Fig.3-4-2.

OFFSET 000002 N0120

NO.	GEOM(H)	WEAR(H)	GEOM(D)	WEAR(D)
001	0.0000	0.0000	0.0000	0.0000
002	0.0000	0.0000	0.0000	0.0000
003	0.0000	0.0000	0.0000	0.0000
004	0.0000	0.0000	0.0000	0.0000
005	0.0000	0.0000	0.0000	0.0000
006	0.0000	0.0000	0.0000	0.0000
007	0.0000	0.0000	0.0000	0.0000
008	0.0000	0.0000	0.0000	0.0000
009	0.0000	0.0000	0.0000	0.0000
010	0.0000	0.0000	0.0000	0.0000

ACT POS(RELATIVE)
X 0.0000 Y 0.0000 Z 0.0000
DATA S0000 T0010
MDI
【INPUT】 【+INPUT】 【-INPUT】 【RETURN】▶

Fig. 3-4-2

The offset value may be input directly or operated with the actual position value. H stands for length compensation, and D for radius compensation.

2) Pitch X page Press 【PITCH X】soft key to enter this page（see Fig. 3-4-3):

X PIT-ERROR(MOST: 256) 000002 N0120

NO.	DATA	NO.	DATA	NO.	DATA
0000	0	0012	0	0024	0
0001	0	0013	0	0025	0
0002	0	0014	0	0026	0
0003	0	0015	0	0027	0
0004	0	0016	0	0028	0
0005	0	0017	0	0029	0
0006	0	0018	0	0030	0
0007	0	0019	0	0031	0
0008	0	0020	0	0032	0
0009	0	0021	0	0033	0
0010	0	0022	0	0034	0
0011	0	0023	0	0035	0

DATA S0000 T0100
EDIT
【◆OFFSET】【PITCH X】【PITCH Y】【PITCH Z】【PITCH 4】▶

Fig. 3-4-3

Note The display of pitch offset for Y, Z, 4TH, 5TH axis is the same as that of X axis.

3.4.2 Modification and setting of the offset value

3.4.2.1 Modification and setting of the offset value

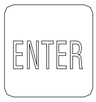
The steps for tool offset in Tool Offset page are as follows:

1) Press 

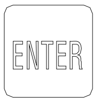
key to enter the Tool Offset page;

2) Move the cursor to the offset number for inputting;

Step 1: press page keys to display the offset page to be modified, move the cursor by pressing cursor keys to locate the offset number to be modified.

Step 2: press "SEARCH" key, input offset number and press 

key for location.

3) In <MDI > mode, input the offset value. After pressing 

key, the offset amount is computed by system automatically and displayed on LCD.

Note 1 During the tool offset setting, the new offset value is ineffective till its offset number T code is specified.

Note 2 The offset value can be modified during the program execution. If the value is needed to be effective during the program execution, the modification must be completed before the tool offset number is used.

Note 3 If length offset is needed to be added the relative coordinate value of Z axis, the offset value should follow Z code, then it will be automatically accumulated.

Example

If Z10 is input, the offset value is the one the actual relative coordinate added by 10.

3.4.2.2 Modification and setting of pitch offset

1）Set the offset value required by modifying the parameters（NO.221~NO.225）.

2）Enter offset value for every point by sequence.

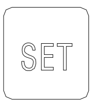
3）The pitch error offset is associated with the offset interval and offset override.

Note Refer to Volume Four Connection of *GSK218M Connection and PLC* **manual for the pitch setting.**

3.5 Setting display

3.5.1 Setting page

1 Entry of page

Pressing 

key to enter the SETTING page, there are 【SETTING】,【SWITCH】,

【G54-G59】, 【PANEL】, 【SERVO】, 【DATA】,【PSW】 sub-modes in this page. They can be viewed or modified by corresponding soft keys, the pages are shown as following (see Fig. 3-5-1):

```
SETTING                                    O00002 N0120

MIRROR X  =    1    (0: OFF 1: ON)
MIRROR Y  =    1    (0: OFF 1: ON)
MIRROR Z  =    1    (0: OFF 1: ON)
CODE      =    1    (0: EIA,  1:  ISO)
IN UNIT   =    0    (0: MM,   1: INCH)
I/O CHAN. =    0    ( 0—3  CHANNEL NO.)
ABS PRG   =    0    (0: ABS, 1: INC)
AUTO SEQ  =    0    (0: OFF 1: ON)
SEQ STOP  =    0000 (PROGRAM NO.)
SEQ STOP  =    0000 (SEQUENCE NO.)
    2006 Y 11 M 14 D      14 H 26 M 45 S
                                       S0000  T0100
DATA                                       MDI
  【SETTING】【SWITCH】【G54-G59】【PANEL】【SERVO】【▶】
```

Fig. 3-5-1

2 【SETTING】explanation

Press 【SETTING】soft key to enter the page shown as Fig.3-5-1. After entering the page, the user can view and modify the parameters. The operation steps are as following:

（**a**）Enter < MDI> mode;

（**b**）Move the cursor to the items to be altered by pressing cursor keys;

（**c**）Key in 1 or 0 by following steps:

1) X，Y，Z axis mirror image

1: Mirror image on 0：Mirror image off

2) ISO code

When the data in memory are input or output, the code selected:

1: ISO code 0: EIA code

Note Use ISO code if GSK218M universal programmer is used.

3) Inch programming

Set the input unit of the program for inch or mm

1: inch 0: mm

4) I/O channel

To be set by user's requirement.

5） Absolute programming

0： Absolute programming 1：Incremental programming

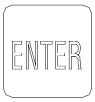
6) Automatic sequence number

0: The number is not inserted by system automatically when inputting program by keyboard in Edit mode.

1: The number is inserted automatically by system when inputting program by keyboard in Edit mode. The number increment of blocks can be set by number parameter No.0210.

7) Stop number

This function can be used to specify the program execution to stop at a block specified, but the program number and the block number should be specified together for this function. E.g. 00060（program number）means program number O00060; 00100（sequence number）means block number N00100.

（**d**） Press 

key to confirm the entry.

3.5.2 Parameter and program on-off page

1 Press 【SWITCH】soft key to enter switch setting page

The page is shown as following (see Fig. 3-5-2):

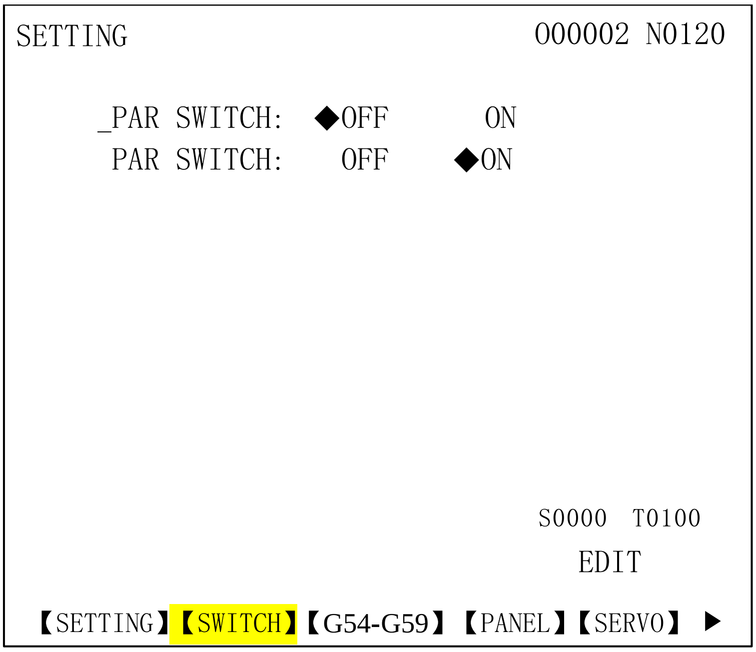


Fig. 3-5-2

2 Operation explanation

In page above, the user can set the parameter and program switch. The operation steps are as following:

（**a**）Enter the <MDI> mode, the parameter ON should be in MDI mode; parameter OFF and program ON and OFF may be in any mode.

（**b**）Enter the <SET> page, input the corresponding password in the 2nd 【PSW】page of the “SETTING”;

（**c**）Move the cursor to the items to be altered in the parameter or program;

(d) Set the parameter or program switch by pressing Left or Right cursor key. When the parameter switch is set for "OFF", the system parameter modification and setting are unallowed; when the program switch is set for "OFF", the program editting is unallowed too.

3.5.3 Coordinate setting interface

1. Press 【G54-G59】soft key to enter coordinate setting interface, which is shown as following (Fig.3-5-3-1):

```
SETTING（G54-G59）                         O00002 N0120
     CURRENT WORKPIECE: G54
    (MACHINE)          (G54)              (G55)
   X     0.00000   X      0.0000   X      0.0000
   Y     0.00000   Y      0.0000   Y      0.0000
   Z     0.00000   Z      0.0000   Z      0.0000

     (EXT)             (G56)              (G57)
   X     0.00000   X      0.0000   X      0.0000
   Y     0.00000   Y      0.0000   Y      0.0000
   Z     0.00000   Z      0.0000   Z      0.0000


 DATA                                   S0000   T0100
                                           MDI
 【SETTING】【SWITCH】【G54-G59】【PANEL】【SERVO】 ▶
```

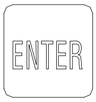
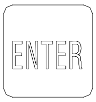
Fig. 3-5-3-1

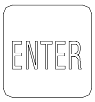
Besides 6 (from G54 to G59) workpiece coordinate system (standard), 50 additional workpiece coordinate systems can also be used in this system, as are shown in Fig. 3-5-3-2. And each coordinate system can be viewed or modified by page keys. See details for these additional workpiece coordinate systems in *PROGRAMMING* Section 4.2.9. Additional workpiece coordinate system.

```
SETTING（G54-G59）                         O00002 N0120
     CURRENT WORKPIECE: G54
    (MACHINE)          (G58)              (G59)
   X     0.00000   X      0.0000   X      0.0000
   Y     0.00000   Y      0.0000   Y      0.0000
   Z     0.00000   Z      0.0000   Z      0.0000

     (EXT)             (G54.001)          (G54.002)
   X     0.00000   X      0.0000   X      0.0000
   Y     0.00000   Y      0.0000   Y      0.0000
   Z     0.00000   Z      0.0000   Z      0.0000


 DATA                                   S0000   T0010
                                           MDI
 【SETTING】【SWITCH】【G54-G59】【PANEL】【SERVO】 ▶
```

Fig. 3-5-3-2

2. There two ways for coordinate entry:

1）After entering this page in <MDI> mode, move the cursor to the coordinate system to be altered. Press the axis name to be assigned and then press  key for confirmation, the value in current machine coordinate system will be set for the origin of the G coordinate system. e.g. If “X ” is pressed and then  key, the X machine coordinate of a point is entered automatically by system. Another example, if “X10” is entered, and then press  key, which means X machine coordinate is +10; and“X-10” may also be entered.

2）After entering this page in <MDI> mode, move the cursor to the coordinate axis to be altered, input the machine coordinates or other values directly to define the G coordinate system origin, press  key for confirmation.

3.5.4 Display and setting of the machine soft panel

1. Press【PANEL】soft key to enter machine panel page, which is shown as following(See Fig. 3-5-4):

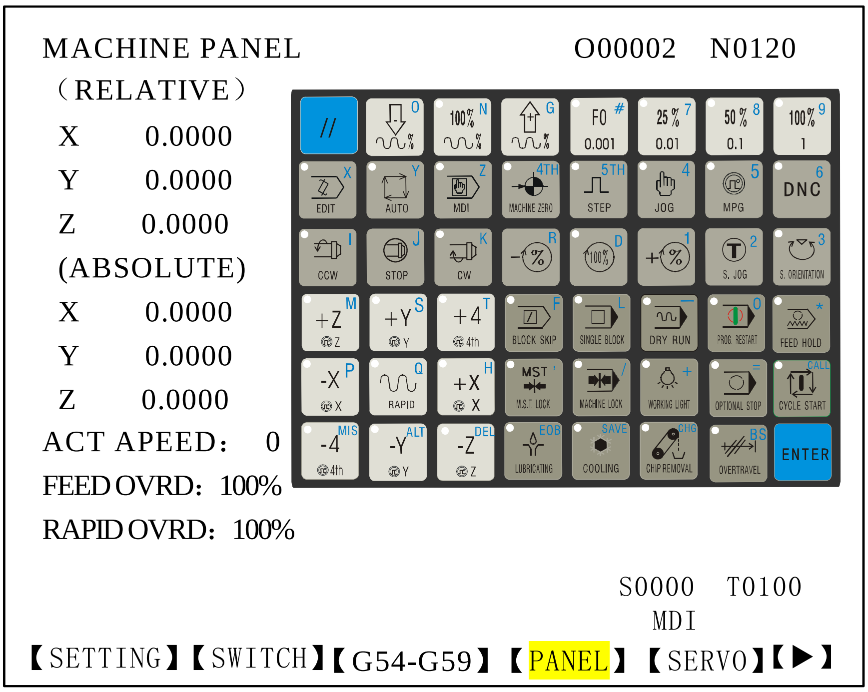


Fig. 3-5-4

2. Usage:

The functions of all soft keys on machine soft panel are identical with that of the keys on machine panel. In this page, the soft keys correspond to the machine operator panel keys to the right of the displayer by the key's up-right letter signs one by one. The corresponding indicator on the machine panel and the up-left indicator of the soft panel light up if a soft key is selected, which is consistent with the key operation on the machine panel.

The soft key operations are set by bit parameters No. 57.0, No. 57.5, No. 57.6, No. 57.7.

3.5.5 Servo page

Press 【SERVO】soft key to enter this page, it is shown as following(See Fig. 3-5-5):

```
SETTING（SERVO）                    O00002   N00120
                   X AXIS      Y AXIS      Z AXIS
PROPORTION：       0.0000      0.0000      0.0000
INTEGRAL：         0.0000      0.0000      0.0000
DIERENTIAL：       0.0000      0.0000      0.0000
FEEDBACK：         0.0000      0.0000      0.0000
SET PERIOD：       0.0000      0.0000      0.0000
FIL TER：          0.0000      0.0000      0.0000
FEED DIRECT：      0.0000      0.0000      0.0000
CMR：              0.0000      0.0000      0.0000
FEEDGEARN/M：      0.0000      0.0000      0.0000
REF.COUNTER：      0.0000      0.0000      0.0000
DATA                                S00000      T0010
                                      MDI
【SETTING】【SWITCH】【G54-G59】【PANEL】【SERVO】【▶】
```

Fig. 3-5-5

In this page servo transmission parameters can be modified, but user needs to get a well know about these parameters to avoid machine damage or hurt to personnel.

3.5.6 Backup, restore and transfer of the data

In <SETTING> mode the 2nd page, press 【DATA】soft key to enter data page. The user data (such as mode parameter, number parameter, tool parameter, pitch data, ladder and programs) can be backup (saved) and reverted (read); and the data input or output to PC is also available in this system. The part programs saved in CNC are unaffected during the data backup and reversion.（See Fig.3-5-6）

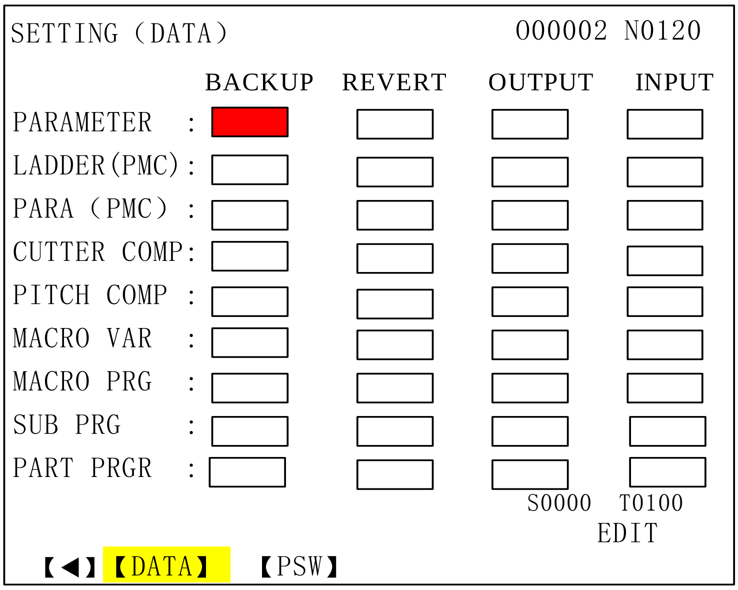
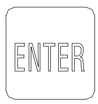


Fig.3-5-7

Operation:

1 In the 2nd page of <SETTING> mode, set the corresponding password in 【PSW】 soft key page. The ladders, parameters can be only operated under the machine builder's authority level. System parameters, tool offset, pitch compensation and system macro variables can be operated under the system debugger level or above.

2 Return to 【DATA】page, after the cursor moves to the target position, the backup or reversion of the data can be finished by pressing 

key.

Note Data input and output system needs to connect with PC to transfer data by the relevant software.

3.5.7 Password authority setting and modification

To prevent the part programs and CNC parameter from malicious modification, the password authority setting is available in this GSK218M system. It is classified for 5 levels, which are the 1st level (system manufacturer), the 2nd level (machine builder), the 3rd level (system debugger), the 4th level (terminal user), the 5th level (operator) by descending sequence. The system defaults the lowest level at power-on（See Fig.3-5-8）.

The 1st and the 2nd level: The modifications of mode parameters, number parameters, tool offset data and PLC ladders transfer etc. are allowed in these levels.

The 3rd level: The modifications of CNC mode parameters, number parameters, tool offset data etc. are allowed in this level.

The 4th level: The modifications of macro variables, tool offset data are allowed in this level. But the modifications of CNC mode parameters, number parameters, pitch compensation data are not allowed in this level.

The 5th level: No password. The operation of the machine operator panel is allowed in this level, but the modifications of tool offset parameters, CNC mode parameters, number parameters, pitch compensation data are not allowed.

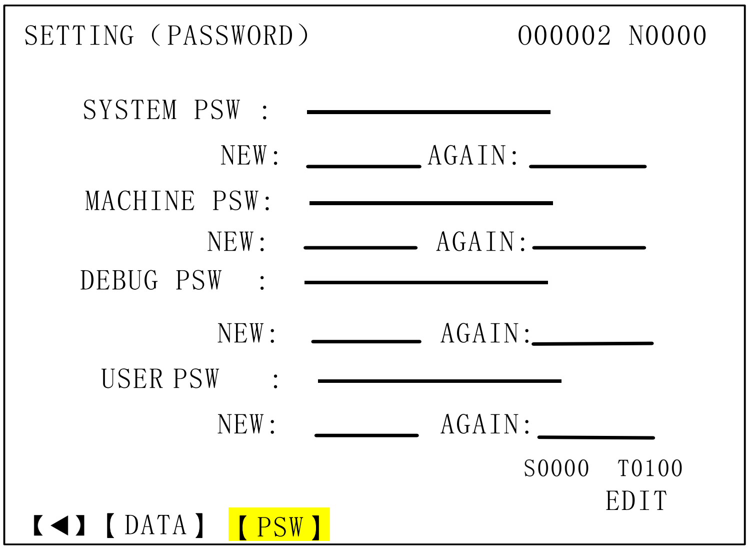


Fig. 3-5-8

1）After entering this page in MDI mode, move the cursor to the item to be altered;

2）Key in the password under the corresponding level, then press  key. If the password is correct, the message “Password is correct.” is issued by the system. If not, “Password is not correct.” is issued.

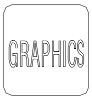
3）Modify the corresponding parameters and setting for the system password;

a The program on-off is required to be set for ON during the parameter modification.

b K parameter is needed to be set for ON during the ladder modification.

4）After modification, the password is automatically deregistered after the system power-off and reset.

3.6 Graphic display

Press 

key to enter the graphic page that has two display modes: 【G. PARA】 and 【◆GRAPH】. They can be switched over by pressing the corresponding soft keys. (See Fig.3-6-1)

```
GRAPH (PARA)                          000000 N00120
  0: XY   1:XZ   2:ZX   3:YZ   4: XYZ   5:ZXY
 AXES      =          0
 GRPH MOD  =          0 (0:GRPH CENT 1:MIN&MAX)
 AUTO ERA  =          0  (0: OFF 1: ON)
 SCALE     =     1.0000
 GRPH CEN  =     0.0000 (X COORDINATE)
 GRPH CEN  =     0.0000 (Y COORDINATE)
 GRPH CEN  =     0.0000 (Z COORDINATE)
 MAX X     =   240.0000
 MAX Y     =   240.0000
 MAX Z     =   240.0000
 MIN X     =-240.0000
 MIN Y     =-240.0000
 MIN Z     =-240.0000                 S0000  T0010
 DATA                                      MDI
        【G. PARA】【◆GRAPH】
```

Fig. 3-6-1

1）Graphic parameter page　Press 【G. PARA】soft key to enter this page, see Fig.3-6-1.

A、 Graphic parameter meaning

①Coordinate selection: set drawing plane that has 6 types as shown in the next line

②Graphic mode: set graphic display mode

③Scaling: set drawing ratio

④Graphic center: set the coordinate of the LCD center in workpiece coordinate system

⑤The maximum and minimum value: The scaling and the graphic center are automatically set when the maximum and minimum value of the axis are set.

Maximum value of X axis: the maximum value along X axis in graphics (unit: 0.001mm)

Minimum value of X axis: the maximum value along X axis in graphics (unit: 0.001mm)

Maximum value of Y axis: the maximum value along X axis in graphics (unit: 0.001mm)

Minimum value of Y axis: the maximum value along X axis in graphics (unit: 0.001mm)

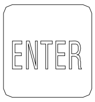
Maximum value of Z axis: the maximum value along X axis in graphics (unit: 0.001mm)

Minimum value of Z axis: the maximum value along X axis in graphics (unit: 0.001mm)

B、 The graphic parameters setting steps:

a、 Move the cursor to the parameter to be set;

b、 Key in the value desired;

c、 Press  key to confirm it.

2）Graphic page Press【GRAPH】soft key to enter this page (See Fig.3-6-2):

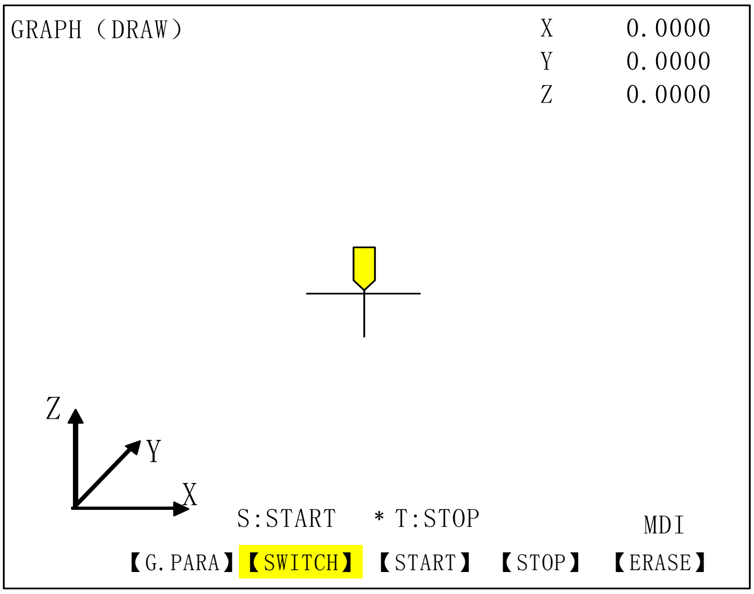
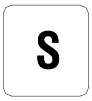
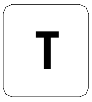


Fig. 3-6-2

The figure machined can be monitored in graphic page,

A Press  key or【START】soft key to enter the DRAW START mode, then the sign‘*’is headed to **S: START**;

B Press  key or【STOP】soft key to enter the DRAW STOP mode, then the sign‘*’is headed to **T**: **STOP**;

C Press 【SWITCH】soft key once to switch over the graph in the corresponding **0**～**5** coordinate display page;

D Press  key or【ERASE】soft key to erase the graph drawn.

3.7 Diagnosis display

The status of DI/DO signals between CNC and the machine, the signals transferred between CNC and PLC, PLC internal data and CNC internal status etc. are shown in the diagnosis display. Refer to *GSK218M CNC System Connection and PLC Manual* for the meaning and setting of the corresponding diagnosis number. The diagnosis of this part is used to detect the CNC interface signals and the internal running signals, and it can’t be modified.

3.7.1 Diagnosis data display

Press DIAGNOSIS key to enter the Diagnose page, which has 5 modes: 【NC】, 【PLC—>NC】, 【MT】, 【PLC—>MT】and 【WAVE】. They can also be viewed by pressing the soft keys（See Fig.3-7-1 to Fig.3-7-5）.

1 NC interface Press 【NC】soft key in <DIAGNOSE> page to enter this interface, as is shown in Fig.3-7-1:

DIAGNOSE（NC） 000002 N00120

NO.	DATA	NO.	DATA
000	0 1 0 0 0 0 0 0	012	0 0 0 0 0 0 1 0
001	0 1 0 0 0 0 0 0	013	0 0 0 0 0 0 1 0
002	0 0 0 1 0 0 0 1	014	0 0 0 0 0 0 1 0
003	0 0 0 0 0 0 0 0	015	0 0 0 0 0 0 0 0
004	0 0 0 0 0 0 0 0	016	0 0 0 0 0 0 0 0
005	0 0 0 0 0 0 0 0	017	0 0 0 0 0 0 0 0
006	0 0 0 0 0 0 0 0	018	0 0 0 0 0 0 0 0
007	0 0 0 0 0 0 0 0	019	0 0 0 0 0 0 0 0
008	0 0 0 0 0 0 0 0	020	0 0 0 0 0 0 0 0
009	0 0 0 0 0 0 0 0	021	0 0 0 0 0 0 0 0
010	0 0 0 0 0 0 0 0	022	0 0 0 0 0 0 0 0
011	0 0 0 0 0 0 0 0	023	0 0 0 0 0 0 0 0

NO. S0000 T0100
EDIT

【N C】【PLC-NC】【 MT 】【PLC-MT】【WAVE】

Fig.3-7-1

This is the signal sent to PLC by CNC system. See *GSK218M CNC System Connection and PLC Manual* for the meaning and setting of the corresponding diagnosis number.

2 PLC—>NC interface In <DIAGNOSE> page, press【PLC—>NC】soft key to enter PLC—>NC interface, as is shown in Fig.3-7-2:

DIAGNOSE（PLC--NC） 000002 N00120

NO.	DATA	NO.	DATA
000	0 1 0 0 0 0 0 0	012	0 0 0 0 0 0 1 0
001	0 1 0 0 0 0 0 0	013	0 0 0 0 0 0 1 0
002	0 0 0 1 0 0 0 1	014	0 0 0 0 0 0 1 0
003	0 0 0 0 0 0 0 0	015	0 0 0 0 0 0 0 0
004	0 0 0 0 0 0 0 0	016	0 0 0 0 0 0 0 0
005	0 0 0 0 0 0 0 0	017	0 0 0 0 0 0 0 0
006	0 0 0 0 0 0 0 0	018	0 0 0 0 0 0 0 0
007	0 0 0 0 0 0 0 0	019	0 0 0 0 0 0 0 0
008	0 0 0 0 0 0 0 0	020	0 0 0 0 0 0 0 0
009	0 0 0 0 0 0 0 0	021	0 0 0 0 0 0 0 0
010	0 0 0 0 0 0 0 0	022	0 0 0 0 0 0 0 0
011	0 0 0 0 0 0 0 0	023	0 0 0 0 0 0 0 0

NO. S0000 T0100
MDI

【N C】【PLC-NC】【 MT 】【PLC-MT】【WAVE】

Fig.3-7-2

This is the signal sent to CNC system by PLC. See *GSK218M CNC System Connection and PLC Manual* for the meaning and setting of the corresponding diagnosis number.

3 MT In <DIAGNOSE> page, press 【MT】soft key to enter MT page, as is shown in Fig.3-7-3:

```
DIAGNOSE（MT）                                000002 N0120
 NO.          DATA            NO.          DATA
 000    0 1 0 0 0 0 0 0       012    0 0 0 0 0 0 1 0
 001    0 1 0 0 0 0 0 0       013    0 0 0 0 0 0 1 0
 002    0 0 0 1 0 0 0 1       014    0 0 0 0 0 0 1 0
 003    0 0 0 0 0 0 0 0       015    0 0 0 0 0 0 0 0
 004    0 0 0 0 0 0 0 0       016    0 0 0 0 0 0 0 0
 005    0 0 0 0 0 0 0 0       017    0 0 0 0 0 0 0 0
 006    0 0 0 0 0 0 0 0       018    0 0 0 0 0 0 0 0
 007    0 0 0 0 0 0 0 0       019    0 0 0 0 0 0 0 0
 008    0 0 0 0 0 0 0 0       020    0 0 0 0 0 0 0 0
 009    0 0 0 0 0 0 0 0       021    0 0 0 0 0 0 0 0
 010    0 0 0 0 0 0 0 0       022    0 0 0 0 0 0 0 0
 011   0 0 0 0 0 0 0 0        023    0 0 0 0 0 0 0 0
 NO.                                   S0000  T0100
                                             MDI
     【N C】【PLC-NC】【 MT 】【PLC-MT】【WAVE】
```

Fig.3-7-3

This is the signal sent to PLC by machine. See *GSK218M CNC System Connection and PLC Manual* for the meaning and setting of the corresponding diagnosis number.

4 PLC—>MT interface In <DIAGNOSE> page, press【PLC—>MT】soft key to enter PLC—>MT interface, as is shown in Fig.3-7-4:

```
DIAGNOSE（PLC--MT）                           000002 N0120
 NO.          DATA            NO.          DATA
 000    0 1 0 0 0 0 0 0       012    0 0 0 0 0 0 1 0
 001    0 1 0 0 0 0 0 0       013    0 0 0 0 0 0 1 0
 002    0 0 0 1 0 0 0 1       014    0 0 0 0 0 0 1 0
 003    0 0 0 0 0 0 0 0       015    0 0 0 0 0 0 0 0
 004    0 0 0 0 0 0 0 0       016    0 0 0 0 0 0 0 0
 005    0 0 0 0 0 0 0 0       017    0 0 0 0 0 0 0 0
 006    0 0 0 0 0 0 0 0       018    0 0 0 0 0 0 0 0
 007    0 0 0 0 0 0 0 0       019    0 0 0 0 0 0 0 0
 008    0 0 0 0 0 0 0 0       020    0 0 0 0 0 0 0 0
 009    0 0 0 0 0 0 0 0       021    0 0 0 0 0 0 0 0
 010    0 0 0 0 0 0 0 0       022    0 0 0 0 0 0 0 0
 011   0 0 0 0 0 0 0 0        023    0 0 0 0 0 0 0 0
 NO.                                   S0000  T0100
                                             MDI
     【N C】【PLC-NC】【 MT 】【PLC-MT】【WAVE】
```

Fig.3-7-4

This is the signal sent to machine by PLC. See *GSK218M CNC System Connection and PLC* manual for the meaning and setting of the corresponding diagnosis number.

5 WAVE interface In <DIAGNOSE> page, press【WAVE】soft key to enter WAVE interface, as is shown in Fig.3-7-5:

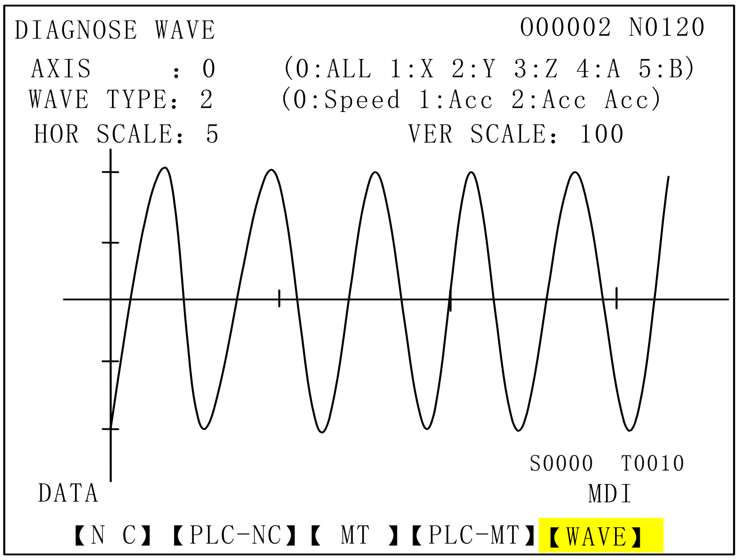
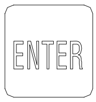


Fig. 3-7-5

AXIS: select the axis name for WAVE

WAVE TYPE: select the WAVE type

HOR, VER SCALE: select the WAVE ratio

Data: in MDI mode, move the cursor to select the data to be modified, and press 

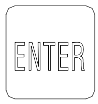
key for confirmation.

3.7.2 Signal viewing

1) Press 

key to select the DIAGNOSE page.

2) The respective address explanation and meaning are shown at the down-left of the screen when moving the cursor to the left or right.

3) Move the cursor or key in the parameter address to be searched, then press 

key, the target address will be found.

4) In【WAVE】interface, the feedrate, acceleration, acceleration of acceleration of each axis can be displayed. It is easy to debug the system and find the optimum suited parameters for the drive and the motor.

3.8 Alarm display

When an alarm is issued, “ALARM” is displayed at the bottom line of the LCD. Press the 

key to display the alarm page, there are 4 modes 【ALARM】, 【USER】, 【HISTORY】, 【OPERATE】 in this page, which can be viewed by the corresponding soft keys（See Fig.3-8-1 to Fig.3-8-4）. They can also be set by parameter No.24.6 for switching to alarm interface if an alarm is issued.

1 Alarm interface In <ALARM> page, press 【ALARM】soft key to enter this interface, as is shown in Fig.3-8-1:

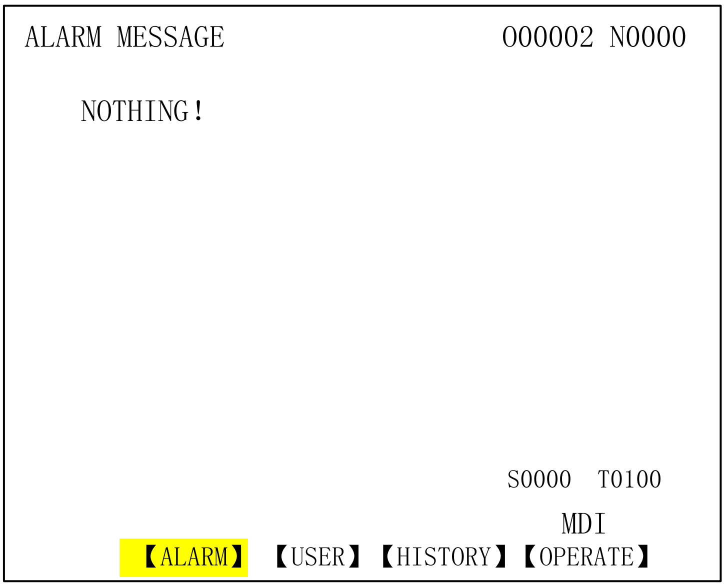


Fig.3-8-1

In alarm page, it displays the message of current P/S alarm number. See details for the alarm in Appendix 2.

2 USER interface In <ALARM> page, press 【USER】soft key to enter this interface, as is shown in Fig.3-8-2:

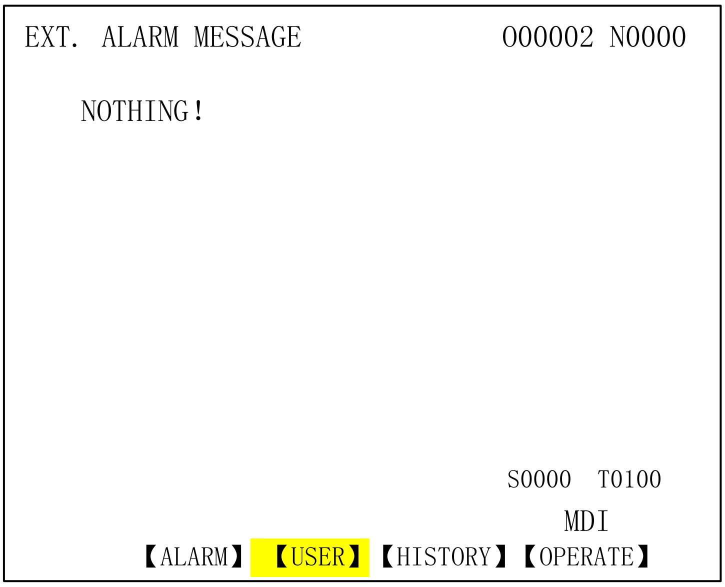


Fig.3-8-2

See *GSK218M CNC System Connection and PLC* manual for the details of the user alarm.

Note The external alarm number can be set and edited by user according to the site conditions. The alarm after editing is input into the system via a transfer software. However, the name of the file edited must be"PLCALM.TXT".

3 HISTORY interface In <ALARM> page, press 【HISTORY】soft key to enter this interface, as is shown in Fig.3-8-3:

```
ALARM MESSAGE                    O00002 N0000

     07-03-01 13:28
         0000:Please power off!




                                  S0000  T0100
                                         EDIT
        【ALARM】 【USER】【HISTORY】【OPERATE】
```

Fig.3-8-3

4 OPERATE interface In <ALARM> page, press 【OPERATE】soft key to enter this interface, as is shown in Fig.3-8-4:

The OPERATE page displays the modification message on the system parameters and ladders, e.g. the modification content and time.

```
OPERATE HISTORY                  O00002 N0000

     07-03-01 13:32
         MODIFY BIT PARA 0021.7
     07-03-02 15:39
         MODIFY BIT PARA 0023.7




                                  S0000  T0100
                                         EDIT
        【ALARM】 【USER】【HISTORY】【OPERATE】
```

Fig.3-8-4

OPERATE and HISTORY alarm interface can display 34 pages of alarm history message, such as the alarm time, alarm number, alarm message and page numbers and they can be viewed by page keys.

The recording of the HISTORY and OPERATE can be deleted by pressing <**DELETE**> key (system debugger level or above).

3.9 PLC display

Press the PLC key to display the PLC page, there are 5 modes 【INFO】,【PLCGRA】,【PLCPAR】,【PLCGDN】, 【◆PLCTRA】in this page, which can be viewed as following by the corresponding soft keys（See Fig.3-9-1 to Fig.3-9-5）.

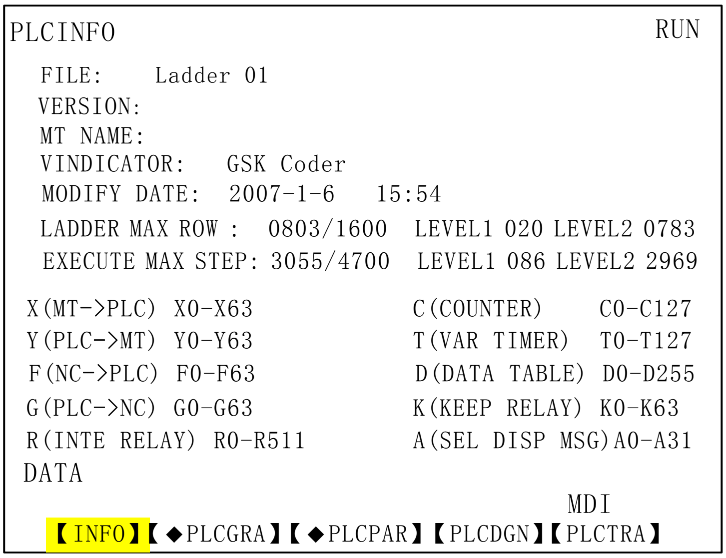


Fig.3-9-1

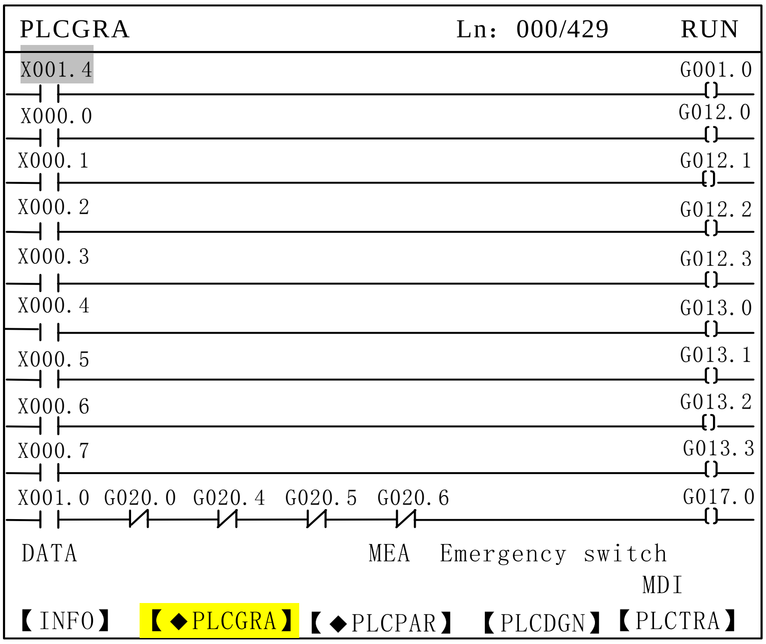


Fig.3-9-2

PLCPara RUN

ADDR	N.7	N.6	N.5	N.4	N.3	N.2	N.1	N.0
K000	0	0	0	0	0	0	0	0
K001	0	0	0	0	1	0	0	0
K002	0	0	0	0	0	0	0	0
K003	0	0	0	0	0	0	0	0
K004	0	0	0	0	0	0	0	0
K005	0	0	0	0	0	1	0	0
K006	0	0	0	0	0	0	1	1
K007	0	0	0	0	0	0	0	0
K008	0	0	0	0	0	0	0	0
K009	0	0	0	0	0	0	0	0
K010	0	0	0	0	0	0	0	0
K011	0	0	0	0	0	0	0	0

DATA

MDI

【INFO】【◆PLCGRA】【◆PLCPAR】【PLCDGN】【◆PLCTRA】

Fig.3-9-3

PLCDGN RUN

ADDR	N. 7	N. 6	N. 5	N. 4	N. 3	N. 2	N. 1	N. 0
F000	0	1	0	0	0	0	0	0
F001	0	0	0	0	1	0	0	0
F002	0	0	0	0	0	0	0	0
F003	0	0	0	0	0	0	0	0
F004	0	0	0	0	0	0	0	0
F005	0	0	0	0	0	0	0	0
F006	0	0	0	0	0	0	0	0
F007	0	0	0	0	0	0	0	0
F008	0	0	0	0	0	0	0	0
F009	0	0	0	0	0	0	0	0
F010	0	0	0	0	0	0	0	0
F011	0	0	0	0	0	0	0	0

No.

MDI

【INFO】【◆PLCGRA】【◆PLCPAR】【PLCDGN】【PLCTRA】

Fig.3-9-4

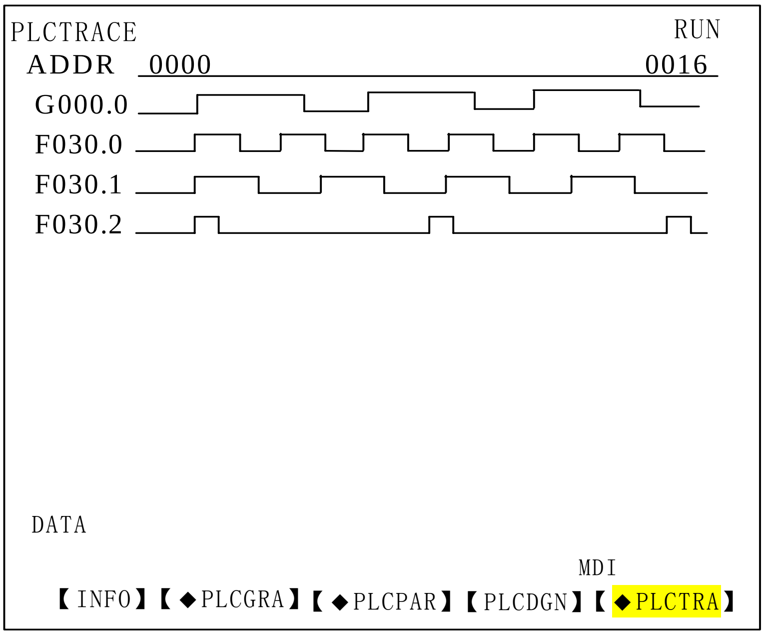


Fig. 3-9-5

Note **Refer to** ***GSK218M CNC System Connection and PLC*** **manual for the PLC ladder modification and relevant message.**

3.10 Index display

Press the  key to display the alarm page, there are 7 modes 【OPRT】,【ALARM】,【G CODE】, 【PARA】,【MACRO】,【PLCADDR】,【CALCULA】 in this page, which can be viewed by the corresponding soft keys（See Fig.3-10-1 to Fig.3-10-7）.

1 OPRT interface In <INDEX> page, press 【OPRT】soft key to enter this interface, as is shown in Fig.3-10-1:

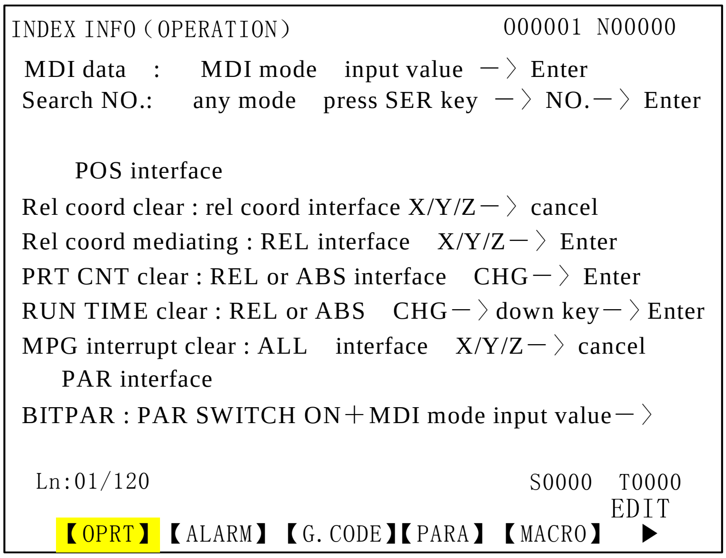


Fig.3-10-1

The manual operation steps for various interfaces are introduced in **<INDEX>** page, you may find the corresponding introduction in INDEX pages if you are unfamiliar with some operations.

2 ALARM interface In <INDEX> page, press 【ALARM】soft key to enter this interface, as is shown in Fig.3-10-2:

INDEX INFO(ALARMS) 000001 N00000

NO.	MEANING
0000	Power not off after parameter input
0001	File open fail
0002	Data input overflow
0003	Program number exists
0004	Digit or character "—" input without address. Modify program.
0005	Address with no data but another address or EOB Code.modify program
0006	Character "—" input wrongly for address or 2 or more "—" input. Modify program.
0007	"."wrongly input (for address),2 or more"."input. Modify program.

No. Ln:1/381 S0000 T0000

EDIT

【OPRT】【ALARM】【G. CODE】【PARA】【MACRO】 ▶

Fig.3-10-2

In this interface, alarms meaning and operations are shown.

3 G code interface In <INDEX> page, press 【G. CODE】 soft key to enter this interface, as is shown in Fig.3-10-3:

INDEX INFO (G CODE) 000001 N00000

G00	G01	G02	G03	G04	G10
G11	G15	G16	G17	G18	G19
G20	G21	G27	G28	G29	G30
G31	G40	G41	G42	G43	G44
G49	G50	G51	G53	G54	G55
G56	G57	G58	G59	G60	G62
G61	G63	G64	G65	G68	G69
G73	G74	G76	G80	G81	G82
G83	G84	G85	G86	G87	G88
G89	G90	G91	G92	G94	G95
G96	G97	G98	G99		

S0000 T0000

Rapid positioning G00

EDIT

【OPRT】【ALARM】【G. CODE】【PARA】【MACRO】 ▶

Fig.3-10-3

The meanings of G codes used in system are shown in G code interface, they can be viewed by cursor selection. And the G codes definitions are shown in the down left of the interface, as is shown in Fig.3-10-3. If you want to know the format and usage of a G code, you can press the <ENTER> key on the panel after you select a G code, as is shown in Fig.3-10-4.

```
INDEX INFO (G CODE)                                O00001 N00000
 Rapid positioning G00
 Instruction format: (G90/G91) G00 X_Y_Z_
 Function:
      G00 instruction, tool traverse via linear,
      Interpolation to workpiece coordinate system
      Position specified by absolute or incremental
      Instruction.
 Explanation:
      In absolute programming, parameter represents
      Programming final coordinate; in incremental
      Programming, parameter represents axes moving
      Distance and direction.

                     P: 1/46                  S0000   T0000
                                                   MDI
   【OPRT】【ALARM】【G.CODE】【PARA】【MACRO】【▶】
```

Fig.3-10-4

The format, function, explanation and restriction of instructions are introduced in this page, you may find the corresponding introduction in this page if you are unfamiliar with these instructions.

4 Parameter interface In <INDEX> page, press【PARA】soft key to enter this interface, as is shown in Fig.3-10-5:

```
INDEX INFO(PARAMETER/DIAGNOSE)          O00001 N00000
    NO.                  MEANING
  0000   parameters related to "SETTING"
               ( bit  par.:0000 — 0002,   num  par.:0000 —
  0001   parameters related to axis control
               ( bit  par.:0003 — 0008,   num  par.:0005 —
  0002   parameters related to coordinate system
               ( bit  par.:0009 — 0010,   num  par.:0010 —
  0003   parameters related to travel detection
               ( bit  par.:0011         ,   num  par.:0066 —
  0004   parameters related to feedrate
               ( bit  par.:0012 — 0014,   num  par.:0086 —
  0005   parameters related to acc/dec control
               ( bit  par.:0015 — 0017,   num  par.:0105 —

  No.                  P:1/5              S0000   T0000
                                               EDIT
  【OPRT】【ALARM】【G.CODE】【PARA】【MACRO】【▶】
```

Fig.3-10-5

The function parameter settings are introduced in this page, you may find the corresponding introduction in it if you are unfamiliar with some parameter settings.

5 MACRO interface In <INDEX> page, press 【MACRO】 soft key to enter this interface, as is shown in Fig.3-10-6:

```
INDEX INFO (MACROINSTRUCTION)          000001 N00000
   G65 H(M)  P(#I)  Q(#J)  R(#K)
       M  : 01~99 operation instruction
       #I : operation result (var,seq,alarm)
       #J : operand 1 (variable,invariable)
       #K : operand 2 (variable,invariable)
       H01: #I=#J
       H02: #I=#J+#K
       H03: #I=#J-#K
       H04: #I=#J * #K
       H05: #I=#J / #K
       H11: #I=#J or #K
       H12: #I=#J and #K
  P:1/4                                  S0000  T0000
                                               EDIT
    【OPRT】【ALARM】 【G.CODE】【PARA】【MACRO】【▶】
```

Fig.3-10-6

The MACRO format and operation instructions are introduced in this page, the local variables, common variables and the system setting range are also shown in this page, you may find the corresponding introduction in it if you are unfamiliar with the macro instruction operations.

6 PLCADDR interface In <INDEX> page, press 【PLCADDR】 soft key to enter this page, as is shown in Fig.3-10-7:

INDEX INFO (PLC ADDRESS) 000001 N00000

MEANING	SYMBOL	ADDRESS
Feed pause alarm signal	SPL	F000#4
Cycle start alarm signal	STL	F000#5
Servo ready signal	SA	F000#6
Automatic run signal	OP	F000#7
Alarm signal	AL	F001#0
Reset signal	RST	F001#1
Spindle speed inpos sig.		F001#3
Spindle enabling signal	ENB	F001#4
Tapping signal	TAP	F001#5
Canceling rigid tap sig.	D TAP	F001#6
Inch input signal	INCH	F002#0
Rapid feedrate signal	RPDO	F002#1

Ln: 1/319 S0000 T0000 EDIT

【◀】【PLCADDR】 【CALCULA】

Fig.3-10-7

The PLC addresses, signs, meanings are introduced in this page, you may find the corresponding introduction in it if you are unfamiliar with the PLC addresses.

7 CALCULA interface In the 2nd page of <INDEX> interface, press 【CALCULA】 soft key to enter this interface, as is shown in Fig.3-10-8:

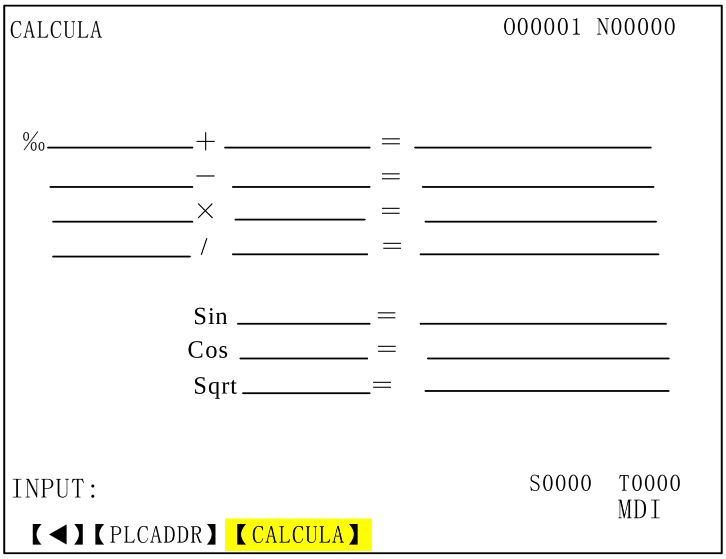
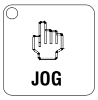


Fig.3-10-8

The operation formats of addition, subtraction, multiplication, division, sine, cosine, extraction are shown in this interface. The cursor may be moved to the space for inputting, and press <ENTER> key for confirmation. After the data input is completed, the system will calculates automatically and input the result to the space behind the “=” sign.

4 Manual Operation

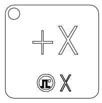
The JOG mode that contains JOG feed, spindle control and machine panel control can be entered by pressing 

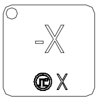
key.

4.1 Coordinate axis movement

In JOG mode, the 3 axes can be moved by JOG feed or manual rapid traverse separately.

4.1.1 Manual feed

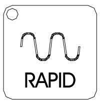
X axis can be moved to positive or negative direction by pressing and holding the 

or 

key in Feed Axis and Direction Selection area. If the key is released, the X axis movement stops. And the feedrate can be overriden to change the feed rate; that of the Y and Z axes are the same as X axis. The three axes simultaneous moving are not available in this system, but the simultaneous zeroing of three axes is supported by the system.

Note The axis JOG feedrate is set by parameter No.098; the manual rapid traverse is set by parameters No.088~ No.092.

4.1.2 Manual rapid traverse

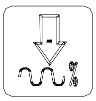
Press 

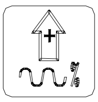
key till the indicator for rapid traverse on panel lights up. Then press manual RAPID key, each axis will traverse rapidly.

Note 1 The manual rapid speeds are set by the parameter No.088～092.

Note 2 The effective manual rapid traverse before reference return is set by the bit parameter No.12.0.

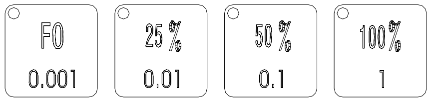
4.1.3 JOG feedrate and manual rapid traverse speed selection

The manual feedrate override classified for 16 gears (0%--150%) is available in JOG feed by pressing 

or 

key for selection.

Note 1 There is an error of 3% for the overrides.

The traverse speed can be selected by pressing 

keys in manual rapid traverse. The override for rapid traverse includes four gears: F0，25％，50％，100％(25％，50％，100％ overrides are set by parameters No.088～092, F0 override by number parameter No.093).

Note 2 The rapid overrides are effective for the following speed:

(1) G00 rapid traverse

(2) Rapid traverse in canned cycle

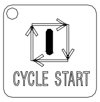
(3) Rapid traverse in G28

(4) Manual rapid traverse

Example: If the rapid traverse speed is 6m/min and override is 50%, speed is 3m/min.

Note 3 The adjusting by override keys during the axis moving is ineffective.

4.1.4 Manual intervention

While a program run in Auto, MDI or DNC modes shifts to JOG mode after a dwell operation, the manual operation is available. Move the axes manually then shift to Auto mode, press 

key to run the program, the axes traverse to the original intervention point by G00 and go on the program execution.

Explanation:

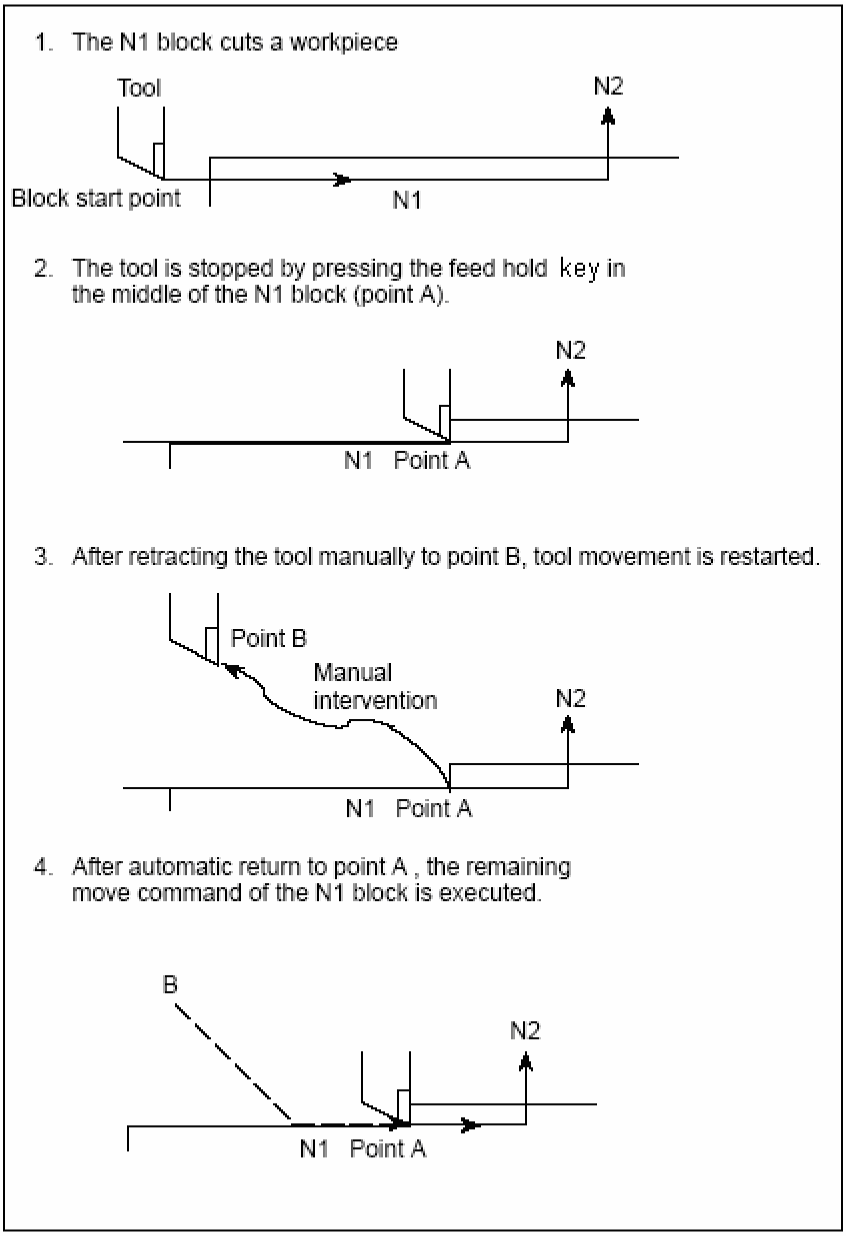
1 If the single block is executed during the returning, the tool will stop at a halt position. When the cycle start is put on, the running is restored.

2 If alarm or resetting occurs during the manual intervention or returning, this function will be cancelled.

3 Don't use machine lock, mirror image, scaling functions during manual intervention.

4 Processing and workpiece figure should be taken into consideration to prevent tool or machine damage prior to manual intervention.

The manual intervention operation is shown in the following figure:

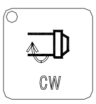


4.2 Spindle control

4.2.1 Spindle CCW

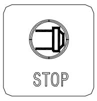
: The spindle is started for CCW rotation if this key is pressed in JOG./MPG/Step mode after S speed is specified in MDI mode.

4.2.2 Spindle CW



: The spindle is started for CW rotation if this key is pressed in Manual./MPG/Step mode after S speed is specified in MDI mode.

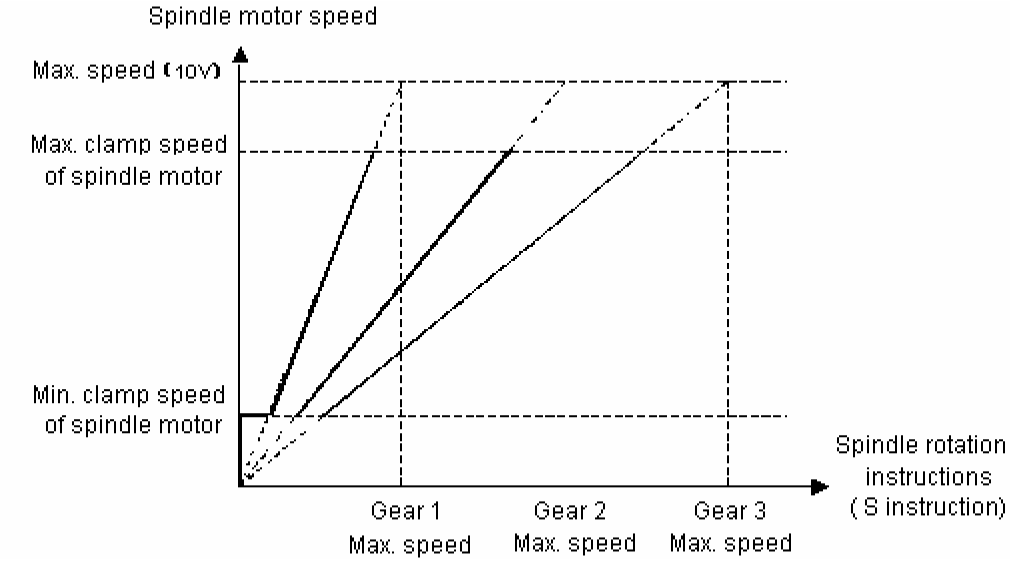
4.2.3 Spindle stop



: The spindle is stopped if this key is pressed in Manual./MPG/Step mode.

4.2.4 Spindle auto gear shift

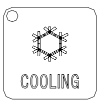
The frequency conversion control or gear control for spindle is set by the parameter No.1.2. If parameter No.1.2=1, the spindle auto gears are controlled by PLC. Three gears(1 to 3 gear) are available in this system, the maximum speed of each gear is set by parameter（P246,P247, P248） respectively, which can be output by modifying the ladder. During the spindle CW or CCW rotation in JOG or Auto mode, the increase or decrease for the corresponding spindle gear can be adjusted by pressing positive/negative override keys. In MDI mode, the system will automatically select the corresponding gear as the specified speed is entered.



Note When the spindle auto gear is effective, the spindle gear is detected by gear in-position signal and S instruction is executed.

4.3 Other manual operations

4.3.1 Cooling control

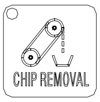


: Compound key. The cooling function is switched between ON and OFF by pressing this key. The indicator lighting up is for ON, gone out for OFF.

4.3.2 Lubricating control

: Compound key. The lubricating function is switched between ON and OFF by pressing this key.

4.3.3 Chip removal



: Compound key. The chip removal function is switched between ON and OFF by pressing this key. The indicator lighting up is for ON, gone out for OFF.

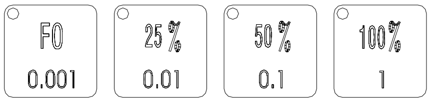
5 Step Operation

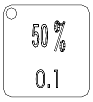
5.1 Step feed

Press 

key to enter the Step mode, in this mode, the machine moves by the system defined step each time.

5.1.1 Selection of moving amount

Press a key to select a moving increment in 

keys, the increment will be shown on the screen. E.g. If press 

key, in <POSITION> interface it displays a step: 0.100 (See Fig. 5-1-1):

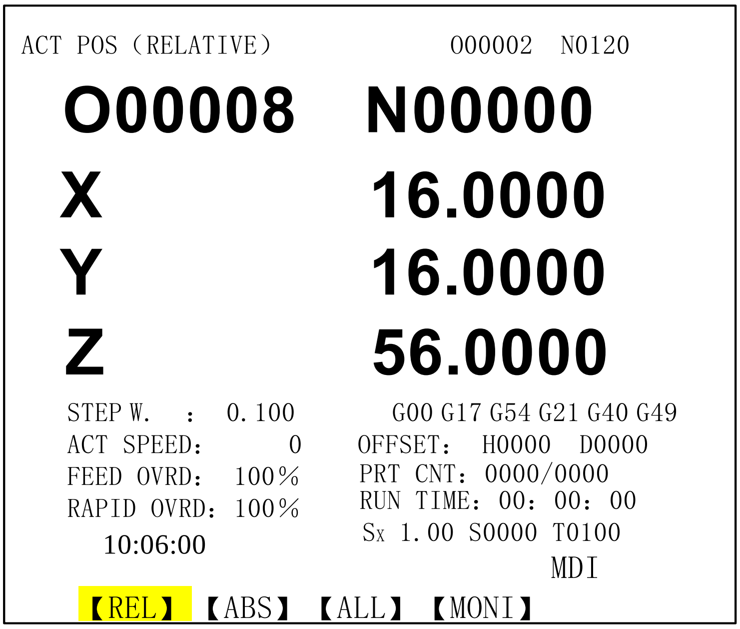
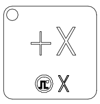
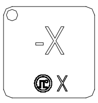


Fig. 5-1-1

The machine axis moves 0.1mm when pressing this key once.

5.1.2 Selection of moving axis and direction

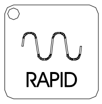
X axis may be moved in positive or negative direction by pressing axis and direction key 

or 

. Press the key once, the corresponding axis will be moved for a step distance defined by system. And the feedrate can be overridden by pressing override keys. The operation for X or Z axis is identical with that of X axis. The manual synchronous 3 axes moving is not supported in this system, but the synchronous 3 axes zero returning is.

5.1.3 Step feed explanation

1 The step moving speed is identical with the JOG feedrate.

2 The rapid override is effective after the 

key is pressed for rapid traverse.

5.2 Step interruption

While the program running in Auto, MDI, DNC mode is shifted to Step mode by dwell, the control will execute the step interruption. The coordinate system of step interruption is consistent with that of MPG, and the operation of it is also the same as that of MPG. See details in the Section 6.2 Controlling in MPG interruption.

The step interruption coordinate system clearing steps: press CTRL+X till “X” flickers, then press <CANCEL> key, the coordinate system will be cleared. The operations of Y, Z are the same as above; while the zero returning is being performed, the coordinate system is cleared automatically.

5.3 Auxiliary control in Step mode

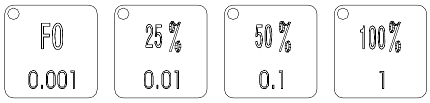
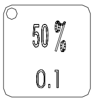
The auxiliary control in Step mode is the same as that in JOG mode. See details in section 4.2 and 4.3 of this manual.

6 MPG Operation

6.1 MPG feed

Press  key to enter the MPG mode, in this mode, the machine movement is controlled by a handwheel.

6.1.1 Moving amount selection

The moving increment will be displayed if a key in  is pressed. e.g. If press  key, it displays the MPG increment in <POSITION> interface: 0.100（See Fig.6-1-1）:

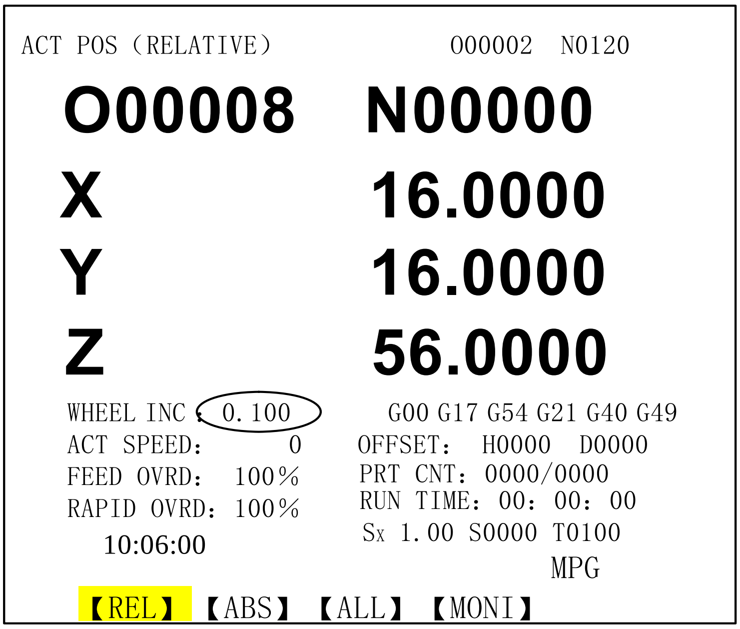
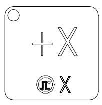


Fig. 6-1-1

6.1.2 Selection of moving axis and direction

In MPG mode, select the moving axis to be controlled by handwheel, press the corresponding key, then the axis can be moved by handwheel.

In MPG mode, if X axis is to be controlled by handwheel, press  key, then X axis can be moved by rotating the handwheel（See Fig.6-1-2）:

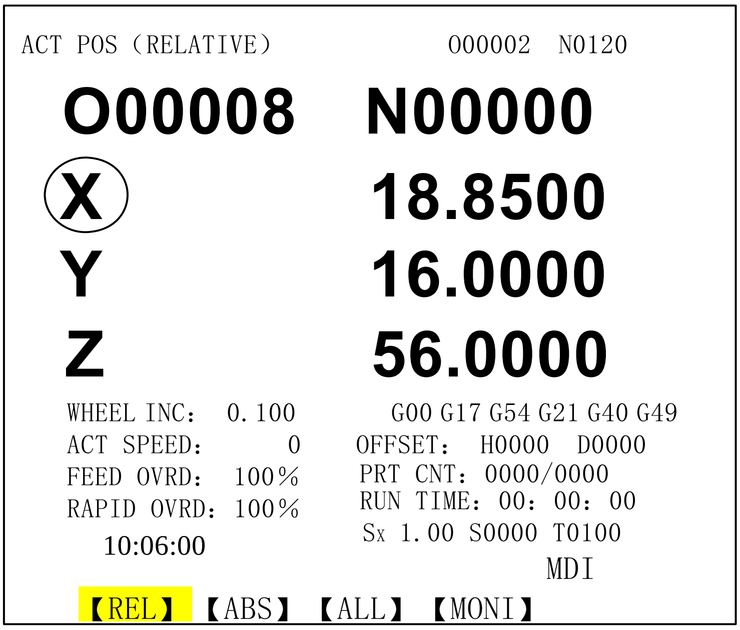


Fig. 6-1-2

The MPG feed direction is decided by the handwheel rotation direction. See details in the machine builder's manual. Usually, the CW of handwheel is the positive feed, CCW for negative feed.

6.1.3 Explanation of MPG feed

1 The relation of the handwheel scale and the machine moving amount are as following table:

	Moving amount of a handwheel scale			
MPG increment (mm)	0.001	0.01	0.1	1
Machine moving amount (mm)	0.001	0.01	0.1	1

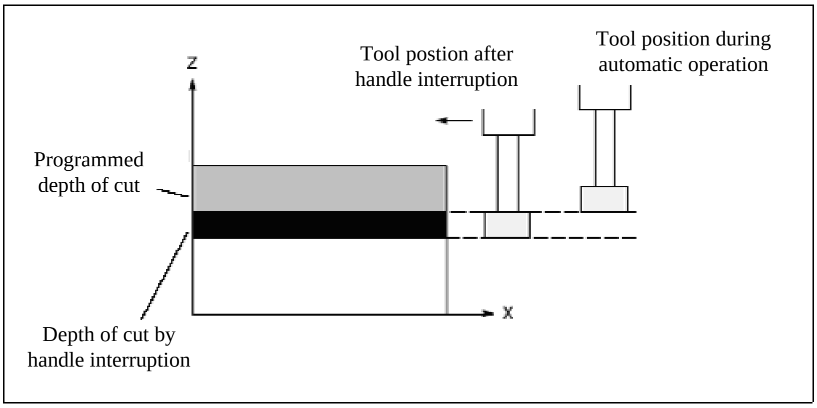
2 The value in the table varies with the mechanical transmission. See details in the machine builder's manual;

3 The speed of the handwheel rotated should be less than 5 r/s. If not, there may be inconsistent between the scale and the moving amount.

6.2 Control in MPG interruption

6.2.1 MPG interruption operation

MPG interruption operation can be overlapped with the automatic movement in Auto mode.



Operation steps:

1）After the dwell of the program execution in Auto mode, switch over the control to MPG mode.

2）For the tool offset by handwheel, move Z axis downward or X, Y axis parallel modify the coordinate system.

3）After the control is switched to Auto mode, the workpiece coordinates remain unchanged till the coordinates restore to their actual values after the machine zero return operation.

As the program run in Auto, MDI, DNC mode is shifted to MPG mode by dwell, the control will execute the MPG interruption. The coordinate system for MPG interruption is shown in Fig.6-2-1.

ACTUAL POSITION O00002 N00120

（RELATIVE）		（ABSOLUTE）		（MACHING）	
X	0.0000	X	0.0000	X	0.0000
Y	0.0000	Y	0.0000	Y	0.0000
Z	0.0000	Z	0.0000	Z	0.0000
（HANDLE INTR）		（SUBSPEED）		（REM DIST）	
X	0.0000	X	0.0000	X	0.0000
Y	0.0000	Y	0.0000	Y	0.0000
Z	0.0000	Z	0.0000	Z	0.0000

S00000 T0010

MDI

【REL】 【ABS】 【ALL】 【MONI】

Fig.6-2-1

The MPG interruption coordinate system clearing steps: press CTRL+X till “X” flickers, then press <CANCEL> key, the coordinate system will be cleared. The operations of Y, Z are the same as above; while the zero returning is being performed, the coordinate system is cleared automatically.

6.2.2 Relation of MPG interruption with other functions

Display	Relation
Machine lock	If machine lock is effective, the machine move is ineffective in MPG interruption.
Absolute coordinate value	MPG interruption does not change the absolute coordinates.
Relative coordinate value	MPG interruption does not change the relative coordinates.
Machine coordinate value	The changing amount of machine coordinate is the displacement amount induced by MPG rotation.

Note **The moving amount of MPG interruption is cleared when the manual reference point return is performed by each axis.**

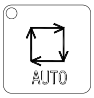
6.3 Auxiliary control in MPG mode

The auxiliary operation in MPG mode is identical with that in JOG mode. See Section 4.2 and 4.3 for details.

7 Auto Operation

7.1 Selection of the auto run programs

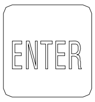
1 Program loading in auto mode

(a) Press 

key to enter the Auto mode;

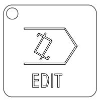
(b) Press 

key to enter the program page, move the cursor to find the target program;

(c) Press 

key for confirmation.

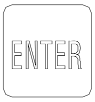
2 Program loading in Edit mode

(a) Press 

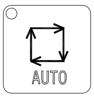
key to enter the Edit mode;

(b) Press 

key to enter the program page, move the cursor to find the target program;

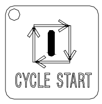
(c) Press 

key for confirmation.

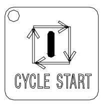
(d) Press 

key to enter the Auto mode;

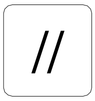
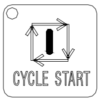
7.2 Auto run start

After select the program by the two ways of section 7.1 above, press 

key to execute the program, the program execution can be viewed by switching to <POSITION>, <MONI><GRAPH> etc. interfaces.

The program execution is started from the line where the cursor locates, so check that whether the cursor is located at the program to be executed before pressing the 

key. If

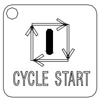
the cursor is not located at the start line from which the program is to be executed, press reset key , then press 

key to run the program automatically from the start line.

7.3　Auto run stop

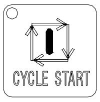
In Auto run, to make the program being executed automatically to be stopped, five ways are provided in this system:

1　Program stop (M00)

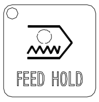
After the block containing M00 is executed, the auto running pauses and the modal message is saved. If 

key is pressed, the program execution is continued.

2　Program optional stop (M01)

If the OPTIONAL STOP （M01）key is pressed during the program execution, the automatic running pauses and the modal message is saved when the block containing M01 is being executed. If 

key is pressed, the program execution is continued.

3　Press 

key

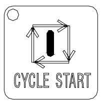
If the 

key is pressed during the auto running, the machine status is:

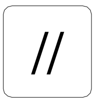
1) Machine feeding slows down and stops;

2) Dwell continues if Dwell is being executed（G04 instruction）;

3) The remaining modal message is saved;

4）The program execution is continued after 

key is pressed.

4　Press  key

See Section 2.3.1.

5　Press EMERGENCY STOP button

See Section 2.3.2.

In addition if the control is switched to other mode from Auto mode, DNC mode, MDI interface of MDI mode in which the program being executed, the machine can also be stopped. The steps are as following:

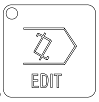
1）If the control is switched to Edit, MDI, DNC mode, the machine stops after the current block is executed.

2）If the control is switched to JOG, MPG, Step mode, the machine interruption stops immediately.

3）If the control is switched to Machine zero interface, the machine slows down to stop.

7.4 Auto running from an arbitrary block

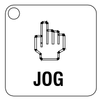
This system permits the current program to be executed from an arbitrary block of it. The steps are as following:

1、Press 

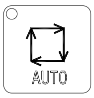
key to enter Edit mode, then press 

key to enter program page, select the program to be executed in【DIR】;

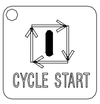
2、Open the program and move the cursor to the block to be executed;

3、Start spindle and other miscellaneous functions by pressing 

key to enter JOG mode;

4、Press 

key to enter Auto mode;

5、Press 

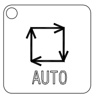
key to execute the program automatically.

Note 1 Before execution, confirm the current coordinate point to be the end of the block preceding to the block to be executed (confirmation of the current coordinate point is unnecessary if the block to be executed is absolute programming and contains G00/G01);

Note 2 If the block to be executed is tool change operation etc, ensure that the interference between the tool and the workpiece at current position, which may cause machine damage or personnel hurt, will not occur.

7.5 Dry run

Before the program execution, a dry run can be performed to have a check for the program, which is usually used together with “MACHINE LOCK”, “M.S.T. LOCK”.

Press  key to enter Auto mode, press  key (the Dry Run indicator in panel lighting up means the current mode is DRY RUN).

In rapid feed, the program speed is dry run speed × rapid override

In cutting feed, the program speed is dry run speed × feedrate override

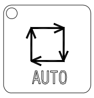
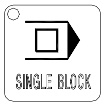
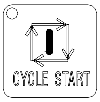
Note 1 The dry run speed is set by the number parameter No.86;

Note 2 The effectiveness of dry run in cutting feed is set by the bit parameter No.12.6.

Note 3 The effectiveness of dry run in rapid positioning is set by the bit parameter No.12.7.

7.6 Single block running

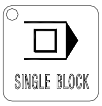
"Single Block" can be selected for checking a block execution.

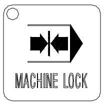
Press  key to enter Auto mode, press  key (The SINGLE BLOCK indicator in panel lighting up means the current mode is Single Block. In this mode, the system stops after a block is executed. Press  key to go on next block execution, perform the operation repeatedly till the whole program is executed.

Note 1 In G28 mode, the single block stop can be performed at an intermediate point.

Note 2 The Single Block function is ineffective if the subprogram calling （M98）or the subprogram calling return（M99）instruction is specified. But for a block with M98 or M99, if M98 or M99 block contains an address other than N，O，P, the Single Block function is effective.

7.7 Running with machine lock

In Auto mode, press  key (The MACHINE LOCK indicator in panel lighting up means the current mode is Machine lock. In this mode, the machine axes don't move. But the position coordinates displayed are the same as that during machine moving. And M, S, T are effective too. This function is used for program verification.

Note **Due to that the machine position is not consistent with its coordinate position after** 

key is pressed and program running, the machine zero operation is needed to be performed.

7.8 Running with M.S.T. lock

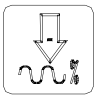
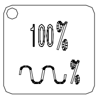
In <AUTO> mode, press 

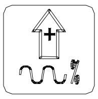
key (The M.S.T. LOCK indicator in panel lighting up means the current mode is M.S.T. LOCK). In this mode, the M, S, T instructions are not executed. This function is used for program verification together with the Machine Lock.

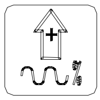
Note **M00，M30，M98，M99 is executed by convention.**

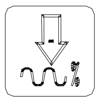
7.9 Feedrate and rapid override in auto run

In <AUTO> mode, the feedrate and rapid traverse speed can be overriden by the system.

In auto run, the feedrate override classified for 16 gears can be selected by pressing  

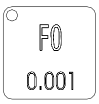
 keys.

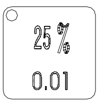
The feedrate override ascends for a gear（5％） till 150％ each time the  key is pressed;

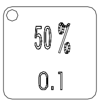
The feedrate override descends for a gear（5％）till 0 each time the  key is pressed.

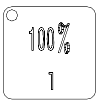
Note F value in feedrate override program

The actual federate = F value specified × feedrate override

During auto running, the rapid traverse speed can be selected by pressing 







key. The 4 gears override of F0，25％，50％，100％ are available for the rapid traverse.

Note The rapid traverse speed value overriden by rapid override and number parameters No.088, No.089, No.090 can be obtained by following equation:

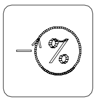
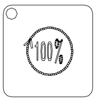
The actual rapid traverse speed of X axis = the value set by parameter No.088 × rapid override

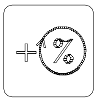
If the override is F0, the axis stop is set by bit parameter No.12.4. If it is set for non-stop 0, the actual rapid traverse speed is set by number parameter No.093 (for all axes).

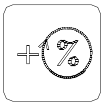
The actual rapid traverse speed of Y or Z axis is as above.

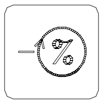
7.10 Spindle override in auto run

In auto run, the spindle speed can be overriden if it is controlled by analog quantity.

The spindle speed can be overriden by pressing spindle override keys  

 in auto mode, which are classified for 16 gears from 0％～150％.

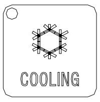
The spindle override ascends for a gear(5%) till 150％ each time the  key is pressed;

The spindle override descends for a gear(5%) till 0％ each time the  key is pressed.

The actual spindle speed=speed specified× spindle override

The max. spindle speed is set by number parameter No.258, if the spindle speed exceeds the max. value set, it uses the max. speed.

7.11 Cooling control

Press 

key in the panel to switch on the cooling on-off, this key is a compound key. The cooling indicator lighting up means the cooling ON, indicator gone off means the cooling OFF.

7.12 Background edit in auto run

The background edit function in processing is supported in this system.

During the program execution in Auto mode, press <PROGRAM> key to enter the program page, then press 【◆PRG】soft key to enter the background edit interface, as is shown in Fig.7-12-1.

```
PROGRAM

000002;
N0060 X100;
N0120 X0;
N0180 G01 X50 Y50 F2000
N0240 G41 X100 D1
N0300 G01 Y100
N0360 G02 X200 R50
N0420 G01 Y0 F2500
N0180 X0
N0180 Y50

DATA        Ln：2
                                        AUTO
【BG. EDT】【BG. END】【CHECK】【SAVE】【RETURN】【▶】
```

Fig. 7-12-1

Press【BG.EDT】soft key to enter the program background edit interface, the program editing operation is the same as that in Edit mode(Refer to Chapter 10 **Program Edit** in this manual). Then press 【B.LOG】soft key to save the edited program and exit this interface.

8 MDI Operation

Except the input, modification, offset operations in MDI mode, the MDI running function is also available in this system. By this function the instructions can be input directly for execution. The input, modification, offset operations etc. are introduced in Chapter 3 “Page display as well as data modification and setting”. This chapter will describe the MDI running function in MDI mode.

8.1 MDI instructions input

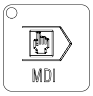
The input in MDI mode is classified for two types:

1 By 【MDI】 type, multiple blocks can be input continuously;

2 By 【CUR/MOD】type, only one block can be input.

The input in 【MDI】is identical with the program input in Edit mode, see Chapter 10 Program Edit in this manual for details. The【CUR/MOD】input is introduced as following.

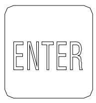
Example: To input a block “G00 X50 Y100” in 【CUR/MOD】page, the steps are:

1 Press 

key to enter the MDI mode;

2 Press 

key to enter the Program page, press 【CUR/MOD】soft key to enter the 【CUR/MOD】page (see Fig.8-1-1):

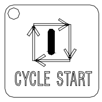
3 Key in the block “G00X50Y100” by sequence and press 

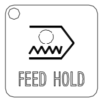
key to confirm, then the block will be displayed on the page (see Fig. 8-1-1):

```
PROGRAM（CURRENT/MODAL）                     O00002  N0120

        （CURRENT）                      （MODAL）
   G0    X   50             G00      F        1000
         Y   100            G17      S        1000
         Z                  G90      M          30
         A                  G94      T        0000
         B                  G54      H        0000
         R                  G21      D        0000
         I                  G40        (ABSOLUTE)
         J                  G49
         K                  G11      X      0.0000
         P                  G98      Y      0.0000
         Q                  G15      Z      0.0000
         F                  G50
         L       S          G69        SPRM      02500
         M       T          G64        SMAX     100000
         H       D          G97
   DATA                                     S0000   T0100
                                                 MDI
  【◆PRG】【MDI】【CUR/MOD】【CUR/NXT】【DIR】
```

Fig. 8-1-1

8.2 Run and stop of MDI instructions

After the instructions are input by the steps in section 8.1, press 

key to run the MDI instructions. During the running, the instructions execution can be stopped by pressing 

key.

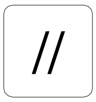
Note 1 MDI running must be performed in MDI mode.

Note 2 The program input in【CUR/MOD】interface is executed prior to that input in MDI mode.

8.3 Words modification and clearing of MDI instructions

If an error occurs during word inputting, 

key can be pressed to cancel the input word by word, or press 

key to cancel the whole block input; if the error is found after the input is finished, reinput the correct words to replace the wrong ones or press 

key to clear all for reinputting.

8.4 Modes changing

When the control is switched to MDI, DNC, Auto, Edit mode during the program execution in Auto, MDI, DNC mode, the system will stop the program execution after the current block is executed.

When the control is switched to Step mode by a dwell during the program execution in Auto, MDI, DNC mode, it will execute the step interruption. See section 5.2 Step interruption. If the control is switched to MPG mode by a dwell, it will execute MPG interruption, see section 6.2 MPG interruption. If the control is switched to JOG mode by a dwell, it will execute manual intervention, see section 4.1.4 Manual interruption.

When the control is switched to Step, MPG, JOG, Machine Zero mode during the program execution in Auto, MDI, DNC mode, the system will execute deceleration and stop.

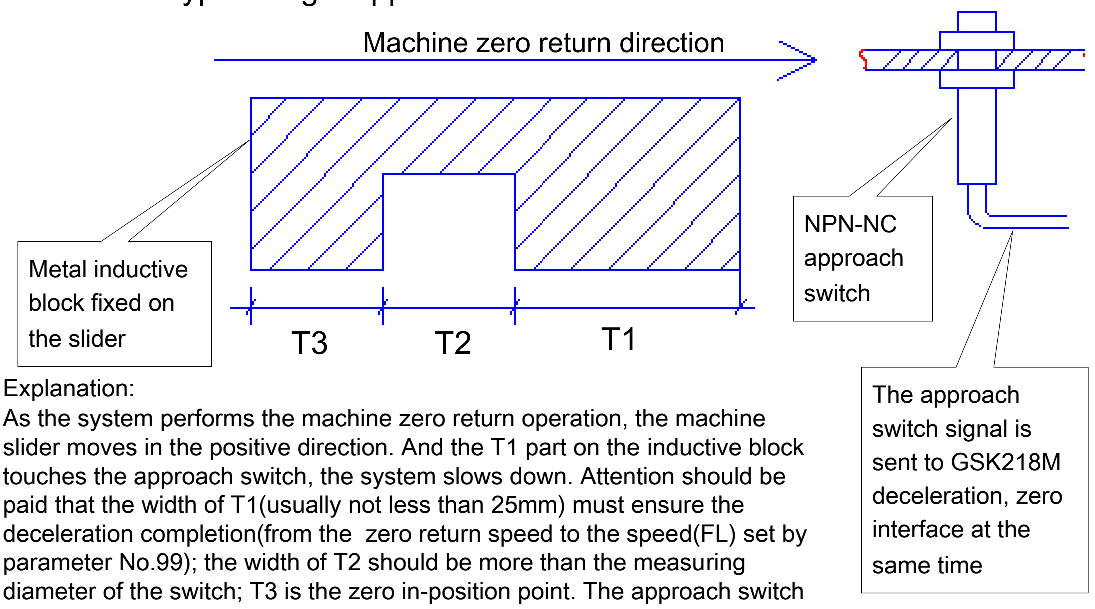
9 Machine Zero Operation

9.1 Conception of machine zero

The machine coordinate system is the inherent coordinate system by machine. Its origin is called mechanical zero (or machine zero), as is called reference point in this manual. It is usually fixed at the maximum stroke point of X axis, Y axis or Z axis. This origin that is a fixed point is set after the machine is designed, manufactured and adjusted. As the machine zero is not confirmed by the CNC system at power-on, the auto or manual machine zero return is usually performed.

The machine zero return has two types: one-revolution-signal, non-one-revolution-signal. It is set by bit parameter No.6#6. For the zero return of the non-one-revolution-signal by the motor, it is classified for the A, B two types. It is set by bit parameter No.6#7.

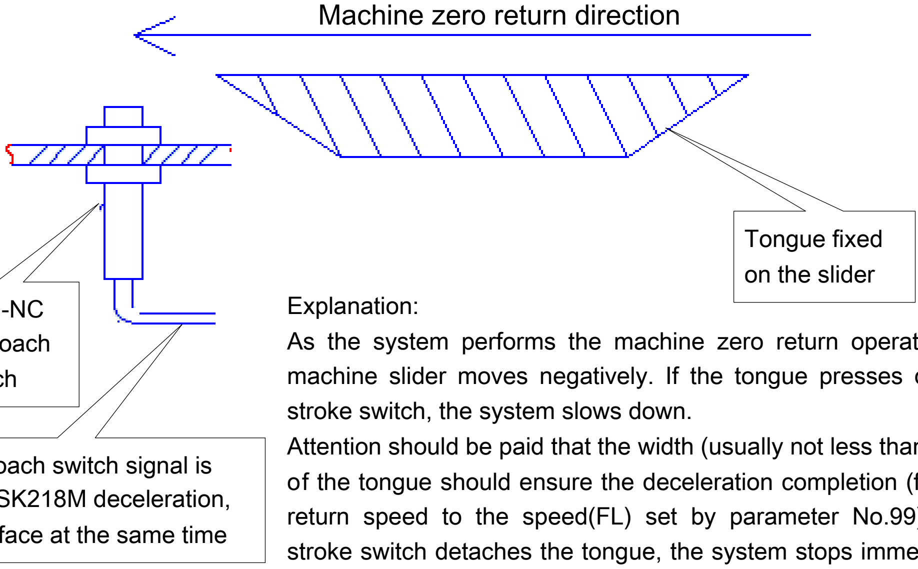
Zero return type using stepper motor with no encoder: A



Explanation:
As the system performs the machine zero return operation, the machine slider moves in the positive direction. And the T1 part on the inductive block touches the approach switch, the system slows down. Attention should be paid that the width of T1(usually not less than 25mm) must ensure the deceleration completion(from the zero return speed to the speed(FL) set by parameter No.99); the width of T2 should be more than the measuring diameter of the switch; T3 is the zero in-position point. The approach switch is NPN normal-close type. i.e.The intervals of T1 and T3 are in break-off state. The intervals of T2 and others are through.

Machine zero type of GSK218M system- A

Zero return type using stepper motor with no encoder: B



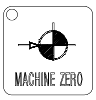
Explanation:

As the system performs the machine zero return operation, The machine slider moves negatively. If the tongue presses down the stroke switch, the system slows down.

Attention should be paid that the width (usually not less than 25mm) of the tongue should ensure the deceleration completion (from zero return speed to the speed(FL) set by parameter No.99); as the stroke switch detaches the tongue, the system stops immediately to complete zero return.

Machine zero type of GSK218M system- B

9.2 Steps for machine zero

1 Press  to enter Machine Zero mode, the characters“machine zero”will be displayed at the down-right of the LCD screen;

2 Select the axis X, Y, or Z for machine zero and its direction is set by bit parameter **No. 7#3**～**N0.7#5**;

3 The machine moves towards the machine zero. Before the deceleration point is reached the machine traverses rapidly(traverse speed set by number parameter No.100～No.102), then moves to the machine zero point(i.e. reference point) by a speed of FL(set by number parameter No.099) if the machine touches the deceleration switch. As the machine zero is reached, the corresponding axis moving stops and the Machine Zero indicator lights up.

9.3 Machine zero steps by program

After the bit parameter No.4#3 is set for 0, the machine zero can be specified by G28 instruction. Because it detects the stroke tongue, this instruction is equivalent to manual machine zero.

Note 1 If the machine zero is not fixed on your machine, don’t perform the machine zero operation.

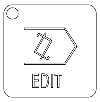
Note 2 The indicator of the corresponding axis lights up when the machine zero is finished.

Note 3 The indicator is gone out on condition that the axis is moved out from the machine zero by the operator.

Note 4 Refer to the machine builder’s manual for the direction of the machine zero (reference point).

10 Edit Operation

10.1 Program edit

The part program edit should be operated in Edit mode. The Edit mode can be entered by pressing  key.

Press  key to enter program page, then press 【◆PROG】 soft key to enter the program edit and modification interface, as is shown in Fig.10-1-1:

```
PROGRAM

O00002;
N0060 X100;
N0120 X0;
N0180 G01 X50 Y50 F2000
N0240 G41 X100 D1
N0300 G01 Y100
N0360 G02 X200 R50
N0420 G01 Y0 F2500
N0180 X0
N0180 Y50

DATA        Ln: 2
                                        EDIT
【BG. EDT】【BG. END】【CHECK】【SAVE】【RETURN】【▶】
```

Fig.10-1-1

Press 【▶】 soft key to enter next page

```
【◀】【REPLACE】【CUT】 【COPY】 【PASTE】【RETURN】【▶】
```

Press 【▶】 soft key to enter next page

```
【◀】【RSTR】                              【RETURN】
```

Press 【◀】 soft key to return to last page

```
【◀】【REPLACE】【CUT】 【COPY】 【PASTE】【RETURN】【▶】
```

The replacement, cut, copy, paste, reset operations etc. can be done by pressing the corresponding soft keys.

The switch of the program must be opened before program edit. See the section 3.5.2 Parameter and program switch in this manual for its operation.

Note The maximum lines a program file contains are 200,000.

10.1.1 Program creation

10.1.1.1 Auto creation of the sequence number

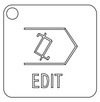
Set the “auto sequence number” for 1 by the steps in section 3.5.1（See Fig. 10-1-1）:

```
SETTING                                   O00002 N0120

 MIRROR X  =   1    (0: OFF 1: ON)
 MIRROR Y  =   1    (0: OFF 1: ON)
 MIRROR Z  =   1    (0: OFF 1: ON)
 CODE      =   1    (0: EIA,  1:  ISO)
 IN UNIT   =   0    (0: MM,   1: INCH)
 I/O CHAN. =   0    ( 0—3  CHANNEL NO. )
 ABS PRG   =   0    (0: ABS, 1: INC)
 AUTO SEQ  =   1    (0: OFF 1: ON)
 SEQ STOP  =   0000 (PROGRAM NO. )
 SEQ STOP  =   0000 (SEQUENCE NO. )
    2006 Y 11 M 14 D      14 H 26 M 45 S
                                      S0000  T0100
 DATA                                     EDIT
   【SETTING】【SWITCH】【G54-G59】【PANEL】【SERVO】 ▶
```

Fig. 10-1-1

Therefore the sequence number will be automatically inserted into the blocks during editing. The incremental amount of the sequence number is set by number parameter No.0210.

10.1.1.2 Program input

1、 Press 

key to enter Edit mode;

2、 Press 

key to enter program page (See Fig. 10-1-2);

```
  PROGRAM                                  O00002 N0180

  O00003;
  N0060 X100;
  N0120 X0;
  N0180 G01 X50 Y50 F2000
  N0240 G41 X100 D1
  N0300 G01 Y100
  N0360 G02 X200 R50
  N0420 G01 Y0 F2500
  N0180 X0
  N0180 Y50

 DATA:          Ln: 3                  S0000  T0100
                                              EDIT
  【◆PRG】  【MDI】【CUR/MOD】【CUR/NXT】【DIR】
```

Fig. 10-1-2

3 Press address key **0**, then key in numerical keys **0**, **0**, **0**, **0**, **2** by sequence (an example by setting up a program named O00002), it displays O00002 behind the data column（See Fig. 10-1-3）:

```
PROGRAM                              O00002  N0180

 O00003;
 N0060 X100;
 N0120 X0;
 N0180 G01 X50 Y50 F2000
 N0240 G41 X100 D1
 N0300 G01 Y100
 N0360 G02 X200 R50
 N0420 G01 Y0 F2500
 N0180 X0
 N0180 Y50

DATA: O0002          Ln: 3          S0000   T0100
                                            EDIT
【◆PRG】 【MDI】【CUR/MOD】【CUR/NXT】【DIR】
```

Fig. 10-1-3

4 Press **EOB** key to set up the new program name, it displays (Fig. 10-1-4**)**:

```
PROGRAM                              O00002  N0180

 O00002;










DATA:                                S0000   T0100
                                            EDIT
【◆PRG】 【MDI】【CUR/MOD】【CUR/NXT】【DIR】
```

Fig. 10-1-4

5 Input the blocks programmed word by word, then press interface switching key (e.g **POSITION** page) or the mode switchover key, the program will be saved automatically and the program input is finished.

Note 1 In Edit mode, only the complete word can be entered. Single letter and numerical number input is not supported by system.

Note 2 If word error is found in program inputting, it can be cancelled by pressing BACKSPACE key to delete one by one or pressing CANCEL key to delete the whole word.

10.1.1.3 Search of sequence number, word and line number

Sequence number search operation is usually used to search for a sequence number in a program so that the execution and edit can be started from the block containing this sequence number. Those blocks that are skipped do not affect the CNC. (This means that the data in the skipped blocks such as coordinates, M, S, T and G codes does not affect the CNC coordinates and modal values.)

If the execution needs to be done from a searched block in a program, specify M, S, T and G codes, coordinates and so forth as required (by MDI) after closely checking the machine and CNC states at that point.

The word search function is used to search a special address word or number in a program, and it is usually used for editing.

Steps for sequence number, line number or word search:

1 Select mode: <Edit >or <Auto>

2 Look up the target program in 【DIR】page;

3 Press ENTER key to enter the target program;

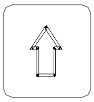
4 Key in the word or sequence number to be searched and press UP or DOWN keys to look for it

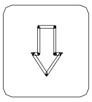
5 If the line number in program is needed to be searched, press SEARCH key, key in the line number to be searched and press ENTER key for confirmation.

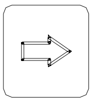
Note The search function is automatically cancelled when the sequence number, word searching reaches the end of the program.

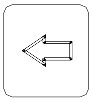
10.1.1.4 Location of the cursor

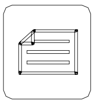
Select Edit mode, then press PROGRAM key to display the program.

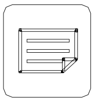
a) Press  key to shift the cursor upward for a line, if the column where the cursor locates exceeds the end column of the last line, the cursor moves to the end of the last line.

b) Press  key to shift the cursor downward for a line, if the column where the cursor locates exceeds the end column of the next line, the cursor moves to the end of the next line.

c) Press  key to move the cursor for a column to the right, if the cursor is at the line end, it moves to the beginning of the next line.

d) Press  key to move the cursor for a column to the left, if the cursor is at the beginning of the line, it moves to the end of the last line.

e) Press  key to scroll the screen upward, and the cursor moves to the first line and first column of the last page; if it is scrolled to the program beginning, the cursor locates at the second line and the first column of the page.

f) Press  key to scroll the screen downward, and the cursor moves to the first line and first column of the next page; if it is scrolled to the program end, the cursor locates at the last line and the first column of the program.

g) Press 

key, the cursor moves to the beginning of the line it locates.

h) Press 

＋

keys, the cursor moves to the beginning of the program.

i) Press 

key, the cursor moves to the end of the line it locates.

j) Press 

＋

keys, the cursor moves to the end of the program.

10.1.1.5 Insertion, deletion and modification of word

Select <EDIT> mode, then press PROGRAM key to display the program. Locate the cursor to the position to be edited.

1. Word insertion

After keying in the data, press INSERT key, the data will be inserted to the left of the cursor.

2. Word deletion

Locate the cursor to the word to be deleted, press DELETE key, the word will be deleted.

If the DELETE key is pressed continuously, the words to the right of the cursor will be deleted.

3. Word modification

Move the cursor to the place to be modified, and key in the new content, then press ALTER key to replace the old content by the new one.

10.1.1.6 Deletion of a single block

Select <EDIT> mode, then press PROGRAM key to display the program. Locate the cursor to the beginning of the block to be deleted. Press N + DELETE keys to delete the block where the cursor locates.

Note N could be keyed in to delete the block whether the block is headed with sequence number.(cursor heading the line)

10.1.1.7 Deletion of multiple blocks

The blocks from the currently displayed word to the specified sequence number block can be deleted.

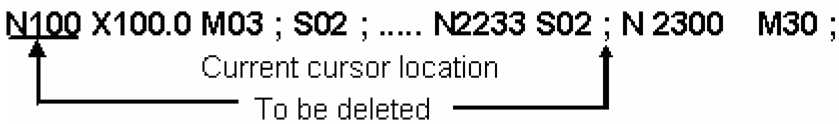


Fig. 10-1-1-7

Select <EDIT> mode, press PROGRAM key to display the program. Locate the cursor to the beginning of the target to be deleted (position of word N100 as figure above), then key in the last word of the multiple blocks to be deleted, e.g. **S02** (as Fig.10-1-1-7 above), press DELETE key to delete the blocks from the current cursor location to the address specified.

Note 1 The blocks that can be deleted are two hundred thousand lines at most.

Note 2 If several words to be deleted are same in program, it will delete the blocks to the word nearest to the cursor location.

10.1.2 Deletion of a single program

The steps for deleting a program in memory are as follows:

a) Select <EDIT> mode;

b) Press PROGRAM key to display the program, there are two ways to delete program;

c) Key in address key O; key in the program name (e.g. For O0001 program), key in the numerical key 0, 0, 0, 0);press DELETE key, the corresponding program in memory will be deleted.

d) Select 【DIR】page in program interface, then select the program name to be deleted by moving cursor and press DELETE key, the program selected will be deleted.

10.1.3 Deletion of all programs

The steps for deleting all programs in the memory are as follows:

a) Select <EDIT> mode;

b) Enter the program page;

c) Key in the address key O;

d) Key in the address keys 9, 9, 9, 9, 9 by sequence；

e) Press DELETE key, all the programs in the memory will be deleted.

10.1.4 Copy of a program

Steps for copying current program and saved for a new name:

a) Select <EDIT> mode;

b) Enter the program page; in 【DIR】page select the program to be copied by cursor keys, then press ENTER key to enter the program page;

c) Press address key 0 and key in the new program number;

d) Press the CHANGE key, the file will be copied and the control enters the new program edit page.

e) Return to 【DIR】page, the name of the new program copied can be viewed.

The copy of the program can also be done in the program edit page:

1 Press address key 0 and key in the new program number;

2 Press the 【COPY】 soft key, the file will be copied and the control enters the new program edit page.

3 Return to 【DIR】page, the name of the new program copied can be viewed.

10.1.5 Copy and paste of blocks

The steps for program copy and paste are as following:

a）Locate the cursor to the beginning of the blocks to be copied;

b）Key in the last character of the blocks to be copied;

c）Press CHANGE key, the blocks from the cursor to the character keyed in will be copied.

d）Locate the cursor to the position to be pasted, press CTRL + P key to complete the paste.

The copy of the program can also be done in the program edit page:

1 Locate the cursor to the beginning of the blocks to be copied;

2 Key in the last character of the blocks to be copied;

3 Press 【COPY】soft key, the blocks from the cursor to the character keyed in will be copied.

4 Locate the cursor to the position to be pasted, press 【PASTE】soft key to complete the paste.

Note If several words to be copied are same in program, it will copy the blocks to the word nearest to the cursor location.

10.1.6 Cut and paste of block

Steps of block cut are as following:

a) Enter the program edit page(as Fig.10-1-1);

b) Locate the cursor to the beginning of the block to be cut;

c) Key in the last character of the block to be cut;

d) Press the 【CUT】soft key , the block will be cut into clip board.

e) Locate the cursor to the position to be pasted, and press【PASTE】soft key, the block will be pasted.

Note If several words to be cut are same in program, it will cut the blocks to the word nearest to the cursor location.

10.1.7 Replacement of the blocks

Steps of block replacement are as following:

a) Enter the program edit page(Fig.10-1-1);

b) Locate the cursor to the character to be altered;

c) Key in the new character;

d) Press the 【REPLACE】soft key, the character where the cursor locates will be replaced by the new one.

The block replacement can also be done by the ALTER key on the panel, see details in Section 10.1.1.5.

10.1.8 Rename of a program

Rename the current program:

a) Select <EDIT> mode;

b) Enter the program page(cursor specifies the program name);

c) Press address key 0, key in the new name;

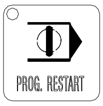
d) Press ALTER key to complete the renaming.

10.1.9 Program restart

The program restart function is used under the situation that accident occurs during running, such as tool braking-off, system restarting after power-off, emergency stop. After the accident is eliminated, this function can be used for returning to program braking-off position to go on execution and retracting to original point by G00.

The steps for program restart are as following:

1 Solve the machine accident such as tool change, offset changing, machine zero.

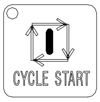
2 In <AUTO> mode, press the  key on the panel.

3 Press  key to enter the program page, then press【RESTART】soft key to enter program restart interface (Fig.10-1-9)


```
PROGRAM RESTART                          O00014   N00012
       (LOADED MODAL)               (CURRENT MODAL)
  G01    G49    F   3000          G00    G49    F   300
  G17    G80    S   1000          G17    G80    S   1000
  G90    G98    M 03, 09          G90    G98    M   30
  G94    G15    T   0003          G94    G15    T   0003
  G54    G50    H   0000          G54    G50    H   0001
  G21    G69    D   0001          G21    G69    D   0001
  G40    G64    .N    20          G40    G64    .N      2
       (DISTANCE)      (ABSOLUTE)   (REM DIST)
  (1)  X   -54.000     X   -54.000    X   0.000
  (2)  Y    12.000     Y     7.800    Y   4.200
  (3)  Z    29.500     Z    29.500    Z   0.000
                                  S00000   T0003
                                      AUTO
       【RSTR】                                【RETURN】
```

Fig.10-1-9

4 In MDI mode, input modes according to the pre-loaded modal values in Fig.10-1-9

5 Press the  key, the control returns to the interruption point by G00 and go on execute the program. This execution can be restarted at any place.

Note 1 The" (1), (2), (3) "headed the coordinates in figure is the moving sequence for the axes moving to the program restarting position. They are set by parameter P376.

Note 2 Check whether the collision occurs when the tool moves to the program restart position, if this possibility exists, move the tool to the place that has no obstructions and restart.

Note 3 When the coordinate axis restart the position moving, switch on the single block running, the tool stops each time it finishes an axis movement.

Note 4 If there is no absolute position detector, it must restart the line reference point return of advancing after the power is on.

Note 5 Don't perform the resetting during the program execution from block research at restarting to restarting, or the restarting must be done from the first step.

10.2 Program management

10.2.1 Program directory search

Press 

key, then press【DIR】soft key to enter the program directory page（See Fig.10-2-1）:

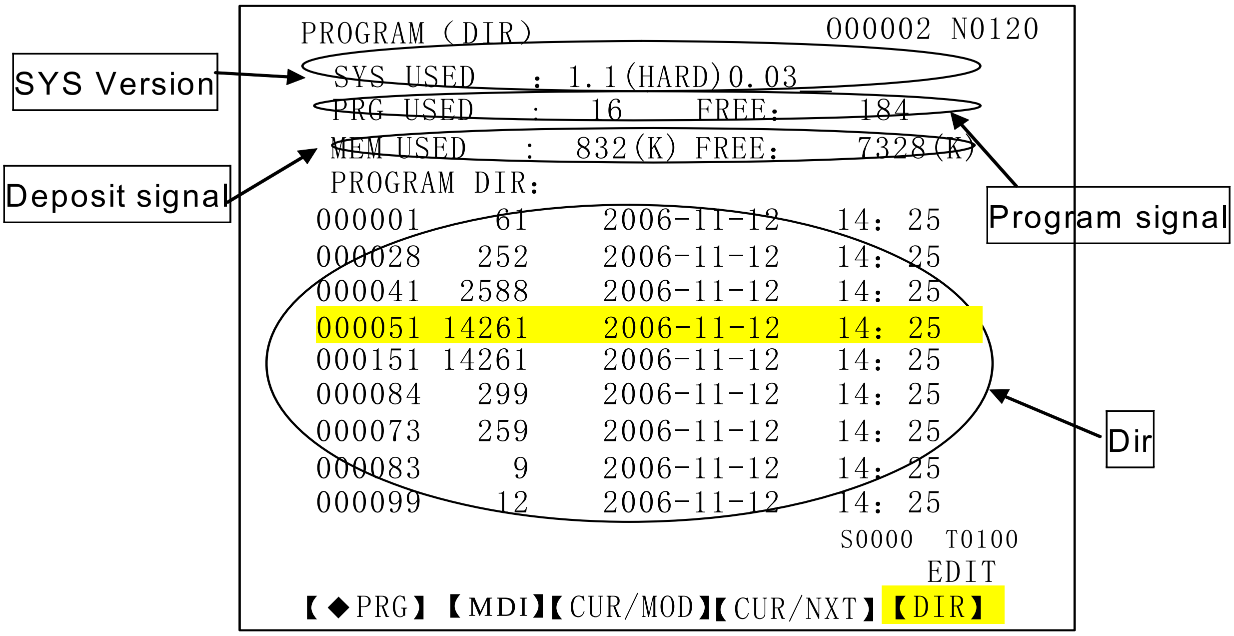


Fig.10-2-1

1）Open the program

Open the program specified: O+sequence number+ENTER key (or EOB key) or sequence number + ENTER key (or EOB key)

In Edit mode, if the sequence number input does not exist, a new program will be created.

2) Deletion of the program:

1. Edit mode Press DEL key to delete the program where cursor locates
2. Edit mode O+ sequence number + DEL or sequence number + DEL

10.2.2 Number of the program stored

The maximum number of the programs stored in this system is 400. Look up in Fig. 10.2.1 above for the message on the number of the program currently stored in the program directory page.

10.2.3 Memory capacity

Look up in Fig.10.2.1 above for the message on memory capacity in the program directory page.

10.2.4 Viewing of the program list

A program directory page can display 10 CNC program names at most. If the CNC programs are over 10, they can't be fully displayed in a page, so press the PAGE key to display the program names on the next page. If the page key is pressed continuously, all the CNC program names will be displayed by cycle on LCD.

Because the programs are listed by their name sizes, press CHANGE key to view them and the programs will be listed by the date sequence with the latest modified program headed.

10.2.5 Program lock

The program switch is set in this system to protect the user programs to be modified by unauthorized personnel. After the program editing, set the program switch for OFF to lock the program. And the program edit is disabled. See Section 3.5.1 for its explanation.

11 Communication

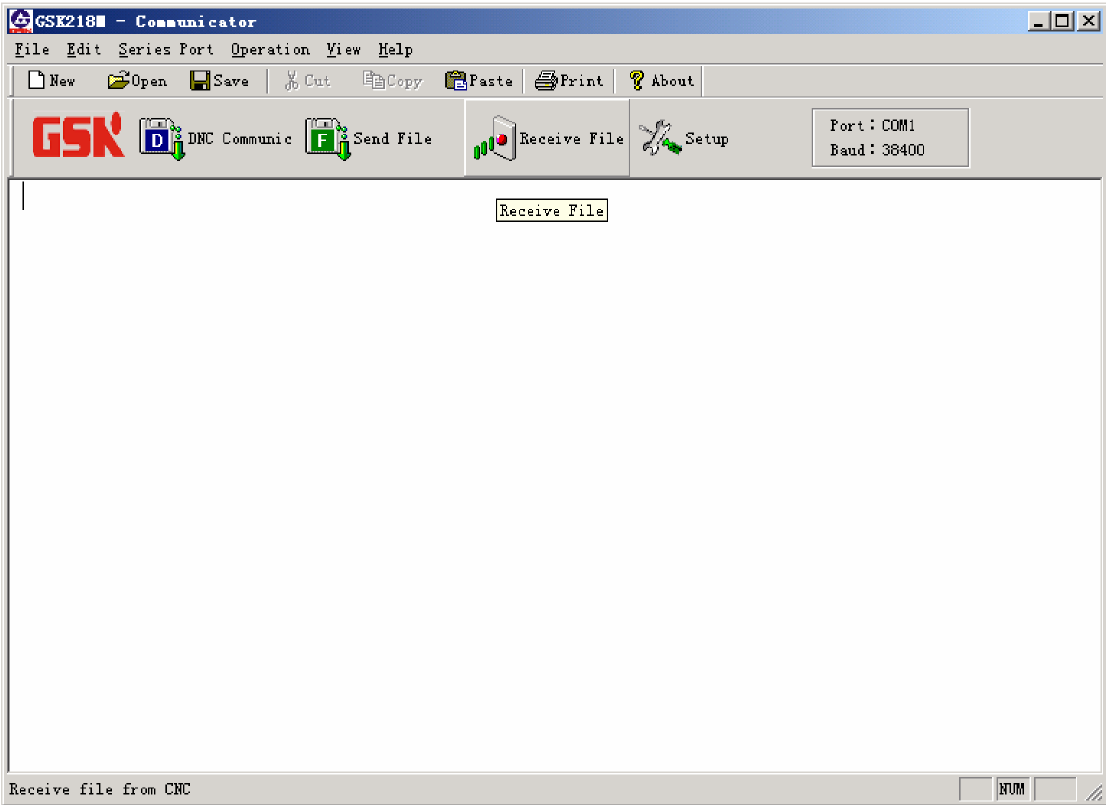
This system can communicate with PC or USB via interface connection.

11.1 Serial communication

The serial communication software of this GSK218M system uses Windows interface, which is used to send and receive files, or execute DNC machining from PC terminal to CNC terminal. This software can be run in Win98, WinMe, WinXP or Win2K operation systems.

11.1.1 Program start

Run the CommGSK218M.exe program directly. The interface of it is as following:



11.1.2 Function introduction

1 File menu

The file menu involves the functions of File Creation, Open, Save, Print and Print setting and the latest the file list etc.

2 Edit menu

The edit menu involves the function such as Cut, Copy, Paste, Retraction, Find, Replace.

3 Serial menu

It is mainly used for the open and setting of the serial ports.

4 Transfer menu

It involves the transfer types of DNC, file sending, file receiving.

5 View menu

It is used for the tool column display and hiding.

6 Help menu

It is used to view the software version.

11.1.3 Software usage

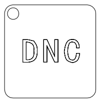
1 DNC transfer

1) Open the program file by the“OPEN”button in File menu or the 

button in tool column, do a further editing by this software if necessary;

2) Open and set the serial port, the default DNC baudrate is 38400, which can be reset by the parameter (refer to GSK218M Operation Manual). The data bit has 8, stop bit has 1, and there is no parity check. Data bit, stop bit and parity check can’t be changed.

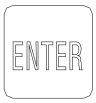
3) The sequence of the 1st and 2nd step can be exchanged which doesn’t affect the following transfer and machining; but the following steps must be operated by sequence, or the transfer and machining will be affected.

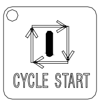
4) As the CNC system and machine are ready, press the 

key on panel;

5) Open the “DNC”item in Transfer type menu or press the DNC transfer button 

in tool column to transfer data;

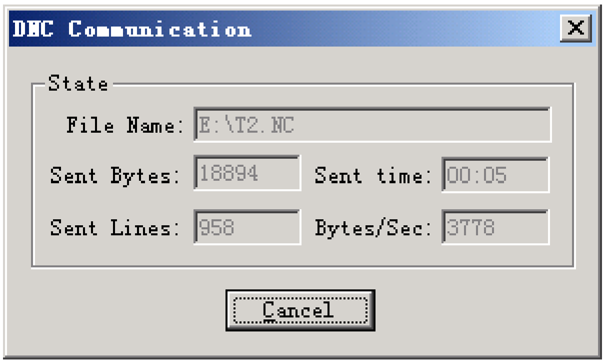
6) As “Sent Bytes” stops, press the 

key on panel to receive data, then press 

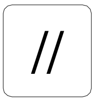
key to start running;

7) Then operate by normal machining pattern;

8) During the transmission, the transfer information involving the file names, bytes, lines transferred and the transmission time and speed (bytes/s) will be displayed, which is shown as following:

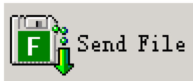


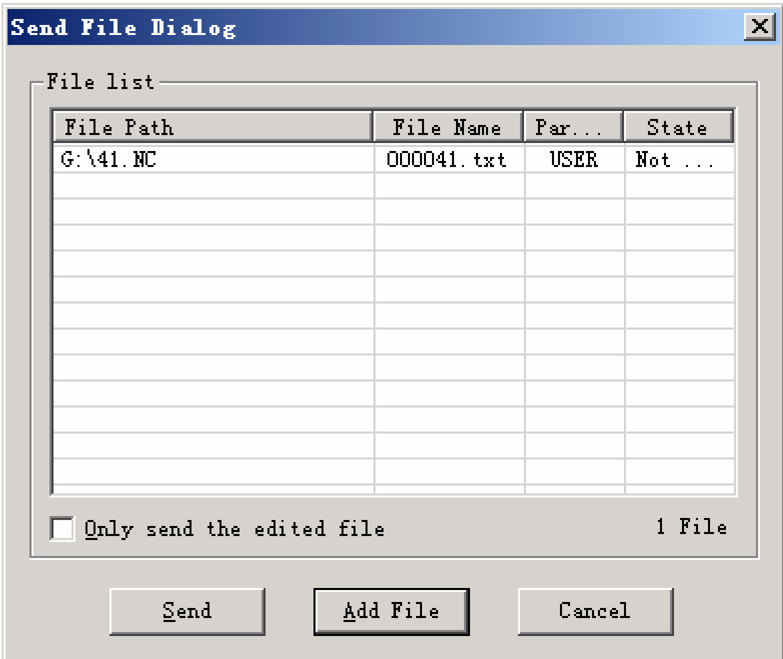
Please don't do other operations by this software except concluding the transmission.

Press  key to cancel the operation after the processing completion.

2 Transfer type for sending files

1) Open and set the serial port with a fixed baudrate 115200, the data bits, stop bit and parity check are identical with that in DNC transmission and it can't be changed.

2) Open the "Send file"item of transfer type menu or press the  button in tool column, the following dialogue block will pop up:



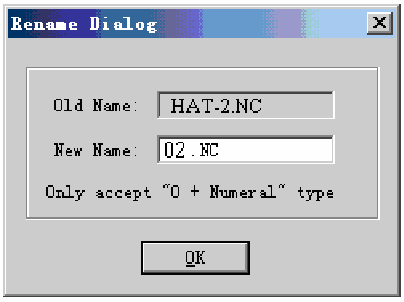
3) Select "Add file"button, the dialogue block "Partition Selection"will appear:

The program files can only be sent to “user part”,while the system configuration and backup files can only be sent to system partition, or they won’ t be identified by system.. To send system configuration and backup files requires the machine builder or dealer level authority, you can enter the relevant password in CNC“password”setting page.

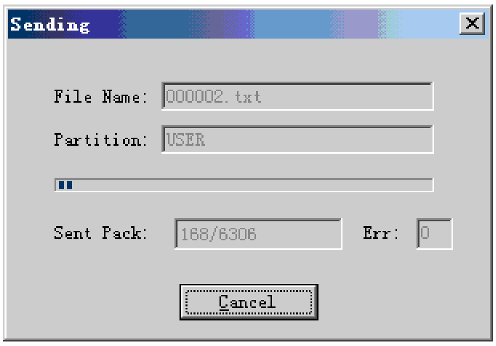
4) “Open file”dialogue block will appear after partition selection, press and hold SHIFT or CTRL key to select multiple files, the maximum 100 files can be selected;

5) Click “Open”button to return to “Send file”dialogue block after the file is selected;

6) The name of the program file sent to user partition should be headed with letter “O”, followed with a number within 5 digits (including 5). Or the following dialogue block will pop up to prompt you to alter the program name:



7) After returning to “Send file” dialogue block, click “Send”button, the file sending will be on, and the following dialogue block will be popped up:



8) Transmission is over.

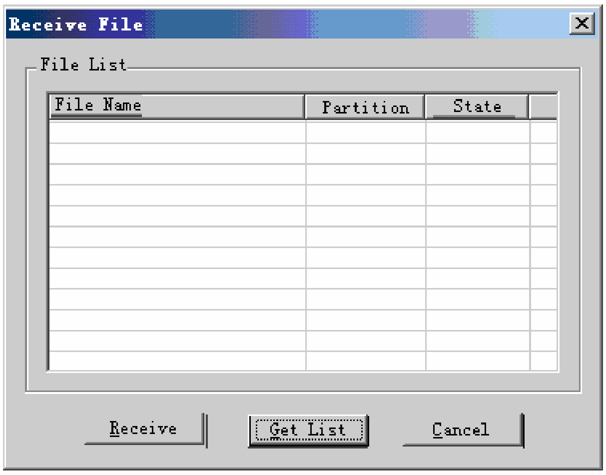
Note The system can’t send files in DNC mode.

3 Transfer type for receiving files

1) Open and set the serial port with a fixed baudrate 115200, the data bits, stop bit and parity check are identical with that in DNC transmission and it can’t be changed.

2) Open the “Receive file”item of transfer type menu or press the 

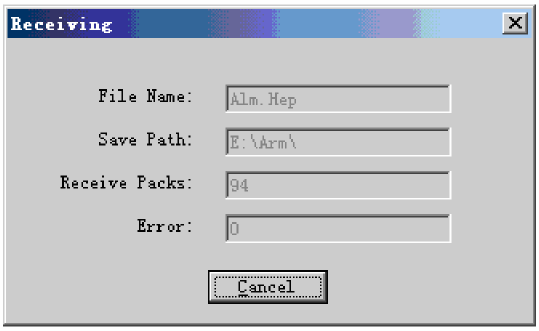
button in tool column, the following dialogue block will pop up:



3) Click “Obtain directory”button, the files in CNC system will be listed;

4) Select the files to be transferred, multiple files can be selected by pressing and holding SHIFT or CTRL key;

5) Click “Start receiving”button for receiving, and the following dialogue block will be popped up;



6) Transmission is over.

Note The system can’t receive files in DNC mode.

11.2 USB communication

11.2.1 General and precautions

1 This U disk system only supports FAT16 file format, if your U disk format is FAT32 or others, please format your U disk to FAT format in advance in your computer, or it won’t be identified by this system.

2 Due to the detachment of the U disk system and the CNC system, the U disk system can’t be entered during the machining, accidents such as the workpiece damage may occur.

3 This U disk system supports hot plug and play for many times, make sure that the USB interface is not inserted by U disk before power on. If inserted, the U disk will not be identified. It is better to insert U disk after the U disk operation interface is entered.

4 When the U disk operation is finished, pull out the U disk after waiting for a while till the indicator for U disk does not blink, it will avoid the U disk data not fully operated.

5 This U disk only displays the text file with the program name O + five-digit number, for example "O000001".

11.2.2 U disk entry

1 Enter into the【DATA】page in <SETTING> page, move the cursor to "CNC part program", in <MDI> mode press ENTER key, then the U disk system begins to start, see details in *OPERATION* Section 3.5.6.

2 After entering into U disk program, insert U disk.

11.2.3 USB part program operation steps

1 To copy CNC programs from U disk to system disk:

a) Press【U disk】soft key to switch to U disk file display;

b) Press UP or DOWN key to select the CNC program in U disk;

c) Press 【COPY】soft key, it prompts at the page bottom"Are you sure to copy this file to system disk?" , press <CANCEL> key to cancel the copy; press <ENTER> key to start the copy, and the page prompts "copy …". After the copy is finished, it prompts at the page bottom "Copy is finished".

Note The operator should note that if there is a file with the same name in CNC storage disk, this file will be covered.

2 To delete files from U disk:

a) Press【U disk】soft key to switch to U disk file display;

b) Press UP or DOWN key to select the CNC program in U disk;

c) Press 【DELETE】soft key, it prompts at the page bottom"Are you sure to delete current file?" , press <CANCEL> key to cancel the deletion; press <ENTER> key to start the deletion.

d) After the file is deleted, there is no name of this file in U disk file display.

3 To copy CNC program from system user disk to U disk:

a) Press【SYSTEM disk】soft key to switch to system disk file display;

b) Press UP or DOWN key to select the CNC program in system disk;

c) Press 【COPY】soft key, it prompts at the page bottom"Are you sure to copy this file to system disk?" , press <CANCEL> key to cancel the copy; press <ENTER> key to start the copy, and the page prompts "copy …". After the copy is finished, it prompts at the page bottom "Copy is finished".

Note The operator should note that if there is a file with the same name in U disk, this file will be covered.

4 To delete files from the system user disk:

a) Press【SYSTEM disk】soft key to switch to system disk file display;

b) Press UP or DOWN key to select the CNC program in system disk;

c) Press 【DELETE】soft key, it prompts at the page bottom"Are you sure to delete current file?" , press <CANCEL> key to cancel the deletion; press <ENTER> key to start the deletion.

d) After the file is deleted, there is no name of this file in system disk file display.

11.2.4 DNC processing operation steps

1 After CNC system start, set I/O channel value for 1 in <SETTING> page; see details in *OPERATION* Section 3.5.1.

2 Insert the U disk.

3 Press 【DNC】 key, it prompts at the page bottom"Please select file in program directory page?", press <PROGRAM> key to enter the program page; press <DIR> soft key to display the U disk programs. Move the cursor to select the processing program, then press <ENTER> key to open this program, and press <CYCLE START> key to execute the DNC processing.

11.2.5 U disk system exit

1 Pull out U disk as the indicator for U disk doesn't blink;

2 Press 【RETURN】 soft key to return to 【DATA】soft page in <SETTING> page.

11.2.6 Remarks for U disk model

Due to the variation of models and drive chips for U disk in market, this CNC system is incapable of identifying all U disks at present. So the U disks shown in the following table are usable via test for this CNC system, but others not shown are untested.

Company	Model	Product standard	Capacity	Chip
Tsinghua e era	Mini	USB2.0	128M	
Lenovo	T108	USB2.0	128M	
Lenovo	B210	USB2.0	256M	
Lenovo		USB2.0	256M	
Manna		USB2.0	128M	
Aigo	Special	USB2.0	1G	
BD		USB1.1	32M	OTi006808

APPENDIX 1

GSK218M PARAMETER LIST

Explanation:

The parameters are classified as following patterns according to the data type:

4 data types and data value range

Data type	Effective data range	Remark
Bit Axis	0 or 1	
Byte Word-axis	-127~127 0~255	Sign is not used in some parameters
Word Word-axis	-32767~32767	Sign is not used in some parameters
Double word Double word-axis	-99999999 ~99999999	

1 For bit and axis parameters, the data are comprised by 8 bits with each bit having different meaning.

2 Axis parameter can be set to each axis separately.

3 The data value range in above table is the common effective range. The specific parameter value range actually differs. See the parameter explanation for details.

Example

（1）Meaning of the bit and axis type parameters

Data number

Data number			**BIT7**	**BIT6**	**BIT5**	**BIT4**	**BIT3**	**BIT2**	**BIT1**	**BIT0**	

（2）Meaning of parameters other than the bit and axis type

0	**2**	**1**	
Data number			Data

Note 1 The blank bits in the parameter explanation and the parameter numbers that are displayed on screen but not in parameter list are reserved for further expansion. They must be set for 0.

Note 2 If 0 or 1 of the parameter is not specified with a meaning, it is assumed that: 1 for affirmative, 0 for negative.

Note 3 If INI is set for 0, in metric input, the parameter setting unit for linear axis is mm,

mm/min; that for rotary axis is deg, deg/min.

If INI is set for 1, in inch input, the parameter setting unit for linear axis is inch, inch/min; that for rotary axis is deg, deg/min.

1 Bit parameter

System parameter number

0	0	0				SEQ			INI	ISO	

ISO =1: ISO code

=0: EIA code

INI =1: Inch input

=0: Metric input

SEQ =1: Automatic sequence number insertion

=0: Not automatic sequence number insertion

If INI is set for 0, in metric input, the basic unit for linear axis is mm, mm/min; that for rotary axis is deg, deg/min.

If INI is set for 1, in inch input, the basic unit for linear axis is inch, inch/min; that for rotary axis is deg, deg/min.

Standard setting: 0 0 0 0 0 0 1 0

System parameter number

0	0	1		SJZ		MIRZ	MIRY	MIRX	SPT		

SPT =1: Spindle control type: I/O point control

=0: Spindle control type: frequency conversion or others

MIRX =1: Mirror setting of X axis: mirror ON

=0: Mirror setting of X axis: mirror OFF

MIRY =1: Mirror setting of Y axis: mirror ON

=0: Mirror setting of Y axis: mirror OFF

MIRZ =1: Mirror setting of Z axis: mirror ON

=0: Mirror setting of Z axis: mirror OFF

SJZ =1: Reference point memorizing: yes

=0: Reference point memorizing: no

Standard setting: 0 0 0 0 0 0 0 0

System parameter number

0	0	2			IOP			ASI1	SB1	ASI0	SB0

SB0 =1: Stop bits of communication channel 0: 2

=0: Stop bits of communication channel 0: 1

ASI0 =1: Data input code of channel 0: ASII

=0: Data input code of channel 0: EIA or ISO

SB1 =1: Stop bits of communication channel 1: 2

=0: Stop bits of communication channel 1: 1

ASI1 =1: Data input code of channel 1: ASII

=0: Data input code of channel 1: EIA or ISO

IOP =1: Program input and output stop: [STOP] key

=0: Program input and output stop: NC reset

Standard setting: 0 0 0 0 1 1 0 0

System parameter number

0	**0**	**3**			**DIR5**	**DIR4**	**DIRZ**	**DIRY**	**DIRX**	**INM**

INM =1: Min. moving unit of linear axis: Inch

=0: Min. moving unit of linear axis: Metric

If INM is set for 0, in metric output, the basic unit for linear axis is mm, mm/min; that for rotary axis is deg, deg/min.

If INM is set for 1, in inch output, the basic unit for linear axis is inch, inch/min; that for rotary axis is deg, deg/min.

DIRX =1: X axis feeding direction

=0: X axis feeding direction reversing

DIRY =1: Y axis feeding direction

=0: Y axis feeding direction reversing

DIRZ =1: Z axis feeding direction

=0: Z axis feeding direction reversing

DIR4 =1: 4^{th} axis feeding direction

=0: 4^{th} axis feeding direction reversing

DIR5 =1: 5^{th} axis feeding direction

=0: 5^{th} axis feeding direction reversing

Standard setting: 0 0 1 1 1 1 1 0

System parameter number

0	**0**	**4**				**XIK**	**AZR**	**SFD**		**JAX**

JAX =1: Synch. controlled axes for manual reference point mode: 1 axes(only zero return mode)

=0: Synch. controlled axes for manual reference point mode: multiple axes

SFD =1: Reference point offset use: yes

=0: Reference point offset use: no

AZR =1: For G28 when reference point not setup: alarm

=0: For G28 when reference point not setup: use tongue

XIK =1: For non-linear positioning axes interlock: all axes stop

=0: For non-linear positioning axes interlock: axes interlock

Standard setting: 0 0 0 1 0 0 0 0

System parameter number

0	0	5		IPR						ISC	

ISC =1: Min. moving unit of 0.0001mm,0.0001deg

=0: Min. moving unit of 0.001mm,0.001deg

IPR =1: Axes min. setting unit is 10 times of min. moving unit: effective

=0: Axes min. setting unit is 10 times of min. moving unit: ineffective

Standard setting: 0 0 0 0 0 0 0 0

System parameter number

0	0	6		MAOB	ZPLS	EDN	EDP				ZRN

ZRN =1: System alarms if instruction other than G28 is specified during auto running.

=0: System doesn't alarm if instruction other than G28 is specified during auto running.

EDP =1: Rapid traverse and cutting effective of each axis external positive deceleration signal

=0: Rapid feed effective of each axis external positive deceleration signal

EDN =1: Rapid traverse and cutting effective of each axis external negative deceleration signal

=0: Rapid feed effective of each axis external negative deceleration signal

ZPLS =1: Zero type selection: one-revolution signal

=0: Zero type selection: non-one-revolution signal

MAOB =1: Zero type selection for non-one-revolution signal: B

=0: Zero type selection for non-one-revolution signal: A

Standard setting: 0 1 0 0 0 0 0 0

System parameter number

0	0	7		ZMI5	ZMI4	ZMIZ	ZMIY	ZMIX		AXS5	AXS4

AXS4 =1: Set 4th axis for linear axis

=0: Set 4th axis for rotary axis

AXS5 =1: Set 5th axis for linear axis

=0: Set 5th axis for rotary axis

ZMIX =1: Direction setting of X axis reference point return: negative

=0: Direction setting of X axis reference point return: positive

ZMIY =1: Direction setting of Y axis reference point return: negative

=0: Direction setting of Y axis reference point return: positive

ZMIZ =1: Direction setting of Z axis reference point return: negative

=0: Direction setting of Z axis reference point return: positive

ZMI4 =1: Direction setting of 4th axis reference point return: negative

=0: Direction setting of 4th axis reference point return: positive

ZMI5 =1: Direction setting of 5th axis reference point return: negative

=0: Direction setting of 5th axis reference point return: positive

Standard setting: 1 0 0 0 0 0 0 0

System parameter number

0	**0**	**8**						**RRLX**	**RABX**	**ROAX**

ROAX =1: Rotation axis cycle effective

=0: Rotation axis cycle ineffective

RABX =1: Rotation direction setting of absolute instruction: instruction value sign

=0: Rotation direction setting of absolute instruction: near to the target

RRLX =1: Moving amount per revolution rounding for relative coordinates

=0: Moving amount per revolution not rounding for relative coordinates

Standard setting: 0 0 0 0 0 0 0 0

System parameter number

0	**0**	**9**			**AWK**			**ZCL**		

ZCL =1: To cancel local coordinate system when performing manual reference point return

=0: Not cancel local coordinate system when performing manual reference point return

AWK =1: To change display immediately when workpiece origin offset is changed

=0: To change next block display when workpiece origin offset is changed

Standard setting: 0 0 0 0 0 0 0 0

0	**1**	**0**				**G52**	**RLC**			

RLC =1: To cancel local coordinate system after resetting

=0: Not cancel local coordinate system after resetting

G52 =1: To add tool compensation vector at local coordinate system setting

=0: Not add tool compensation vector at local coordinate system setting

Standard setting: 0 0 0 0 0 0 0 0

System parameter number

0	1	1		BFA	LZR						OUT2

OUT2 =1: Outer area entry of 2nd stroke unallowed

=0: Inner area entry of 2nd stroke unallowed

LZR =1: To perform travel check before manual reference point return after power-on

=0: Not perform travel check before manual reference point return after power-on

BFA =1: To make alarm after overtravel when overtravel instruction is given

=0: To make alarm before overtravel when overtravel instruction is given

Standard setting: 0 0 0 0 0 0 0 1

System parameter number

0	1	2		FDR	RDR	TDR	RFO			LRP	RPD

RPD =1: Manual rapid effective before reference point return after power-on

=0: Manual rapid ineffective before reference point return after power-on

LRP =1: The positioning(G00) interpolation type is linear.

=0: The positioning(G00) interpolation type is nonlinear.

RFO =1: Rapid feed stop when override is F0.

=0: Rapid feed not stop when override is F0.

TDR =1: Dry run effective during tapping.

=0: Dry run ineffective during tapping.

RDR =1: Dry run effective during cutting feeding.

=0: Dry run ineffective during cutting feeding.

FDR =1: Dry run effective during rapid positioning.

=0: Dry run ineffective during rapid positioning.

Standard setting: 0 0 0 0 0 0 0 0

System parameter number

0	1	3								HPC	NPC

NPC =1: Feed per revolution effective with no position encoder

=0: Feed per revolution ineffective with no position encoder

HPC =1: Position encoder installed.

=0: Position encoder not installed.

Standard setting: 0 0 0 0 0 0 1 0

System parameter number

0	1	4								DLF	HFC

HFC =1: Clamp combined by straight line and arc for helical interpolation feedrate

=0: Clamp by straight line and arc separately for helical interpolation feedrate

DLF =1: Reference point return by manual feed after reference point is setup and memorized

=0: Reference point return by rapid traverse after reference point is setup and memorized

Standard setting: 0 0 0 0 0 0 0 0

System parameter number

0	1	5			PACD	PIIS	PILS	PPCK	ASL	PLAC	STL

STL =1: To select prereading working type

=0: To select non-prereading working type

PLAC =1: Acceleration/deceleration type after forecasting interpolation: exponential

=0: Acceleration /deceleration type after forecasting interpolation: linear

ASL =1: Auto corner deceleration function of forecasting: speed difference control

=0: Auto corner deceleration function of forecasting: angular control

PPCK =1: To perform in-position check by forecasting.

=0: Not perform in-position check by forecasting.

PILS =1: Forecasting interpolation type: circular interpolation

=0: Forecasting interpolation type: linear interpolation

PIIS =1: Overlapping interpolation effective in acceleration/deceleration blocks before forecasting.

=0: Overlapping interpolation ineffective in acceleration /deceleration blocks before forecasting.

PACD =1: Acceleration /deceleration type before forecasting: S

=0: Acceleration /deceleration type before forecasting: linear

Standard setting: 0 0 0 0 0 0 0 1

System parameter number

0	1	6		ALS					FLLS	FBLS	FBOL

FBOL =1: Rapid traverse type: post acceleration /deceleration

=0: Rapid traverse type: pre-acceleration /deceleration

FBLS =1: Pre-acceleration /deceleration type of rapid traverse: S

=0: Pre-acceleration /deceleration type of rapid traverse: linear

FLLS =1: Post acceleration /deceleration type of rapid traverse: exponential

=0: Post acceleration /deceleration type of rapid traverse: linear

ALS =1: Auto corner feed effective.

=0: Auto corner feed ineffective.

Standard setting: 0 0 0 0 0 0 1 0

System parameter number

0	**1**	**7**		**CPCT**	**CALT**	**WLOE**		**HLOE**	**CLLE**	**CBLS**	**CBOL**

CBOL =1: Cutting feed type: post acceleration /deceleration

=0: Cutting feed type: pre-acceleration /deceleration

CBLS =1: Pre-acceleration /deceleration type of cutting feed: S

=0: Pre-acceleration /deceleration type of cutting feed: linear

CLLE =1: Post acceleration /deceleration type of cutting feed: exponential

=0: Post acceleration /deceleration type of cutting feed: linear

HLOE =1: JOG running type: exponential

=0: JOG running type: linear

WLOE =1: MPG running type: exponential

=0: MPG running type: linear

CALT =1: Cutting feed acceleration clamping.

=0: Cutting feed acceleration not clamping.

CPCT =1: To control the in-position precision in cutting feed.

=0: Not control the in-position precision in cutting feed.

Standard setting: 1 0 1 0 0 1 0 1

System parameter number

0	**1**	**8**		**RVCS**			**RBK**	**FFR**			**RVIT**

RVIT =1 To execute next block after compensation as backlash is over value allowable

=0 To execute next block during compensation as backlash is over value allowable

FFR =1: Cutting and rapid traverse both effective in feedforward control.

=0: Cutting feed effective in feedforward control.

RBK =1: To perform backlash compensation for cutting feed and rapid traverse separately

=0: To perform backlash compensation for cutting feed and rapid traverse together

RVCS =1: Backlash compensation type: ascending or decending

=0: Backlash compensation type: fixed frequency

Standard setting: 0 0 0 0 0 0 0 0

System parameter number

0	1	9	IOV		ALMS	ALMS5	ALMS4	ALMSZ	ALMSY	ALMSX

ALMX =1: High level effective of driver alarm

=0: Low level effective of driver alarm

ALMY =1: High level effective of driver alarm

=0: Low level effective of driver alarm

ALMZ =1: High level effective of driver alarm

=0: Low level effective of driver alarm

ALM4 =1: High level effective of driver alarm

=0: Low level effective of driver alarm

ALM5 =1: High level effective of driver alarm

=0: Low level effective of driver alarm

IOV =1: High level effective of override signal

=0: Low level effective of override signal

Standard setting: 0 0 0 0 0 0 0 0

System parameter number

0	2	0						DIT	ITX	ITL

ITL =1: All axes interlock signal effective

=0: All axes interlock signal ineffective

ITX =1: Each axis interlock signal effective

=0: Each axis interlock signal ineffective

DIT =1: Each axis direction interlock signal effective

=0: Each axis direction interlock signal ineffective

Standard setting: 0 0 0 0 0 0 0 0

System parameter number

0	2	1	DISL	ENG	CHI					COR

COR =1: Displayer color setting: black and white

=0: Displayer color setting: chromatic

CHI =1: To set the actual language not for Chinese

=0: To set the actual language for Chinese

ENG =1: To set the actual language for English

=0: To set the actual language not for English

DISL =1: To display company LOGO at start

=0: Not to display company LOGO at start

Standard setting: 0 0 0 0 0 0 0 1

System parameter number

0	**2**	**2**		**DAC**	**DAL**	**DRC**	**DRL**	**PPD**			**MCN**

MCN =1: Machine position displayed by input unit.

=0: Machine position not displayed by input unit.

PPD =1: Relative position display reset when coordinate system is set.

=0: Relative position display not reset when coordinate system is set.

DRL =1: Add tool length compensation in relative position display.

=0: Not add tool length compensation in relative position display.

DRC =1: Add tool radius compensation in relative position display.

=0: Not add tool radius compensation in relative position display.

DAL =1: Add tool length compensation in absolute position display.

=0: Not add tool length compensation in absolute position display.

DAC =1: Add tool radius compensation in absolute position display.

=0: Not add tool radius compensation in absolute position display.

Standard setting: 0 0 0 0 0 0 0 0

System parameter number

0	**2**	**3**			**POSM**		**SUK**		**DNC**		

DNC =1: To clear DNC running program display by pressing reset key

=0: Not clear DNC running program display by pressing reset key

SUK =1: To display program list by program numbers.

=0: To display program list by logging time.

POSM =1: Mode displayed on program monitoring page.

=0: Mode not displayed on program monitoring page.

Standard setting: 0 1 0 0 0 0 0 0

System parameter number

0	**2**	**4**		**RHD**	**NPA**			**SGD**		**SPS**	**SVS**

SVS =1: To display servo setting page.

=0: Not display servo setting page.

SPS =1: To display spindle setting page.

=0: Not display spindle setting page.

SGD =1: To display servo wave.

=0: Not display servo wave.

NPA =1: To switch to alarm page when alarm occurs.

=0: Not switch to alarm page when alarm occurs.

RHD =1: To update the relative position display at MPG interruption.

=0: Not update the relative position display at MPG interruption.

Standard setting: 0 0 0 0 0 0 0 0

System parameter number

0	2	5		ALM	DGN	GRA	SET	OFT	PAR	PRG	POS

POS =1: To switchover page by repressing POSITION key in position page.

=0: Not switchover page by repressing POSITION key in position page.

PRG =1: To switchover page by repressing PROGRAM key in program page.

=0: Not switchover page by repressing PROGRAM key in program page.

PAR =1: To switchover page by repressing PARAMETER key in parameter page.

=0: Not switchover page by repressing PARAMETER key in parameter page.

OFT =1: To switchover page by repressing OFFSET key in offset page.

=0: Not switchover page by repressing OFFSET key in offset page.

SET =1: To switchover page by repressing SET key in set page.

=0: Not switchover page by repressing SET key in set page.

GRA =1: To switchover page by repressing GRAPHIC key in graphic page.

=0: Not switchover page by repressing GRAPHIC key in graphic page.

DGN =1: To switchover page by repressing DIAGNOSE key in diagnosis page.

=0: Not switchover page by repressing DIAGNOSE key in diagnosis page.

ALM =1: To switchover page by repressing ALARM key in alarm page.

=0: Not switchover page by repressing ALARM key in alarm page.

Standard setting: 1 1 1 1 1 1 1 1

System parameter number

0	2	6		INDX	PMC						PETP

PETP =1: To switch to program page by pressing panel Edit key

=0: Not to switch to program page by pressing panel Edit key

PMC =1: To switchover page by repressing PMC key in PMC page.

=0: Not switchover page by repressing PMC key in PMC page.

INDX =1: To switchover page by repressing INDEX key in index page.

=0: Not switchover page by repressing INDEX key in index page.

Standard setting: 1 1 0 0 0 0 0 1

System parameter number

0	2	7			PSK	CPD	NE9	OSR			NE8

NE8 =1: Editting of subprogram with the number 80000 – 89999 unallowed

=0: Editting of subprogram with the number 80000 – 89999 allowed

OSR =1: (O - search) available for program search.

=0: (O - search) not available for program search.

NE9 =1: Editting of Subprogram with the number 90000 – 99999 unallowed

=0: Editting of Subprogram with the number 90000 – 99999 allowed

CPD =1: ENTER key needed when deleting programs.

=0: ENTER key unneeded when deleting programs

PSK =1: Search for programs protected effective.

=0: Search for programs protected ineffective.

Standard setting: 0 0 0 1 0 0 0 1

System parameter number

0	2	8		MCL			MKP				

MKP =1: To clear the program edited when M02, M30 or % is executed in MDI mode.

=0: Not clear the program edited when M02, M30 or % is executed in MDI mode.

MCL =1: To delete the program edited when pressing RESET key in MDI mode.

=0: Not delete the program edited when pressing RESET key in MDI mode.

Standard setting: 0 0 0 0 0 0 0 0

System parameter number

0	2	9			MCM		IWZ	WZO	MCV	GOF	WOF

WOF =1: Tool wear offset input by MDI disabled.

=0: Tool wear offset input by MDI enabled.

GOF =1: Geometric tool offset input by MDI disabled.

=0: Geometric tool offset input by MDI enabled.

MCV =1: Macro variables input by MDI disabled.

=0: Macro variables input by MDI enabled.

WZO =1: Workpiece origin offset input by MDI disabled.

=0: Workpiece origin offset input by MDI enabled.

IWZ =1: Workpiece origin offset input by MDI during dwell disabled.

=0: Workpiece origin offset input by MDI during dwell enabled.

MCM =1: Custom macro input by MDI: MDI type

=0: Custom macro input by MDI: any type

Standard setting: 0 0 0 0 0 0 0 0

System parameter number

0	3	0				ABS	MAB				DPI

DPI =1: Decimal point omitted in programming, default: mm,sec

=0: Decimal point omitted in programming, default: minimum unit

MAB =1: Absolute or relative setting by parameters in MDI mode.

=0: Absolute or relative setting by G90/G91 in MDI mode.

ABS =1: Instructions regarded as absolute in MDI mode.

=0: Instructions regarded as incremental in MDI mode.

Standard setting: 0 0 0 0 0 0 0 0

System parameter number

0	3	1			CLR	G13	G91	G19	G18	G17	G01

G01 =1: G01 at power-on or clearing.

=0: G00 at power-on or clearing.

G17 =1: G17 plane at power-on or clearing.

=0: Not G17 plane at power-on or clearing.

G18 =1: G18 plane at power-on or clearing.

=0: Not G18 plane at power-on or clearing.

G19 =1: G19 plane at power-on or clearing.

=0: Not G19 plane at power-on or clearing.

G91 =1: To set for G91 mode at power-on or clearing.

=0: To set for G90 mode at power-on or clearing.

G13 =1: To set for G13 mode at power-on or clearing.

=0: To set for G12 mode at power-on or clearing.

CLR =1: MDI reset key, to clear external reset signal, make emergency stop

=0: MDI reset key, to reset external signal, make emergency stop

Standard setting: 0 0 0 0 0 0 1 0

System parameter number

0	3	2			AD2	CIR					

CIR =1: Make alarm if distance from start point to center and radius not specified in circular interpolation.

=0: Do not make alarm if distance from start point to center and radius not specified in circular interpolation.

AD2 =1: Make alarm if two or more same addresses are specified in a block.

=0: Do not make alarm if two or more same addresses are specified in a block.

Standard setting: 0 1 0 0 0 0 0 0

System parameter number

0	3	3		M3B	EOR		M30		M02	POL	NOP

NOP =1: Block with only program number, EOB, sequence number ignored

=0: Block with only program number, EOB, sequence number preread

POL =1: To program using decimal point.

=0: To program not using decimal point.

M02 =1: To return to block beginning when M02 is being executed.

=0: Not to return to block beginning when M02 is being executed.

M30 =1: To return to block beginning when M30 is to be executed.

=0: Not to return to block beginning when M30 is to be executed.

EOR =1: To make alarm if “%”occurs in execution.

=0: To reset if “%”occurs in execution.

M3B =1: At most three M codes allowable in a section of program.

=0: Only one M code allowable in a section of program.

Standard setting: 1 0 0 1 0 0 0 0

System parameter number

0	3	4		CFH							DWL

DWL =1: G04 for dwell per revolution in per revolution feed mode.

=0: G04 not for dwell per revolution in per revolution feed mode.

CFH =1: To clear F,H,D codes at reset or emergency stop.

=0: To reserve F,H,D codes at reset or emergency stop.

Standard setting: 0 0 0 0 0 0 0 0

System parameter number

0	3	5	C07	C06	C05	C04	C03	C02	C01	

C01 =1: To clear G codes of 01 group at reset or emergency stop.

=0: To reserve G codes of 01 group at reset or emergency stop.

C02 =1: To clear G codes of 02 group at reset or emergency stop.

=0: To reserve G codes of 02 group at reset or emergency stop.

C03 =1: To clear G codes of 03 group at reset or emergency stop.

=0: To reserve G codes of 03 group at reset or emergency stop.

C04 =1: To clear G codes of 04 group at reset or emergency stop.

=0: To reserve G codes of 04 group at reset or emergency stop.

C05 =1: To clear G codes of 05 group at reset or emergency stop.

=0: To reserve G codes of 05 group at reset or emergency stop.

C06 =1: To clear G codes of 06 group at reset or emergency stop.

=0: To reserve G codes of 06 group at reset or emergency stop.

C07 =1: To clear G codes of 07 group at reset or emergency stop.

=0: To reserve G codes of 07 group at reset or emergency stop.

Standard setting: 0 0 0 0 0 0 0 0

System parameter number

0	3	6	C15	C14	C13	C12	C11	C10	C09	C08

C08 =1: To clear G codes of 08 group at reset or emergency stop.

=0: To reserve G codes of 08 group at reset or emergency stop.

C09 =1: To clear G codes of 09 group at reset or emergency stop.

=0: To reserve G codes of 09 group at reset or emergency stop.

C10 =1: To clear G codes of 10 group at reset or emergency stop.

=0: To reserve G codes of 10 group at reset or emergency stop.

C11 =1: To clear G codes of 11 group at reset or emergency stop.

=0: To reserve G codes of 11 group at reset or emergency stop.

C12 =1: To clear G codes of 12 group at reset or emergency stop.

=0: To reserve G codes of 12 group at reset or emergency stop.

C13 =1: To clear G codes of 13 group at reset or emergency stop.

=0: To reserve G codes of 13 group at reset or emergency stop.

C14 =1: To clear G codes of 14 group at reset or emergency stop.

=0: To reserve G codes of 14 group at reset or emergency stop.

C15 =1: To clear G codes of 15 group at reset or emergency stop.

=0: To reserve G codes of 15 group at reset or emergency stop.

Standard setting: 0 0 0 0 0 0 0 0

System parameter number

0	3	7								**WDIR**	**SCRW**

SCRW =1: To perform pitch compensation.

=0: Not perform pitch compensation.

WDIR =1: Pitch compensation selection: unidirectional

=0: Pitch compensation selection: bidirectional

Standard setting: 0 0 0 0 0 0 0 0

System parameter number

0	3	8		**PG2**	**PG1**						**SAR**

SAR =1: To detect the spindle speed in-position signal

=0: Not detect the spindle speed in-position signal

PG2,PG1：Gear ratio of spindle and position encoder

00 for 1:1; 01 for 2:1; 10 for 4:1; 11 for 8:1

Standard setting: 0 0 0 0 0 0 0 0

System parameter number

0	3	9			**EVO**		**EVR**				**TLC**

TLC =1： Tool length compensation type: B

=0： Tool length compensation type: A

EVR =1： Offset changed effective by respecifying D in tool radius offset

=0： Offset changed effective in next block in tool radius offset.

EVO =1： Offset changed effective by respecifying H in tool length compensation

=0： Offset changed effective in next block in tool length compensation.

Standard setting: 0 0 0 0 0 0 0 1

System parameter number

0	4	0		**ODI**					**CCN**		**SUP**

SUP =1: Start-up type in tool radius compensation: B

=0: Start-up type in tool radius compensation: A

CCN =1: To move to the intermediate point by G28 and cancel compensation in tool radius compensation.

=0: To move to the intermediate point by G28 and reserve compensation in tool radius compensation.

ODI =1: Tool radius compensation value set by diameter

=0: Tool radius compensation value set by radius

Standard setting: 1 0 0 0 0 1 0 1

System parameter number

0	4	1			CN1	G39				CIM	OIM

OIM =1: Metric and inch conversion, automatic tool offset change enabled.

=0: Metric and inch conversion, automatic tool offset change disabled.

CIM =1: Metric and inch conversion,for workpiece coordinate system automatic change.

=0: Metric and inch conversion, workpiece coordinate system not automatic change.

G39 =1: Corner rounding effective in radius compensation.

=0: Corner rounding ineffective in radius compensation.

CN1 =1: Interference check enabled in radius compensation.

=0: Interference check disabled in radius compensation.

Standard setting: 0 1 1 0 0 0 0 0

System parameter number

0	4	2		M5B	M5T	RD2	RD1			EXC	FXY

FXY =1: Axis for drilling canned cycle is the axis selected by program.

=0: Axis for drilling canned cycle is Z.

EXC =1: To specify external action by G81.

=0: To specify drilling canned cycle by G81.

RD1 =1: To set the retraction direction of G76, G87: positive

=0: To set the retraction direction of G76, G87: negative

RD2=1: To set the retraction axis of G76, G87: X

=0: To set the retraction axis of G76, G87: Y

M5T =1: To output M05 at the spindle CW and CCW shift in tapping cycle.

=0: Not to output M05 at the spindle CW and CCW shift in tapping cycle.

M5B =1: To output M05 at the spindle CW and CCW shift in drilling cycle.

=0: Not to output M05 at the spindle CW and CCW shift in drilling cycle.

Standard setting: 0 0 0 0 0 0 0 0

System parameter number

0	4	3								OZA	SIJ

SIJ =1: Displacement in canned cycle specified by I, J, K.

=0: Displacement in canned cycle specified by Q.

OZA =1: To make alarm if cut-in depth is not specified in peck drilling cycle (G73,G83).

=0: Not to make alarm if cut-in depth is not specified in peck drilling cycle (G73, G83).

Standard setting: 0 0 0 0 0 0 0 0

System parameter number

0	4	4			FHD	PCP	DOV			VGR	G84

G84 =1: Use M codes in rigid tapping

=0: Not use M codes in rigid tapping

VGR =1: Arbitrary gear ratio of the spindle and position encoder enabled in rigid tapping.

=0: Arbitrary gear ratio of the spindle and position encoder disabled in rigid tapping.

DOV =1: Override effective during rigid tapping retraction.

=0: Override ineffective during rigid tapping retraction.

PCP =1: To change rigid tapping for high-speed peck drilling cycle.

=0: Not change rigid tapping for high-speed peck drilling cycle.

FHD =1: Single block effective for feed dwell during rigid tapping.

=0: Single block ineffective for feed dwell during rigid tapping.

Standard setting: 0 0 0 0 0 0 0 0

System parameter number

0	4	5					OV3	OVU	TDR		NIZ

NIZ =1: To perform the rigid tapping finishing.

=0: Not perform the rigid tapping finishing.

TDR =1: To use the same time constant during the rigid tapping advance and retraction.

=0: Not use the same time constant during the rigid tapping advance and retraction.

OVU =1: 10% retraction override for rigid tapping.

=0: 1% retraction override for rigid tapping.

OV3 =1: Spindle speed effective by program instruction.

=0: Spindle speed ineffective by program instruction.

Standard setting: 0 0 0 0 0 0 0 0

System parameter number

0	4	6				ORI				SSOG	DGN

DGN =1: Difference of the spindle and the tapping axis errors

=0: Synch error in rigid tapping.

SSOG =1: For servo spindle control at the beginning of rigid tapping.

=0: For following spindle control at the beginning of rigid tapping.

ORI =1: To perform spindle dwell when rigid tapping starts.

=0: Not perform spindle dwell when rigid tapping starts.

Standard setting: 0 0 0 0 0 0 0 0

System parameter number

0	4	7		SCR	XSC	SCLz	SCLY	SCLx			R1N

R1N =1: Rotational angle of coordinate rotation: by G90/G91 instruction

=0: Rotational angle of coordinate rotation: by absolute instruction

SCLx =1: X axis scaling effective.

=0: X axis scaling ineffective.

SCLY =1: Y axis scaling effective.

=0: Y axis scaling ineffective.

SCLz =1: Z axis scaling effective.

=0: Z axis scaling ineffective.

XSC =1: Each axis scaling mirror effective.

=0: Each axis scaling mirror ineffective.

SCR =1: Scaling override unit: 0.001

=0: Scaling override unit: 0.0001

XSC =1: Axes scaling override by I, J, K

=0: Axes scaling override by P instruction

Standard setting: 0 1 1 1 1 0 0 1

System parameter number

0	4	8								PD1	MDL

MDL =1: G codes of unidirectional positioning set for modal

=0: G codes of unidirectional positioning not set for modal.

PD1 =1: To perform in-position check for unidirectional positioning.

=0: Not perform in-position check for unidirectional positioning.

Standard setting: 0 0 0 0 0 0 0 0

System parameter number

0	5	0		IDX	SIM		G90	INC	ABS	REL	DOP

DOP =1: Use calculator for indexing table decimal point input

=0: Not use calculator for indexing table decimal point input

REL =1: Relative position display setting of indexing table: within 360°

=0: Relative position display setting of indexing table: beyond 360°

ABS =1: Use 360°rotation for indexing table absolute coordinate.

=0: Not use 360°rotation for indexing table absolute coordinate.

INC =1: Select the latest rotation direction.

=0: Not select the latest rotation direction.

G90 =1: Indexing instruction: absolute instruction.

=0: Indexing instruction: specified by G90/G91.

SIM =1: Make alarm if indexing instruction and other axes instructions are in same block.

=0: Do not make alarm if indexing instruction and other axes instructions are in same block.

IDX =1: B type by indexing sequence of indexing table.

=0: A type by indexing sequence of indexing table.

Standard setting: 0 1 0 0 0 0 0 0

System parameter number

0	5	1				SBM					G67

G67 =1: To make alarm if macro instructions cancelled by non-macro modal instrucions.

=0: Do not make alarm if macro instructions cancelled by non-macro modal instrucions.

SBM =1: Single block allowed in macro statement.

=0: Single block unallowed in macro statement.

Standard setting: 0 0 0 0 0 0 0 0

System parameter number

0	5	2		CLV	CCV						

CCV =1: Macro common variables #100 - #199 clearing after reset.

=0: Macro common variables #100 - #199 not clearing after reset.

CLV =1: Macro local variables #1 - #50 clearing after reset.

=0: Macro local variables #1 - #50 not clearing after reset.

Standard setting: 0 0 0 0 0 0 0 0

System parameter number

0	5	3						LAD3	LDA2	LAD1	LAD0

LAD0~LAD3 They are binary combined parameters. If it is 0, magazine use not calling macro; if they are 1～15, magazine use calling O90001~O900015 respectively.

Standard setting: 0 0 0 0 0 0 0 1

System parameter number

0	5	4								
				ZNM						

ZNM =1: To amplify the center and override display.

=0: Not to amplify the center and override display.

Standard setting: 0 0 0 0 0 0 0 0

System parameter number

0	5	5								
										CANT

CANT =1: Automatic clearing for single piece.

=0: Not automatic clearing for single piece.

Standard setting: 0 0 0 0 0 0 0 0

System parameter number

0	5	6								
			HNGD				HCL	IHD		HPF

HPF =1: To select full running for MPG moving.

=0: Not select full running for MPG moving.

IHD =1: MPG moving is output unit.

=0: MPG moving is input unit.

HCL =1: Clearing MPG interruption display by soft keys enabled.

=0: Clearing MPG interruption display by soft keys disabled.

HNGD =1: Axes moving direction are identical with MPG rotation direction.

=0: Axes moving direction are not identical with MPG rotation direction.

Standard setting: 0 0 0 0 0 0 0 1

System parameter number

0	5	7								
			MMDI	OP7	OP6					OP1

OP1 =1: Mode selection by soft keys enabled.

=0: Mode selection by soft keys disabled.

OP6 =1: Block skip, single block, machine lock, and dry run operation by soft keys enabled.

=0: Block skip, single block, machine lock, and dry run operation by soft keys disabled.

OP7 =1: Cycle start and dwell operation by soft keys enabled.

=0: Cycle start and dwell operation by soft keys disabled.

MMDI =1: Panel keyboard can be replaced by soft keyboard.

=0: Panel keyboard can not be replaced by soft keyboard.

Standard setting: 0 0 0 0 0 0 0 0

System parameter number

0	**5**	**8**	**MOU**	**MOA**						

MOA =1: Outputting all when program restarts.

=0: Outputting the last M, S, T, B codes when program restarts.

MOU =1: To output M,S,T,B codes when program restarts.

=0: Not output M, S, T, B codes when program restarts.

Standard setting: 0 0 0 0 0 0 0 0

System parameter number

0	**5**	**9**	**OTOP**	**OTOP**	**LOPT**		**AOV**	**DEC**		**OHPG**

OHPG =1: Feed by external handwheel.

=0: Feed not by external handwheel.

DEC =1: Use external deceleration.

=0: Not use external deceleration.

AOV =1: Use automatic corner override.

=0: Not use automatic corner override.

LOPT =1: Use external operator panel lock.

=0: Not use external operator panel lock.

OTOP =1: Use external editing lock.

=0: Not use external editing lock.

OTOP =1: Use external start and stop.

=0: Not use external start and stop.

Standard setting: 0 0 0 0 0 0 0 0

System parameter number

0	**6**	**0**			**SCL**	**SPK**	**IXC**			**TLF**

TLF =1: Use tool life management.

=0: Not use tool life management.

IXC =1: Use indexing table.

=0: Not use indexing table.

SPK =1: Use small peck drilling cycle.

=0: Not use small peck drilling cycle.

SCL =1: Use scaling.

=0: Not use scaling.

Standard setting: 0 0 1 0 0 0 0 0

System parameter number

0	6	1		FALM	LALM	EALM	SALM	SYC			SSC

SSC =1: To use constant surface speed control.

=0: Not use constant surface speed control.

SYC =1: Use synch spindle.

=0: Not use synch spindle.

SALM =1: Spindle driver alarm ignored.

=0: Spindle driver alarm not ignored.

EALM =1: Emergency stop alarm ignored.

=0: Emergency stop alarm not ignored.

LALM =1: Limit alarm ignored.

=0: Limit alarm not ignored.

FALM =1: Feed axis driver alarm ignored.

=0: Feed axis driver alarm not ignored.

Standard setting: 0 0 0 0 0 0 0 0

2 Number parameter

Parameter number	Definition	Default value
00000	I/O channel, input and output device selection.	0

Setting range: 0～3

0001	Baudrate of communication channel 0	38400

Setting range: 0～115200 (unit: BPS)

0002	Baudrate of communication channel 1	115200

Setting range: 0～115200 (unit: BPS)

0003	Waiting time of screen protection (minute)	0

Setting range: 0～999

0004	System interpolation period (1, 2, 4, 8ms)	1

Setting range: 1～8

0005	Axes controlled by CNC	3

Setting range: 3～5

0006	Program axis name of rotary axis	0

When the CNC controlled axes is set for 4, the program axes names of rotary axes are set for 0, 1, 2, the 4th axis name is displayed for A, B, C respectively.

When the CNC controlled axes is set for 5, the program axes names of rotary axes are set for 1, 2, 12, 10, 20, 21, the 4th and 5th axis names are displayed for AB, AC, BC, BA, CA, CB respectively.

0007	Axis name setting in primary coordinate system	0

0008	Servo axis number of each axis	0

0010	External workpiece origin offset amount along X axis	0.0000

Setting range: -9999.9999～9999.9999 (mm)

0011	External workpiece origin offset amount along Y axis	0.0000

Setting range: -9999.9999～9999.9999 (mm)

0012	External workpiece origin offset amount along Z axis	0.0000

Setting range: -9999.9999～9999.9999 (mm)

0013	External workpiece origin offset amount along 4th axis	0.0000

Setting range: -9999.9999～9999.9999 (mm)

0014	External workpiece origin offset amount along 5th axis	0.0000

Setting range: -9999.9999～9999.9999 (mm)

0015	Origin offset amount of workpiece coordinate system 1 (G54_X)	0.0000

Setting range: -9999.9999～9999.9999 (mm)

0016	Origin offset amount of workpiece coordinate system 1 (G54_Y)	0.0000

Setting range: -9999.9999～9999.9999 (mm)

0017	Origin offset amount of workpiece coordinate system 1 (G54_Z)	0.0000

Setting range: -9999.9999～9999.9999 (mm)

0018	Origin offset amount of workpiece coordinate system 1 (G54_4TH)	0.0000

Setting range: -9999.9999～9999.9999 (mm)

0019	Origin offset amount of workpiece coordinate system 1 (G54_5TH)	0.0000

Setting range: -9999.9999～9999.9999 (mm)

0020	Origin offset amount of workpiece coordinate system 2 (G55_X)	0.0000

Setting range: -9999.9999～9999.9999 (mm)

0021	Origin offset amount of workpiece coordinate system 2 (G55_Y)	0.0000

Setting range: -9999.9999～9999.9999 (mm)

0022	Origin offset amount of workpiece coordinate system 2 (G55_Z)	0.0000

Setting range: -9999.9999～9999.9999 (mm)

0023	Origin offset amount of workpiece coordinate system 2 (G55_4TH)	0.0000

Setting range: -9999.9999～9999.9999 (mm)

0024	Origin offset amount of workpiece coordinate system 2 (G55_5TH)	0.0000

Setting range: -9999.9999～9999.9999 (mm)

0025	Origin offset amount of workpiece coordinate system 3 (G56_X)	0.0000

Setting range: -9999.9999～9999.9999 (mm)

0026	Origin offset amount of workpiece coordinate system 3 (G56_Y)	0.0000

Setting range: -9999.9999～9999.9999 (mm)

0027	Origin offset amount of workpiece coordinate system 3 (G56_Z)	0.0000

Setting range: -9999.9999～9999.9999 (mm)

0028	Origin offset amount of workpiece coordinate system 3 (G56_4TH)	0.0000

Setting range: -9999.9999～9999.9999 (mm)

0029	Origin offset amount of workpiece coordinate system 3 (G56_5TH)	0.0000

Setting range: -9999.9999～9999.9999 (mm)

0030	Origin offset amount of workpiece coordinate system 4 (G57_X)	0.0000

Setting range: -9999.9999～9999.9999 (mm)

0031	Origin offset amount of workpiece coordinate system 4 (G57_Y)	0.0000

Setting range: -9999.9999～9999.9999 (mm)

0032	Origin offset amount of workpiece coordinate system 4 (G57_Z)	0.0000

Setting range: -9999.9999～9999.9999 (mm)

0033	Origin offset amount of workpiece coordinate system 4 (G57_4TH)	0.0000

Setting range: -9999.9999～9999.9999 (mm)

0034	Origin offset amount of workpiece coordinate system 4 (G57_5TH)	0.0000

Setting range: -9999.9999～9999.9999 (mm)

0035	Origin offset amount of workpiece coordinate system 5 (G58_X)	0.0000

Setting range: -9999.9999～9999.9999 (mm)

0036	Origin offset amount of workpiece coordinate system 5 (G58_Y)	0.0000

Setting range: -9999.9999～9999.9999 (mm)

0037	Origin offset amount of workpiece coordinate system 5 (G58_Z)	0.0000

Setting range: -9999.9999～9999.9999 (mm)

0038	Origin offset amount of workpiece coordinate system 5 (G58_4TH)	0.0000

Setting range: -9999.9999～9999.9999 (mm)

0039	Origin offset amount of workpiece coordinate system 5 (G58_5TH)	0.0000

Setting range: -9999.9999～9999.9999 (mm)

0040	Origin offset amount of workpiece coordinate system 6 (G59_X)	0.0000

Setting range: -9999.9999～9999.9999 (mm)

0041	Origin offset amount of workpiece coordinate system 6 (G59_Y)	0.0000

Setting range: -9999.9999～9999.9999 (mm)

0042	Origin offset amount of workpiece coordinate system 6 (G59_Z)	0.0000

Setting range: -9999.9999～9999.9999 (mm)

0043	Origin offset amount of workpiece coordinate system 6 (G59_4TH)	0.0000

Setting range: -9999.9999～9999.9999 (mm)

0044	Origin offset amount of workpiece coordinate system 6 (G59_5TH)	0.0000

Setting range: -9999.9999～9999.9999 (mm)

0045	X coordinate of the 1st reference point in machine coordinate system	0.0000

Setting range: -9999.9999～9999.9999 (mm)

0046	Y coordinate of the 1st reference point in machine coordinate system	0.0000

Setting range: -9999.9999～9999.9999 (mm)

0047	Z coordinate of the 1st reference point in machine coordinate system	0.0000

Setting range: -9999.9999～9999.9999 (mm)

0048	4TH coordinate of the 1st reference point in machine coordinate system	0.0000

Setting range: -9999.9999～9999.9999 (mm)

0049	5TH coordinate of the 1st reference point in machine coordinate system	0.0000

Setting range: -9999.9999～9999.9999 (mm)

0050	X coordinate of the 2nd reference point in machine coordinate system	0.0000

Setting range: -9999.9999～9999.9999 (mm)

0051	Y coordinate of the 2nd reference point in machine coordinate system	0.0000

Setting range: -9999.9999～9999.9999 (mm)

0052	Z coordinate of the 2nd reference point in machine coordinate system	0.0000

Setting range: -9999.9999～9999.9999 (mm)

0053	4TH coordinate of the 2nd reference point in machine coordinate system	0.0000

Setting range: -9999.9999～9999.9999 (mm)

0054	5TH coordinate of the 2nd reference point in machine coordinate system	0.0000

Setting range: -9999.9999～9999.9999 (mm)

0055	X coordinate of the 3rd reference point in machine coordinate system	0.0000

Setting range: -9999.9999～9999.9999 (mm)

0056	Y coordinate of the 3rd reference point in machine coordinate system	0.0000

Setting range: -9999.9999～9999.9999 (mm)

0057	Z coordinate of the 3rd reference point in machine coordinate system	0.0000

Setting range: -9999.9999～9999.9999 (mm)

0058	4TH coordinate of the 3rd reference point in machine coordinate system	0.0000

Setting range: -9999.9999～9999.9999 (mm)

0059	5TH coordinate of the 3rd reference point in machine coordinate system	0.0000

Setting range: -9999.9999～9999.9999 (mm)

0060	X coordinate of the 4th reference point in machine coordinate system	0.0000

Setting range: -9999.9999～9999.9999 (mm)

0061	Y coordinate of the 4th reference point in machine coordinate system	0.0000

Setting range: -9999.9999～9999.9999 (mm)

0062	Z coordinate of the 4th reference point in machine coordinate system	0.0000

Setting range: -9999.9999～9999.9999 (mm)

0063	4TH coordinate of the 4th reference point in machine coordinate system	0.0000

Setting range: -9999.9999～9999.9999 (mm)

0064	5TH coordinate of the 4th reference point in machine coordinate system	0.0000

Setting range: -9999.9999～9999.9999 (mm)

0065	Moving amount per revolution of rotary axis	0.0000

Setting range: 0～999.9999 (deg)

0066	Negative X axis stroke coordinate of storage travel detection 1	-9999

Setting range: -9999.9999～9999.9999 (mm)

0067	Positive X axis stroke coordinate of storage travel detection 1	9999

Setting range: -9999.9999～9999.9999 (mm)

0068	Negative Y axis stroke coordinate of storage travel detection 1	-9999

Setting range: -9999.9999～9999.9999 (mm)

0069	Positive Y axis stroke coordinate of storage travel detection 1	9999

Setting range: -9999.9999～9999.9999 (mm)

0070	Negative Z axis stroke coordinate of storage travel detection 1	-9999

Setting range: -9999.9999～9999.9999 (mm)

0071	Positive Z axis stroke coordinate of storage travel detection 1	9999

Setting range: -9999.9999～9999.9999 (mm)

0072	Negative 4TH axis stroke coordinate of storage travel detection 1	-9999

Setting range: -9999.9999～9999.9999 (mm)

0073	Positive 4TH axis stroke coordinate of storage travel detection 1	9999

Setting range: -9999.9999～9999.9999 (mm)

0074	Negative 5TH axis stroke coordinate of storage travel detection 1	-9999

Setting range: -9999.9999～9999.9999 (mm)

0075	Positive 5TH axis stroke coordinate of storage travel detection 1	9999

Setting range: -9999.9999～9999.9999 (mm)

0076	Negative X axis stroke coordinate of storage travel detection 2	-9999

Setting range: -9999.9999～9999.9999 (mm)

0077	Positive X axis stroke coordinate of storage travel detection 2	9999

Setting range: -9999.9999～9999.9999 (mm)

0078	Negative Y axis stroke coordinate of storage travel detection 2	-9999

Setting range: -9999.9999～9999.9999 (mm)

0079	Positive Y axis stroke coordinate of storage travel detection 2	9999

Setting range: -9999.9999～9999.9999 (mm)

0080	Negative Z axis stroke coordinate of storage travel detection 2	-9999

Setting range: -9999.9999～9999.9999 (mm)

0081	Negative Z axis stroke coordinate of storage travel detection 2	9999

Setting range: -9999.9999～9999.9999 (mm)

0082	Negative 4TH axis stroke coordinate of storage travel detection 2	-9999

Setting range: -9999.9999～9999.9999 (mm)

0083	Positive 4TH axis stroke coordinate of storage travel detection 2	9999

Setting range: -9999.9999～9999.9999 (mm)

0084	Negative 5TH axis stroke coordinate of storage travel detection 2	-9999

Setting range: -9999.9999～0.0000 (mm)

0085	Positive 5TH axis stroke coordinate of storage travel detection 2	9999

Setting range: -9999.9999～9999.9999 (mm)

0086	Dry run speed	5000

Setting range: 0～9999 (mm/min)

0087	Cutting feedrate at power-on	300

Setting range: 0～9999 (mm/min)

0088	Rapid traverse speed of X axis	5000

Setting range: 0～9999 (mm/min)

0089	Rapid traverse speed of Y axis	5000

Setting range: 0～9999 (mm/min)

0090	Rapid traverse speed of Z axis	5000

Setting range: 0～9999 (mm/min)

0091	Rapid traverse speed of 4TH axis	5000

Setting range: 0～9999 (mm/min)

0092	Rapid traverse speed of 5TH axis	5000

Setting range: 0～9999 (mm/min)

0093	F0 rapid override of axis (for all axes)	30

Setting range: 0～1000 (mm/min)

0094	Maximum feedrate (for all axes)	8000

Setting range: 300～9999(mm/min) Maximum control speed in non-forecast mode

0095	Minimum feedrate (for all axes)	0

Setting range: 0～300(mm/min) Minimum control speed in non-forecast mode

0096	Maximum speed in forecasting control mode (for all axes)	6000

Setting range: 300～9999(mm/min)

0097	Minimum speed in forecasting control mode (for all axes)	0

Setting range: 0～300(mm/min)

0098	Feedrate of manual continuous feed for axes (JOG)	2000

Setting range: 0～9999 (mm/min)

0099	Speed(FL) of reference point return (for all axes)	40

Setting range: 0～9999 (mm/min)

0100	X axis reference point return speed	4000

Setting range: 0～9999 (mm/min)

0101	Y axis reference point return speed	4000

Setting range: 0～9999 (mm/min)

0102	Z axis reference point return speed	4000

Setting range: 0～9999 (mm/min)

0103	4^{TH} axis reference point return speed	4000

Setting range: 0～9999 (mm/min)

0104	5^{TH} axis reference point return speed	4000

Setting range: 0～9999 (mm/min)

0105	L type time constant of pre-acceleration /deceleration of rapid X axis	100

Setting range: 0～400（ms）

0106	L type time constant of pre-acceleration /deceleration of rapid Y axis	100

Setting range: 0～400（ms）

0107	L type time constant of pre-acceleration /deceleration of rapid Z axis	100

Setting range: 0～400（ms）

0108	L type time constant of pre-acceleration /deceleration of rapid 4TH axis	100

Setting range: 0～400（ms）

0109	L type time constant of pre-acceleration /deceleration of rapid 5TH axis	100

Setting range: 0～400（ms）

0110	S type time constant of pre-acceleration /deceleration of rapid X axis	100

Setting range: 0～400（ms）

0111	S type time constant of pre-acceleration /deceleration of rapid Y axis	100

Setting range: 0～400（ms）

0112	S type time constant of pre-acceleration /deceleration of rapid Z axis	100

Setting range: 0～400（ms）

0113	S type time constant of pre-acceleration /deceleration of rapid 4TH axis	100

Setting range: 0～400（ms）

0114	S type time constant of pre-acceleration /deceleration of rapid 5TH axis	100

Setting range: 0～400（ms）

0115	L type time constant of post acceleration /deceleration of rapid X axis	80

Setting range: 0～400（ms）

0116	L type time constant of post acceleration /deceleration of rapid Y axis	80

Setting range: 0～400（ms）

0117	L type time constant of post acceleration /deceleration of rapid Z axis	80

Setting range: 0～400（ms）

0118	L type time constant of post acceleration /deceleration of rapid 4TH axis	80

Setting range: 0～400（ms）

0119	L type time constant of post acceleration /deceleration of rapid 5TH axis	80

Setting range: 0～400（ms）

0120	E type time constant of post acceleration /deceleration of rapid X axis	60

Setting range: 0～400（ms）

0121	E type time constant of post acceleration /deceleration of rapid Y axis	60

Setting range: 0～400（ms）

0122	E type time constant of post acceleration /deceleration of rapid Z axis	60

Setting range: 0～400（ms）

0123	E type time constant of post acceleration /deceleration of rapid 4TH axis	60

Setting range: 0～400（ms）

0124	E type time constant of post acceleration /deceleration of rapid 5TH axis	60

Setting range: 0～400（ms）

0125	L type time constant of pre-acceleration /deceleration of cutting feed	100

Setting range: 0～400（ms）

0126	S type time constant of pre-acceleration /deceleration of cutting feed	100

Setting range: 0～400（ms）

0127	L type time constant of post acceleration /deceleration of cutting feed	80

Setting range: 0～400（ms）

0128	E type time constant of post acceleration /deceleration of cutting feed	60

Setting range: 0～400（ms）

0129	FL speed of exponential acceleration /deceleration	10

Setting range: 0～9999（mm/min）

0130	Maximum blocks merged in pre-interpolation	2

Setting range: 0～10

0131	In-position precision of cutting feed	0.03

Setting range: 0～0.5 (mm)

0132	Control precision of circular interpolation	0.03

Setting range: 0～0.5 (mm)

0133	Contour control precision of pre-interpolation	0.01

Setting range: 0～0.5 (mm)

0134	Acceleration of the fore linear acceleration /deceleration interpolated in forecasting control	250

Setting range: 0～2000 (mm/s^2)

0135	Forecasting control, S type pre-acceleration /deceleration time constant	100

Setting range: 0～400（ms）

0136	Linear time constant of the post acceleration /deceleration in forecasting control	80

Setting range: 0～400（ms）

0137	Exponential time constant of the post acceleration /deceleration in forecasting control	60

Setting range: 0～400（ms）

0138	Exponential acceleration/deceleration FL speed of cutting feed in forecasting control	10

Setting range: 0～400（ms）

0139	Contour control precision in forecasting control	0.01

Setting range: 0～0.5 (mm)

0140	Blocks merged in forecasting control	0

Setting range: 0～10

0141	In-position precision in forecasting control	0.05

Setting range: 0～0.5 (mm)

0142	Length condition of circular formation in forecasting control	5

Setting range: 0～30

0143	Angular condition of circular formation in forecasting control	10

Setting range: 0～30

0144	Critical angle of the two blocks during automatic corner deceleration in forecasting control	5

Setting range: 2～178 (mm/min)

0145	Minimum feedrate of automatic corner deceleration in forecasting control	120

Setting range: 10～1000 (mm/min)

0146	Axis error allowable for speed difference deceleration in forecasting control	80

Setting range: 60～1000

0147	Cutting precision grade in forecasting control	2

Setting range: 0～8

0148	External acceleration limit of circular interpolation	1000

Setting range: 100～5000 （mm/s^2）

0149	Lower limit of the external acceleration clamp for circular interpolation	200

Setting range: 0～2000（mm/min）

0150	Acceleration clamp time constant of cutting feed	50

Setting range: 0～1000（ms）

0151	Maximum clamp speed of handwheel incomplete running	2000

Setting range: 0～3000（mm/min）

0152	Linear acceleration /deceleration time constant of handwheel	120

Setting range: 0～400（ms）

0153	Exponential acceleration /deceleration time constant of handwheel	80

Setting range: 0～400（ms）

0154	Acceleration clamp time constant of handwheel	100

Setting range: 0～400（ms）

0155	Maximum clamp speed of step feed	1000

Setting range: 0～3000 (mm/min)

0156	Linear acceleration /deceleration time constant of axes JOG feed	100

Setting range: 0～400（ms）

0157	Exponential acceleration /deceleration time constant of axes JOG feed	120

Setting range: 0～400（ms）

0160	Multiplication coefficient of X axis instruction(CMR)	1

Setting range: 1～256

0161	Multiplication coefficient of Y axis instruction (CMR)	1

Setting range: 1～256

0162	Multiplication coefficient of Z axis instruction (CMR)	1

Setting range: 1～256

0163	Multiplication coefficient of 4TH axis instruction (CMR)	1

Setting range: 1～256

0164	Multiplication coefficient of 5TH axis instruction (CMR)	1

Setting range: 1～256

0165	Frequency dividing coefficient of X axis instruction(CMD)	1

Setting range: 1～256

0166	Frequency dividing coefficient of Y axis instruction(CMD)	1

Setting range: 1～256

0167	Frequency dividing coefficient of Z axis instruction(CMD)	1

Setting range: 1～256

0168	Frequency dividing coefficient of 4TH axis instruction(CMD)	1

Setting range: 1～256

0169	Frequency dividing coefficient of 5TH axis instruction(CMD)	1

Setting range: 1～256

0170	Servo loop gain of X axis	0.0000

Setting range: 0～9999.9999

0171	Servo loop gain of Y axis	0.0000

Setting range: 0～9999.9999

0172	Servo loop gain of Z axis	0.0000

Setting range: 0～9999.9999

0173	Servo loop gain of 4TH axis	0.0000

Setting range: 0～9999.9999

0174	Servo loop gain of 5TH axis	0.0000

Setting range: 0～9999.9999

0175	In-position width of X axis servo	0.0000

Setting range: 0～9999.9999 (mm)

0176	In-position width of Y axis servo	0.0000

Setting range: 0～9999.9999 (mm)

0177	In-position width of Z axis servo	0.0000

Setting range: 0～9999.9999 (mm)

0178	In-position width of 4TH axis servo	0.0000

Setting range: 0～9999.9999 (mm)

0179	In-position width of 5TH axis servo	0.0000

Setting range: 0～9999.9999 (mm)

0180	Cutting feed in-position width setting of axes	0.0000

Setting range: 0～9999.9999 (mm)

0181	Maximum position error allowable for axes moving	0.0000

Setting range: 0～9999.9999 (mm)

0182	Maximum position error allowable for axes stopping	0.0000

Setting range: 0～9999.9999 (mm)

0183	Position error limit when axis servo is off	0.0000

Setting range: 0～9999.9999 (mm)

0184	Servo error allowable for reference point return	0.0000

Setting range: 0～9999.9999 (mm)

0185	Axes grid/reference point offset amount	0.0000

Setting range: 0～9999.9999 (mm)

0186	Alarm time for abnormal load detection	500

Setting range: 0～9999

0186	Alarm time for abnormal load detection	500

Setting range: 0～9999

0189	Reverse precision by backlash compensation	0.0100

Setting range: 0.0001～1.0000 (mm)

Set $\alpha = p(189) \times 0.0001$, in reverse feeding, if the feeding of single servo period is over α, the backlash compensation begins.

Therefore, in machining outer circle contour with a larger radius, in order to make the offset position not to exceed the quardrant, it needs to set a smaller precision. While in machining a curve surface, in order to not to perform backlash compensation in a fixed point of the tool path to form a swollen ridge, it needs to set a larger precision to make the clearance compensation to be distributed in a certain width.

0190	Backlash compensation amount of X axis	0.0000

Setting range: 0～99.9999 (mm)

0191	Backlash compensation amount of Y axis	0.0000

Setting range: 0～99.9999 (mm)

0192	Backlash compensation amount of Z axis	0.0000

Setting range: 0～99.9999 (mm)

0193	Backlash compensation amount of 4TH axis	0.0000

Setting range: 0～99.9999 (mm)

0194	Backlash compensation amount of 5TH axis	0.0000

Setting range: 0～99.9999 (mm)

0195	Compensation step of X axis clearance by fixed frequency	0.0030

Setting range: 0～99.9999 (mm)

0196	Compensation step of Y axis clearance by fixed frequency	0.0030

Setting range: 0～99.9999 (mm)

0197	Compensation step of Z axis clearance by fixed frequency	0.0030

Setting range: 0～99.9999 (mm)

0198	Compensation step of 4TH axis clearance by fixed frequency	0.0030

Setting range: 0～99.9999 (mm)

0199	Compensation step of 5TH axis clearance by fixed frequency	0.0030

Setting range: 0～99.9999 (mm)

0200	Time constant of backlash compensation by ascending and descending	20

Setting range: 0～400（ms）

0201	Delay time of strobe signals MF, SF, TF	0

Setting range: 0～9999 (ms)

0202	Width acceptable for M, S, T completion signal	0

Setting range: 0～9999 (ms)

0203	Output time of reset signal	200

Setting range: 50～400 (ms)

0204	Bits allowable for M codes	2

Setting range: 1～2

0205	Bits allowable for S codes	5

Setting range: 1～6

0206	Bits allowable for T codes	4

Setting range: 1～4

0210	Incremental amount for automatic sequence number insertion	10

Setting range: 0～1000

0211	Tool offset heading number input disabled by MDI	0

Setting range: 0～9999

0212	Tool offset numbers input by MDI disabled	0

Setting range: 0～9999

0214	Error limit of arc radius	0.05

Setting range: -0.1000～0.1000 (mm)

0216	Pitch error compensation number of X axis reference point	0

Setting range: 0～9999

0217	Pitch error compensation number of Y axis reference point	0

Setting range: 0～9999

0218	Pitch error compensation number of Z axis reference point	0

Setting range: 0～9999

0219	Pitch error compensation number of 4TH axis reference point	0

Setting range: 0～9999

0220	Pitch error compensation number of 5TH axis	0

	reference point	

Setting range: 0～9999

0221	Pitch error compensation points of X axis	256

Setting range: 0～1000

0222	Pitch error compensation points of Y axis	256

Setting range: 0～1000

0223	Pitch error compensation points of Z axis	256

Setting range: 0～1000

0224	Pitch error compensation points of 4TH axis	256

Setting range: 0～1000

0225	Pitch error compensation points of 5TH axis	256

Setting range: 0～1000

0226	Pitch error compensation interval of X axis	5

Setting range: 0～99.9999 (mm)

0227	Pitch error compensation interval of Y axis	5

Setting range: 0～99.9999 (mm)

0228	Pitch error compensation interval of Z axis	5

Setting range: 0～99.9999 (mm)

0229	Pitch error compensation interval of 4TH axis	5

Setting range: 0～99.9999 (mm)

0230	Pitch error compensation interval of 5TH axis	5

Setting range: 0～99.9999 (mm)

0231	Pitch error compensation override of X axis	0.001

Setting range: 0～99.9999

0232	Pitch error compensation override of Y axis	0.001

Setting range: 0～99.9999

0233	Pitch error compensation override of Z axis	0.001

Setting range: 0～99.9999

0234	Pitch error compensation override of 4TH axis	0.001

Setting range: 0～99.9999

0235	Pitch error compensation override of 5TH axis	0.001

Setting range: 0～99.9999

0240	Gain adjustment data for spindle analog output	0

Setting range: 0～9999

0241	Compensation value of offset voltage for spindle analog output	0

Setting range: 0～9999

0242	Spindle speed at spindle orientation, or motor speed at spindle gear shift	50

Setting range: 0～9999(r/min)

0245	Time of spindle speed in-position signal detection	200

Setting range: 0～1000(ms)

0246	Spindle maximum speed to gear 1	5000

Setting range: 0～99999 (r/min)

0247	Spindle maximum speed to gear 2	5000

Setting range: 0～99999 (r/min)

0248	Spindle maximum speed to gear 3	5000

Setting range: 0～99999 (r/min)

0250	Spindle motor speed of gear 1—gear 2 shift	50

Setting range: 0～1000 (r/min)

0252	Spindle motor speed of gear 1 — gear 2 shift in tapping cycle	50

Setting range: 0～1000 (r/min)

0254	Axis as counting for surface speed control	0

Setting range: 0～5

0255	Spindle minimum speed for constant surface speed control (G96)	100

Setting range: 0～9999 (r/min)

0257	Spindle upper limit speed in tapping cycle	2000

Setting range: 0～5000 (r/min)

0258	Spindle upper limit speed	5000

Setting range: 0～5000 (r/min)

0259	Spindle servo loop gain	0

Setting range: 0～9999.9999

0260	Spindle speed baudrate with no alarm for spindle speed monitoring	0

Setting range: 0～9999

0261	Spindle encoder lines	1024

Setting range: 0～100000

0262	Spindle override lower limit	0.0000

Setting range: 0～99.9999

0266	Limit with vector ignored when moving along outside corner in tool radius compensation C	0

Setting range: 0～9999.9999

0267	Maximum value of tool wear compensation	400.0000

Setting range: 0～999.9999 (mm)

0270	Retraction amount of high-speed peck drilling cycle G73	2.0000

Setting range: 0～999.9999 (mm)

0271	Reserved space amount of canned cycle G83	2.0000

Setting range: 0～999.9999 (mm)

0272	Spindle speed change ratio in tool retraction without overload torque signal	0

Setting range: 0～9999.0000

0273	Spindle speed change ratio in tool retraction with overload torque signal received	0

Setting range: 0～9999.0000

0274	Cutting feedrate change ratio in tool retraction without overload torque signal	0

Setting range: 0～9999.0000

0275	Cutting feedrate change ratio in small peck drilling cycle	0

Setting range: 0～9999.0000

0276	Macro variable number of retraction actions during output cutting	0

Setting range: 0～9999.0000

0277	Macro variable number output of retraction actions due to overload signal	0

Setting range: 0～9999.0000

0278	Traverse speed back to point R with address I not specified	0

Setting range: 0～9999.0000

0279	Traverse speed to the hole bottom with address I not specified	0

Setting range: 0～9999.0000

0280	Clearance of small peck drilling cycle	0

Setting range: 0～9999.0000

0281	Minimum dwell time at the hole bottom	0

Setting range: 0～1000 (ms）

0282	Maximum dwell time at the hole bottom	9999

Setting range: 1000～9999 (ms）

0283	Override for retraction in rigid tapping	1.0000

Setting range: 0.8000～1.2000

0284	Retraction or spacing amount in peck tapping cycle	0

Setting range: 0～100 (mm)

0285	Synch error range setting for rigid tapping	0

Setting range: 0～100 (mm)

0286	Tooth number of spindle side gear(1st gear)	1

Setting range: 1～999

0287	Tooth number of spindle side gear(2nd gear)	1

Setting range: 1～999

0288	Tooth number of spindle side gear(3rd gear)	1

Setting range: 1～999

0290	Tooth number of position encoder side gear(1st gear)	1

Setting range: 1～999

0291	Tooth number of position encoder side gear(2nd gear)	1

Setting range: 1～999

0292	Tooth number of position encoder side gear (3rd gear)	1

Setting range: 1～999

0294	Maximum spindle speed in rigid tapping(1st gear)	500

Setting range: 0～9999 (r/min)

0295	Maximum spindle speed in rigid tapping(2nd gear)	1000

Setting range: 0～9999 (r/min)

0296	Maximum spindle speed in rigid tapping(3rd gear)	2000

Setting range: 0～9999 (r/min)

0298	Linear acceleration/deceleration time constants of spindle and tapping axis(1st gear)	40

Setting range: 0～400 （ms）

0299	Linear acceleration/deceleration time constants of spindle and tapping axis(2nd gear)	40

Setting range: 0～400 （ms）

0300	Linear acceleration/deceleration time constants of spindle and tapping axis(3rd gear)	40

Setting range: 0～400 （ms）

0302	Time constants of spindle and tapping axis in retraction (1st gear)	20

Setting range: 0～9999 （ms）

0303	Time constants of spindle and tapping axis in retraction (2nd gear)	20

Setting range: 0～9999 （ms）

0304	Time constants of spindle and tapping axis in retraction (3rd gear)	20

Setting range: 0～9999（ms）

0306	Position control loop gain of spindle and tapping axis in rigid tapping (1st gear)	0

Setting range: 0～9999

0307	Position control loop gain of spindle and tapping axis in rigid tapping (2nd gear)	0

Setting range: 0～9999

0308	Position control loop gain of spindle and tapping axis in rigid tapping(3rd gear)	0

Setting range: 0～9999

0310	Spindle loop gain coefficient in rigid tapping (1st gear)	0

Setting range: 0～9999.9999

0311	Spindle loop gain coefficient in rigid tapping (2nd gear)	0

Setting range: 0～9999.9999

0312	Spindle loop gain coefficient in rigid tapping (3rd gear)	0

Setting range: 0～9999.9999

0314	Spindle in-position width in rigid tapping	0

Setting range: 0～100

0315	Tapping axis in-position width in rigid tapping	0

Setting range: 0～100

0316	Position error limit of tapping axis moving in rigid tapping	0

Setting range: 0～100

0317	Position error limit of spindle moving in rigid tapping	0

Setting range: 0～100

0318	Error limit at tapping axis stopping in rigid tapping	0

Setting range: 0～100

0319	Error limit at spindle stopping in rigid tapping	0

Setting range: 0～100

0320	Spindle clearance in rigid tapping (1st gear)	0

Setting range: 0～99.9999

0321	Spindle clearance in rigid tapping (2nd gear)	0

Setting range: 0～99.9999

0322	Spindle clearance in rigid tapping (3rd gear)	0

Setting range: 0～99.9999

0323	Spindle instruction multiplication coefficient （CMR） (1st gear)	1

Setting range: 1～256

0324	Spindle instruction multiplication coefficient （CMR） (2nd gear)	1

Setting range: 1～256

0325	Spindle instruction multiplication coefficient （CMR） (3rd gear)	1

Setting range: 1～256

0326	Spindle instruction frequency dividing coefficient （CMD） (1st gear)	1

Setting range: 1～256

0327	Spindle instruction frequency dividing coefficient （CMD） (2nd gear)	1

Setting range: 1～256

0328	Spindle instruction frequency dividing coefficient （CMD） (3rd gear)	1

Setting range: 1～256

0329	Rotational angle with no rotational angle specified in coordinate rotation	0

Setting range: 0～9999.9999

0330	Scaling with no scaling specified	1

Setting range: 0～9999.9999

0331	Scaling of X axis	1

Setting range: 0～9999.9999

0332	Scaling of Y axis	1

Setting range: 0～9999.9999

0333	Scaling of Z axis	1

Setting range: 0～9999.9999

0334	Dwell time of unidirectional positioning	0

Setting range: 0～10（s）

0335	Direction and overtravel amount of X axis unidirectional positioning	0

Setting range: -99.9999～99.9999

0336	Direction and overtravel amount of Y axis unidirectional positioning	0

Setting range: -99.9999～99.9999

0337	Direction and overtravel amount of Z axis unidirectional positioning	0

Setting range: -99.9999～99.9999

0338	Direction and overtravel amount of 4TH axis unidirectional positioning	0

Setting range: -99.9999～99.9999

0339	Direction and overtravel amount of 5TH axis unidirectional positioning	0

Setting range: -99.9999～99.9999

0340	Axis number of controlled axis in normal direction	0

Setting range: 0～9999.9999

0341	Rotation speed of controlled axis in normal direction	0

Setting range: 0～9999.9999

0342	Rotation insertion ineffective limit of controlled axis in normal direction	0

Setting range: 0～9999.9999

0343	Moving limit to be executed by the last program normal angle	0

Setting range: 0～9999.9999

0344	Rotation limit of the controlled axis in normal direction inserted by a single block	0

Setting range: 0～9999.9999

0345	Minimum angle of indexing table	0

Setting range: 0～9999.9999

0350	Feedrate by tool length measurement	0

Setting range: 0～1000

0351	r value by tool length measurement	0

Setting range: 0～9999.9999

0352	e value by tool length measurement	0

Setting range: 0～9999.9999

0356	Workpieces machined	0

Setting range: 0～9999

0357	Total workpieces to be machined	0

Setting range: 0～9999

0358	Accumulative time of power-on (hour)	0

Setting range: 0～9999

0360	Accumulative time of cutting (hour)	0

Setting range: 0～9999

0361	Tool life management signal ignored	0

Setting range: 0～9999

0362	Tool life left (using times)	0

Setting range: 0～9999

0363	Tool life left (using time)	0

Setting range: 0～9999

0365	Number of MPG used	0

Setting range: 0～9999

0366	Handwheel sliding amount allowable	0

Setting range: 0～10

0371	Positioning error allowable for reverse X axis	0.0050

Setting range: 0～99.9999 (mm)

0372	Positioning error allowable for reverse Y axis	0.0050

Setting range: 0～99.9999 (mm)

0373	Positioning error allowable for reverse Z axis	0.0050

Setting range: 0～99.9999 (mm)

0374	Positioning error allowable for reverse 4TH axis	0.0050

Setting range: 0～99.9999 (mm)

0375	Positioning error allowable for reverse 5TH axis	0.0050

Setting range: 0～99.9999 (mm)

As the set backlash compensation value (P0190---P0194) of an axis is over the reverse positioning allowable error (P0371---P0375) of this axis, the speed at the end point of a single block lowers to minimum speed before this axis backlash compensation begins, which will make the other axes move a small distance in the backlash compensation period, and that will ensure the resultant path deviating the real path least.

0376	Axes moving sequence to program beginning	12345

Setting range: 0～99999

0380	Referential counter capacity of X axis	0

Setting range: 0～9999

0381	Referential counter capacity of Y axis	0

Setting range: 0～9999

0382	Referential counter capacity of Z axis	0

Setting range: 0～9999

0383	Referential counter capacity of 4TH axis	0

Setting range: 0～9999

0384	Referential counter capacity of 5TH axis	0

Setting range: 0～9999

APPENDIX 2

Alarm List

Alarm No.	Content	Remark
0000	Parameter for cutting off power once is modified	
0001	file open fail	
0002	data input overflow	
0003	program number already in use	
0004	address not found	
0005	no data behind address	
0006	illegal negative sign	
0007	illegal decimal point	
0008	the program file is too large to be loaded completely	
0009	illegal address	
0010	G code wrong	
0011	no feedrate instruction	
0012	disk space is not enough	
0013	the program files are up to the upper limit	
0014	G95 can’t be specified, it is not supported by the spindle	
0015	too many axes	
0016	current pitch compensation beyond range	
0017	no authority to modify	
0018	not allowed to modify	
0019	Scaling function is OFF	
0020	beyond radius tolerance	
0021	illegal plane axis	
0022	arc R, I, J, K are all zero	
0023	R, I, J, K of circular interpolation specified together	
0024	Helical interpolation rotation angle is 0	
0025	G12 and other G code can’t be in a same block	
0027	no axis instruction in G43/G44	
0028	illegal plane selection	
0029	illegal offset value	
0030	illegal compensation number	

0031	illegal P specified in G10	
0032	illegal compensation value in G10	
0033	no intersecting point in offset C	
0034	start-up disabled or offset cancelled in arc instruction	
0035	the compensation instruction changed when establishing tool offset	
0036	G31 can't be instructed	
0037	plane change disabled in offset C	
0038	interference in arc block	
0039	tool nose positioning error in offset C	
0040	To change the worpiece coordinate system in offset C executing	
0041	interference in offset C	
0042	more than ten nonmovable instructions in offset C	
0044	G27~G30 instruction can't be instructed in canned cycle	
0045	Address Q not found or Q is 0 (G73/G83)	
0046	illegal reference point return	
0047	machine zero should be executed before executing the instruction	
0048	Z level lower than R level	
0049	Z level higher than R level	
0050	position unchanged when canned cycle mode is changed	
0051	incorrect move after chamfering	
0052	not G01 code after chamfering	
0053	too many address instructions	
0054	DNC transfer setting wrong	
0055	move value wrong in chamfering or corner rounding	
0058	end point not found	
0059	program number not found	
0060	sequence number not found	
0061	X axis not on the reference point	
0062	Y axis not on the reference point	
0063	Z axis not on the reference point	
0064	4th axis not on the reference point	
0065	5th axis not on the reference point	
0066	canned cycle must be cancelled before executing G10	
0067	the setting format is not supported by G10	
0070	storage or memory full	
0071	data end not found	

0072	too many programs	
0073	program number already in use	
0074	illegal program number	
0075	protection	
0076	address P not defined	
0077	subprogram nesting error	
0078	program number not found	
0082	H code specified in G37	
0083	illegal axis instruction in G37	
0085	communication error	
0087	X axis reference point return unfinished	
0088	Y axis reference point return unfinished	
0089	Z axis reference point return unfinished	
0090	4th axis reference point return unfinished	
0091	5th axis reference point return unfinished	
0092	axis not on the reference point	
0094	P type not allowed(coordinate)	
0095	P type not allowed(EXT OFS CHG)	
0096	P type not allowed(WRK OFS CHG)	
0097	P type not allowed (auto execution)	
0098	G28 found in sequence return	
0099	MDI not allowed after retrieval	
0100	parameter write effective	
0101	Memory data disordered after power off, please ensure correct location	
0110	data overflow	
0111	operated data overflow	
0112	divided by zero	
0113	improper instruction	
0114	macro format error	
0115	illegal variable	
0116	write protected variable	
0118	parenthesis nesting error	
0119	M00~M02, M06, M98, M99 ,and M30 can't be in a same block with other M codes	
0122	quadruplicate macro-mode calling	
0123	macro unallowed in DNC	
0124	Illegal program end	

0125	macro format error	
0126	illegal loop number	
0127	NC and macro in a same block	
0128	sequence number by illegal macro	
0129	illegal argument address	
0130	illegal axis operation	
0131	too many external alarm messages	
0132	alarm number not found	
0133	unsupported axis instruction	
0135	illegal angle instruction	
0136	illegal axis instruction	
0139	PLC axis change disabled	
0141	G51 disabled in CRC	
0142	illegal scaling	
0143	scaling motion data overflow	
0144	illegal plane selection	
0148	illegal data setting	
0149	format error in G10L3	
0150	illegal tool group number	
0151	tool group number not found	
0152	no space for tool data	
0153	T code not found	
0154	not using tool in life group	
0155	illegal T code in M06	
0156	P/L instruction not found	
0157	too many tool groups	
0158	illegal tool life data	
0159	tool data setting unfinished	
0160	arc programming only by R in polar system	
0161	The instruction can’t be executed in polar coordinate mode	
0163	The instruction can’t be executed in revolution mode	
0164	The instruction can’t be executed in scaling mode	
0165	Please specify the instruction in a single block	
0166	No axis specified in reference point return	
0167	intermediate point coordinate too large	
0168	the min. dwell time at the hole botton should be shorter than the max. dwell time	
0170	tool radius compensation not cancelled	

0172	P not integer or less than 0 in a block calling subprogram	
0173	Subprogram called beyond 9999 times	
0175	canned cycle can only be executed in G17 plane	
0176	spindle speed not specified before rigid tapping	
0177	spindle orientation not supported	
0178	spindle speed not specified before canned cycle	
0181	illegal M code	
0182	illegal S code	
0183	illegal T code	
0184	tool selected beyond range	
0185	L too small or undefined	
0186	L too large	
0187	Tool radius too large	
0188	U too large	
0189	U less than zero	
0190	V too small or undefined	
0191	W too small or undefined	
0192	Q too small or undefined	
0193	I undefined or I for zero	
0194	J undefined or J for zero	
0195	D undefined or D for zero	
0198	Illegal axis selection	
0199	macro not defined	
0200	illegal S instruction	
0201	feedrate not found in rigid tapping	
0202	position LSI overflow	
0203	program wrong in rigid tapping	
0204	illegal axis operation	
0205	rigid mode DI signal off	
0206	can’t change plane(rigid tapping)	
0207	tapping data wrong	
0212	illegal plane selection	
0224	reference point return	
0231	illegal format in G10 or L50 or L51	
0232	too many helical interpolation axes specified	
0233	device busy	
0235	end of recording	
0236	program restart parameter error	

0237	no decimal point	
0238	address repetition error	
0239	parameter is 0	
0240	G41/G42 disabled in MDI mode	
0251	emergency stop alarm	
0300	n-axis origin return	
0301	APC alarm: n-axis communication	
0302	APC alarm: n-axis overtime	
0303	APC alarm: n-axis data format	
0304	APC alarm: n-axis parity	
0305	APC alarm: n-axis pulse error	
0306	APC alarm: n-axis battery voltage 0	
0307	APC alarm: n-axis battery voltage low 1	
0308	APC alarm: n-axis battery voltage low 2	
0309	APC alarm: n-axis ZRN impossible	
0350	SPC alarm: n axis pulse encoder	
0351	SPC alarm: n-axis communication	
0400	servo alarm: n-axis overload	
0401	servo alarm: n-axis VRDY off	
0404	servo alarm: n-axis VRDY on	
0405	servo alarm: (zero return error)	
0407	servo alarm: superheterodyning	
0409	torque alarm: superheterodyning	
0410	servo alarm: n-axis superheterodyning	
0411	servo alarm: n-axis superheterodyning	
0413	servo alarm: n-axis LSI overflow	
0414	servo alarm: n-axis detection error	
0415	servo alarm: n-axis move too fast	
0416	servo alarm: n-axis detecting broken off	
0417	servo alarm: n-axis parameter error	
0420	synch torque: superheterodyning	
0421	servo alarm: superheterodyning	
0422	servo alarm: speed error	
0423	servo alarm: cumulative travel superheterodyning	
0448	n-axis: unmatched feedback alarm	
0449	n-axis: INV.IPM alarm	
0451	X axis driver alarm	
0452	Y axis driver alarm	

0453	Z axis driver alarm	
0454	4th axis driver alarm	
0455	5th axis driver alarm	
0456	spindle driver alarm	
0500	software overtravel: -X	
0501	software overtravel: +X	
0502	software overtravel: -Y	
0503	software overtravel: +Y	
0504	software overtravel: -Z	
0505	software overtravel: +Z	
0506	software overtravel: -4th	
0507	software overtravel: +4th	
0508	software overtravel: -5th	
0509	software overtravel: +5th	
0510	hardware overtravel: -X	
0511	hardware overtravel: +X	
0512	hardware overtravel: -Y	
0513	hardware overtravel: +Y	
0514	hardware overtravel: -Z	
0515	hardware overtravel: +Z	
0516	hardware overtravel: -4th	
0517	hardware overtravel: +4th	
0518	hardware overtravel: -5th	
0519	hardware overtravel: +5th	
0740	rigid tapping alarm: superheterodyning	
0741	rigid tapping alarm: superheterodyning	
0742	rigid tapping alarm: LSI overflow	
0751	lst spindle alarm (AL-XX) detected	
0754	spindle abnormal torque alarm	
1001	relay or coil address not set	
1002	functional instruction of code input not exist	
1003	incorrect COM / COME instruction use	
1004	User ladder beyond the maximum permissible linage or step number	
1005	Incorrect END1,END2 functional instruction use	
1006	Illegal output in NET	
1007	PLC communication fail due to hardware failure or sysem interruption	
1008	functional instruction wrongly linked	

1009	network horizontal lines not linked	
1010	editing NET loss due to power-off in ladder editing	
1011	address data wrongly input	
1012	sign input undefined or address input beyond range	
1013	illegal character defined	
1014	CTR address repeated	
1015	functional instruction JMP(LBL10) wrongly processed or beyond the capacity	
1016	incomplete NET constitution	
1017	unsupported NET constitution exists	
1018	suspended node exists in NET	
1019	TMR address repeated	
1020	no parameter in functional instruction	
1021	PLC stopped automatically by system when executed overtime	
1022	please input functional code	
1023	programming attempt without ROM and ROM	
1024	unnecessary relay or coil exists	
1025	Functional instruction output wrongly	
1026	NET link linage beyond the supported range	
1027	an output address used in another place	
1030	false vertical line in network	
1031	Message data area is full. Please reduce COD instruction data list capacity.	
1032	ladder 1st level too large to be executed on time	
1033	SFT instructions beyond the max. allowed number	
1034	functional instruction DIFU/DIFD wrongly used	
1039	Instruction or network beyond executable area	
1040	Incorrect functional instruction CALL / SP / SPE use	