This manual describes the various matters concerning the operations of this CNC system as much as possible. However, it is impossible to give detailed descriptions to all the unnecessary or unallowable operations due to space limitation and product specific applications. Therefore, the matters not specially described herein should be considered as "impossible" or "unallowable".

Preface

Dear users,

It is our pleasure for your patronage and purchase of this machining center CNC system of GSK218MC series produced by GSK CNC Equipment Co., Ltd.

This book is "Programming and Operation Manual", which introduces the programming and operation of the machining center CNC system of GSK218MC series in detail.

To ensure the product works in a safe and efficient state, please read this manual carefully before installation and operation.

Warnings

Improper operations may cause unexpected accidents. Only those qualified staff are allowed to operate this system.

Special notes: The power supply fixed on/in the cabinet is exclusively used for the CNC system made by GSK.
It cannot be applied for other purposes, or else it may cause serious danger.

Declaration!

- We try to describe all the various matters as much as possible in this manual. However, it is impossible to give detailed descriptions to all the unnecessary or unallowable operations because there are too many possibilities. Therefore, the matters not specially described herein should be considered as "impossible" or "unallowable".

Warning!

- Before installing, connecting, programming and operating the product, please read this manual and the manual provided by the machine tool builder carefully, and operate the product according to these manuals. Otherwise, the operation may cause damage to the product and machine tool, or even cause personal injury.

Caution!

- The functions and specifications (e.g., precision and speed) described in this manual are only for this product itself. For those CNC machine tools installing this product, the actual function configuration and specifications depend on the designs of the machine tool builders. Moreover, the function configuration and specifications of the CNC machine tool are subject to the manual provided by the machine tool builder.

All specifications and designs in this manual are subject to change without notice.

Safety notes

■ **Transportation and storage**

- Do not pile up the packing boxes over 6 layers.
- Never climb the packing box, neither stand on it, nor place heavy objects on it.
- Do not move or drag the product by the cables connected to it.
- Avoid impact or scratch to the panel and screen.
- Packing box should be protected from dampness, insolation and drench.

■ Open-package inspection

- Confirm the product is the one you purchased after opening the package.
- Check whether the product is damaged during transportation.
- Confirm all the elements are complete without damage by referring to the list.
- If there is incorrect product type, incomplete accessories or damage, please contact us in time.

■ **Connection**

- Only qualified personnel can connect and inspect the system.
- The system must be earthed. The earth resistance should not be greater than 0.1Ω, and a neutral wire (zero wire) cannot be used as an earth wire.
- The connection must be correct and secured. Otherwise, the product may be damaged or unexpected results may occur.
- Connect the surge absorbing diode to the product in the specified direction; otherwise the product may be damaged.
- Turn off the power before inserting or unplugging a plug, or opening the electric cabinet.

■ **Troubleshooting**

- Turn off the power supply before troubleshooting or replacing components.
- Overhaul the system when there is a short circuit or overload, and do not restart it until the trouble is removed.
- Do not turn ON/OFF the product frequently, and the ON/OFF interval should be 1 minute at least.

BOOKⅠ　PROGRAMMING

This part gives an introduction to the specification, product portfolio, parameter configuration, instruction codes as well as program format.

BOOKⅡ　OPERATION

This part gives an introduction to the operation of the machining center CNC system of GSK 218MC series.

APPENDIX

This part gives an introduction to the use of the machining center CNC system and its accessories of GSK218MC series.

Safety responsibility

Manufacturer Responsibility

——Be responsible for the danger which should be eliminated on the design and configuration of the provided CNC systems

——Be responsible for the safety of the provided CNC and its accessories

——Be responsible for the provided information and advice

User Responsibility

——Be trained with the safety operation of CNC system operation procedures and familiar with the safety operation.

——Be responsible for the dangers caused by adding, changing or modifying the original CNC systems and accessories.

——Be responsible for the danger caused by failing to observe the operation, maintenance, installation and storage in the manual.

This user manual shall be kept by the end user.

Thank you for your kind support when you are using the products of Guangzhou CNC Equipment Co., Ltd.

Contents

OVERVIEW

1.1 Overview

This manual consists of the following parts:

BOOK I Programming

This part describes the program configuration and programming basics for the GSK218MC series machining center CNC system as well as the function of each code. Moreover, it introduces the code format, features and limitations when NC language is used to program.

BOOK Ⅱ Operation

This part describes the pages and their settings of the CNC system, the operations and automatic run of the machine tool, the program input/output and program editing as well as the system communication, etc.

Appendix

This part describes the parameter list and alarm list (including parameter default values and parameter setting range) of the GSK218MC series machining center CNC system.

This manual applies to the CNC systems of GSK218MC, GSK218MC-H and GSK218MC-V.

1.2 System introduction

GSK 218MC series product is upgraded from the ones of previous GSK 218M and GSK 218MA. With the adoption of the high-speed spline interpolation algorithm, its control precision and dynamic performance have been improved significantly. The installation structure of the product is divided into three types, inclulding standard integrated type, horizontal type and vertical type. Both the standard integrated type and GSK 218MC-V vertical type CNC systems adopt a 10.4 inch

color LCD, while the GSK 218MC-H horizontal CNC system adopts a 8.4 inch color LCD; moreover, the product is easy to operate by using a friendly and beautiful man-machine interface. Therefore, it is applicable to the CNC application for the machines in automation field, such as milling machines, carving and milling machines, machining centers, grinding machines and gear-hobing machines.

Product features

- Excellent high speed interpolation function, used for complicated curved face machining. Effective machining speed: 8m/min, optimum machining speed: 4m/min.
- Up to 1000 interpolation pre-processing blocks, making the machining precision and workpiece surface smoothness much higher.
- Maximum positioning speed: 30m/min (can be extended to 60m/min), maximum feed speed: 15m/min.
- Display resolution: 800×600, with a more beautiful and delicate interface.
- With RS232 and USB interfaces; data transmission, DNC machining and USB on-line machining function are available.
- Flexible and extendable functions, available to modifiy machines according to customers' production requirements.

1.3 Type signification

GSK 218MC —□

Assembling mode（standard integrated panel is omitted） H：Horizontal assembling V：Vertical assembling

218MC series machining center CNC system

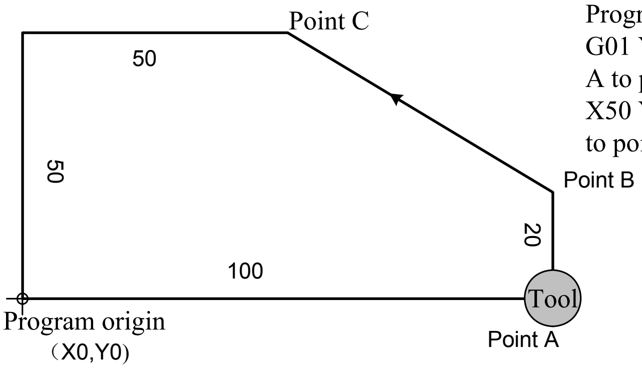
Indicating GSK Equipment CNC CO.,LTD.

BOOK Ⅰ PROGRAMMING

CHAPTER 1　OVERVIEW

1.1　Tool movement along workpiece contour-interpolation

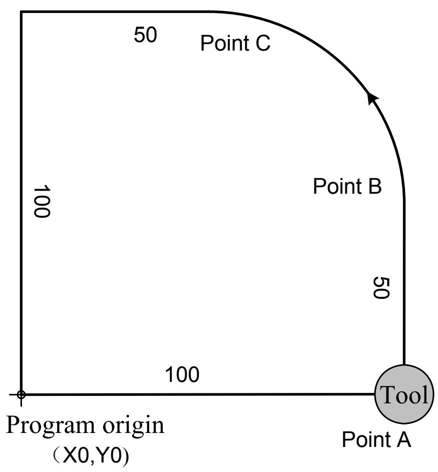
1）　Tool movement along a straight line



Program command
G01 Y20 F100; moves from point A to point B
X50 Y50;　moves from point B to point C

Fig. 1-1-1

2）　Tool movement along an arc



Program command
G01　X100 Y50 F500 ;　moves from point A to point B
G03　X50 Y100 R50;　moves from point B to point C

Fig. 1-1-2

The function of moving a tool along a straight line or an arc is called interpolation.
The programming instructions such as G01, G02 and G03 are called preparatory function, which is used to specify interpolation types for the CNC device.

Book 1　Programming

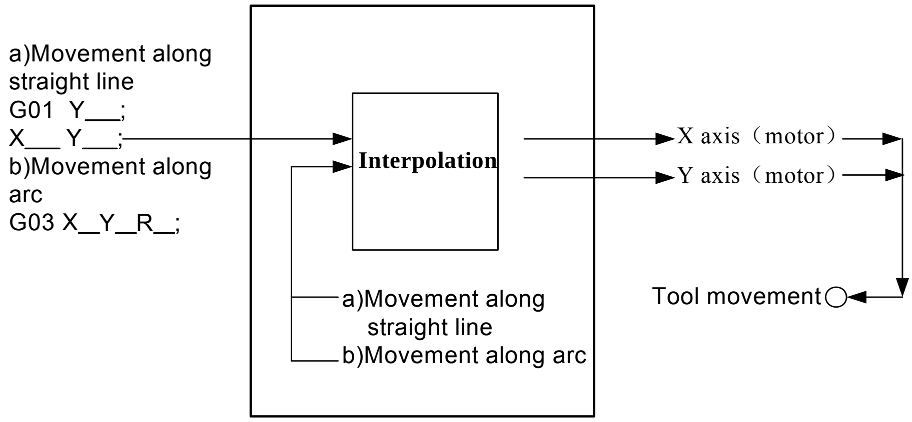


Fig. 1-1-3

Note: For some machines, it is the worktable moves rather than the tool in practice.
It is assumed that the tool moves relative to the workpiece in this manual. Refer to the machine actual movement direction for the actual movement, and protect against personal injury and machine damage.

1.2 Feed——feed function

The feed function, which controls the tool feed speed, is divided into two types.

1. Rapid traverse

   The rapid traverse is used to specify the rapid speed when G00 is used for positioning.

   The rapid traverse speed of each axis is set by parameters, so it is unnecessary to specify it in the program.

2. Cutting feedrate

   Moving a tool at a specified speed to cut a workpiece is called feed. The feedrate is specified with numerical values. E.g., the program code is F150 when the tool is moved at the speed of 150m/min.

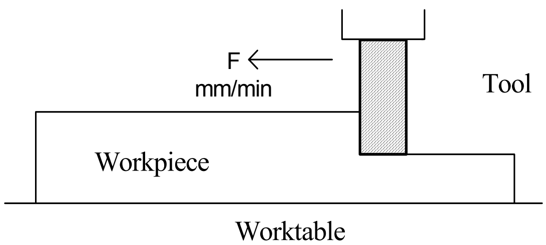


Fig. 1-2-1

1.3 Cutting speed and spindle speed function

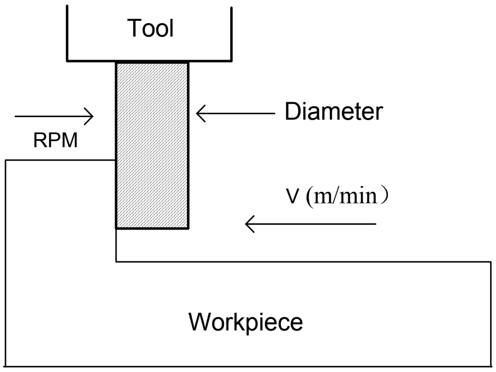


Fig. 1-3-1

The speed of the tool relative to the workpiece when the workpiece is being cut is called the cutting feedrate. CNC can use the spindle speed to specify it——unit (MM/Rev).

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

Instructions related to the spindle speed are called the spindle speed function.

1.4 Instructions for machine tool operations-miscellaneous function

In fact, at the beginning of machining a workpiece, it is necessary to rotate the spindle and supply coolant as required. Therefore, the user must control the ON/OFF operations of spindle motor and cooling pump.

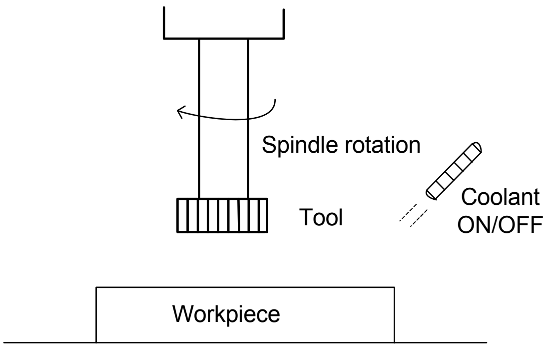


Fig. 1-4-1

The function of controlling programs or the ON/OFF operations of the machine tool using NC codes in CNC is called the miscellaneous function. It is specified by M codes.

E.g., if M03 is specified, the spindle will rotate counterclockwise at the specified speed.

1.5 Tools used in different machining——tool function

Suitable tools must be selected when performing drilling, taping, boring and milling. Each tool is assigned a number. When different numbers are specified in a program, their corresponding tools will be selected.

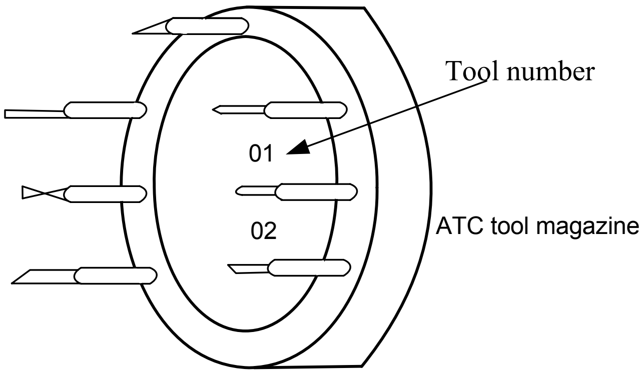


Fig. 1-5-1

E.g., when a tool is placed in ATC number 01, you can select this tool using code T01. This function is called the tool function.

1.6 Tool geometry and tool movement controlled by programs

1.6.1 Tool length compensation

Usually several tools are used for machining one workpiece. If instructions such as G0Z0 are executed in the same coordinate system, because tool lengths of the tools are different, the distances from tool end face to workpiece are different as well. Therefore, it is very inconvenient if the program needs to be changed frequently.

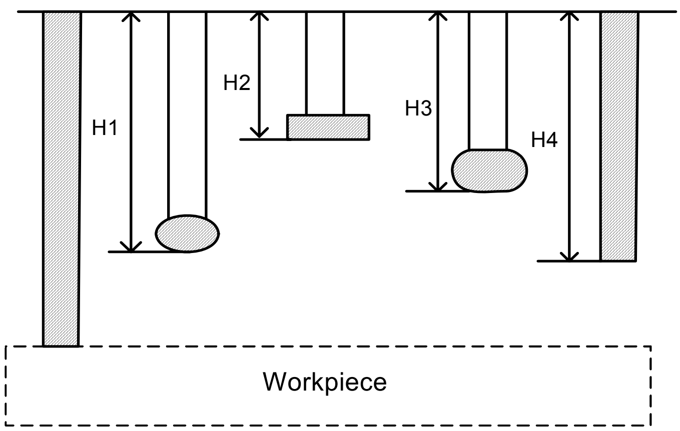


Fig. 1-6-1-1

Therefore, the length of each tool used should be measured in advance, and then set the length difference between the standard tool (usually 1st tool) and other tools in the CNC. When the length compensation program is executed, machining can be performed without altering the program even if the tool is changed, making the distance from tool end face to workpiece the same after the Z axis positioning instruction (e.g., G0Z0) is executed. This is called the length compensation function.

1.6.2 Tool radius compensation

Because a tool has a radius, if the tool performs machining according to the program written in terms with the actual machining contour, a part equal to the radius width will be overcut on the workpiece. To simplify the programming, the program can be run around the workpiece with the tool radius deviated, while the transient path at the intersection between two lines, or a line and an arc can be processed automatically by the system.

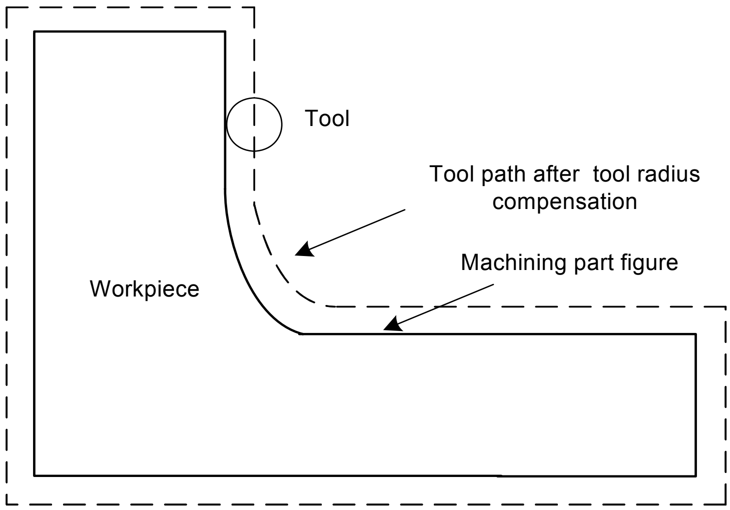


Fig. 1-6-2-1

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

1.7 Tool movement range——stroke

The travel limit switches are fixed at the positive and negative maximum stroke of the machine X, Y and Z axes respectively. If overtravel occurs, the moving axis decelerates and stops after it hits the limit switch, with the overtravel alarm issued at the same time. This function is usually called hardware limit.

The safe operation range for the tool can also be set by parameters. If the tool exceeds the range, the system stops all the moving axes with the overtravel alarm given. This function is called stroke check, namely, the software limit.

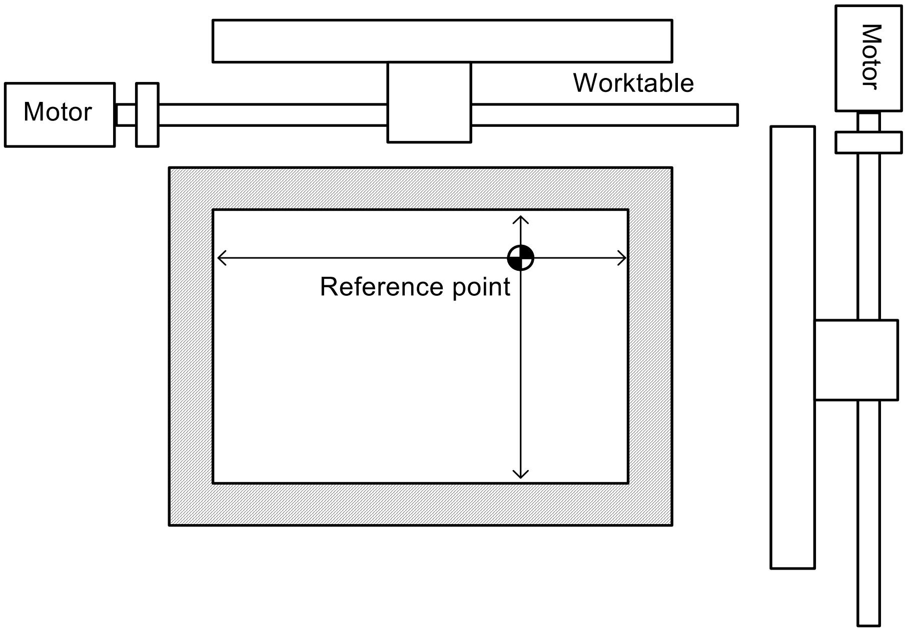


Fig. 1-7-1

CHAPTER 2　PART PROGRAM CONFIGURATION

2.1　Program configuration

A program consists of many blocks, and a block consists of many words. Blocks are isolated by block end codes((LF for ISO、CR for EIA)). Character "；" indicates the block end code in this manual.

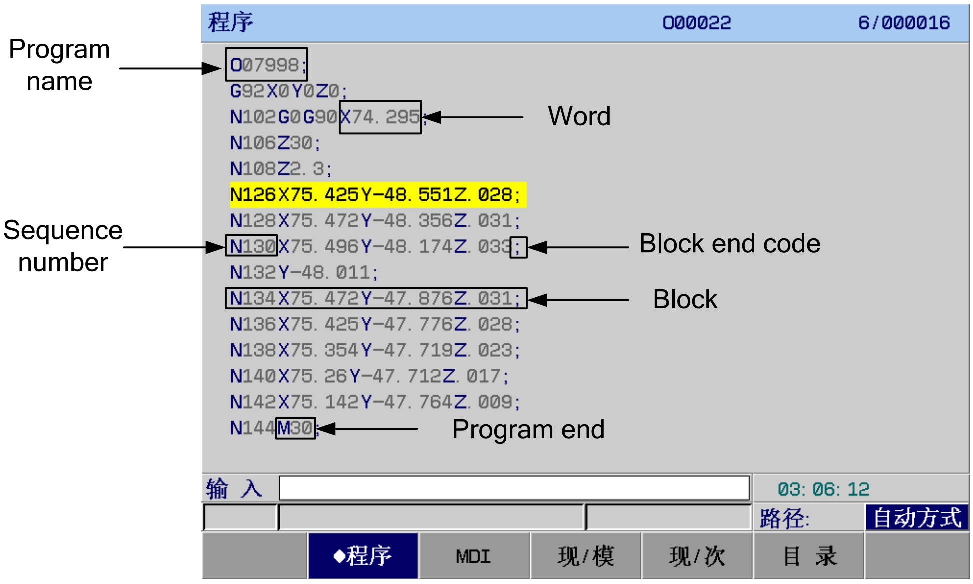


Fig. 2-1-1 Program configuration

A group of instructions for controlling the CNC machine to finish workpiece machining is called the program. After the written program is input to the CNC system, the system will move the tool along a straight line or an arc, or rotate or stop the spindle. Please edit these instructions according to the actual movement sequence of the machine tool in a program. The configuration of a program is shown in Fig. **2-1-1.**

2.1.1　Program name

In this system, the system memory is capable of storing many programs. In order to differentiate these programs, each program begins with an address O followed by a five-digit number, as shown in Fig. 2-1-1-1.

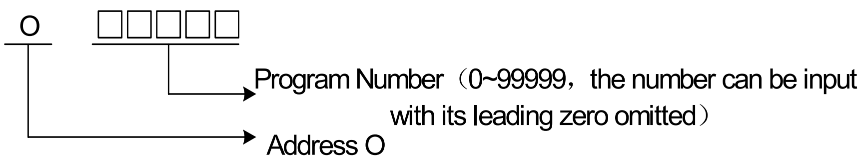


Fig. 2-1-1-1　Configuration of program name

2.1.2 Sequence number and program block

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

Address N with a four-digit sequence number behind it can be used at the beginning of the block (see Fig. 2-1-1), and the leading zero can be omitted. Sequence numbers (whether the sequence number is inserted is set by Parameter NO: 0 # 5, or set the number in the setting page directly. See Section 3.4.1 in Operation) can be specified in a random order, and the intervals between them can be unequal (set by Data Parameter P210). They can be specified in all blocks, or just in some important blocks. However, the numbers should be arranged in ascending order according to general machining sequence. It is for convenience to insert sequence numbers to important parts of the program (e.g. inserting sequence number for tool changing or when the index table moves to a new machining plane).

Note: The N instruction is not processes as a line number when it is in the same block with G10.

2.1.3 Word

A word is a factor that composes a block. It consists of an address and some digits behind it (with sign +or - before the digits sometimes).

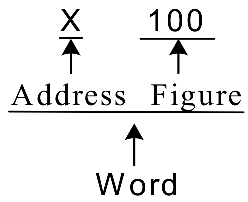


Fig. 2-1-3-1 General structure of a word

An address is one of the English letters from A～Z. It specifies the meaning of the digits behind it. In this system, the addresses and their meanings as well as their ranges are shown in figure **2-1-3-1.**

Sometimes an address may bear different meanings based on different preparatory functions.

When an address is used more than one time in the same instruction, whether an alarm is issued is set by bit parameter N0：32#6

Table 2-1-3-1

Address	Range	Meaning
O	0～99999	Program name
N	0～99999	Sequence number
G	00～99	Preparatory function
X	-99999.9999～99999.9999（mm）	X coordinate address
	0～9999.999（S）	Dwell time
Y	-99999.9999～99999.9999（mm）	Y coordinate address
Z	-99999.9999～99999.9999（mm）	Z coordinate address
R	-99999999.9999～99999999.9999（mm）	Arc radius/angle displacement
	-99999.9999～99999.9999（mm）	R plane in canned cycle
I	-99999999.9999～99999999.9999（mm）	Arc center vector in X axis relative to start point

Address	Range	Meaning
J	-99999999.9999～99999999.9999（mm）	Arc center vector in Y axis relative to start point
K	-99999999.9999～99999999.9999（mm）	Arc center vector in Z axis relative to start point
F	0～99999（mm/min）	Federate per minute
	0.001～500(mm/r)	Federate per revolution
S	Set by parameters	Spindle speed
	00～04	Multi-gear spindle output
T	Set by parameters	Tool function
M	Set by parameters	Miscellaneous function output, program execution process, subprogram call
P	0～99999.9999（ms）	Dwell time
	1～99999	Subprogram number to be called
Q	-99999.9999～99999.9999（mm）	Cutting depth or hole bottom displacement in canned cycle
H	01～99	Operator in G65
	00～256	Length offset number
D	00～256	Radius offset number

Please note that the limits in table **2-1-3-1** are all for the CNC device, but not for the machine tool. Therefore, users are required to refer to the manual provided by the machine tool builder besides this one, in order to get a good understanding of the programming limits before programming.
Note: each word should not exceed 79 characters.

2.2　General structure of a program

The program is divided into main program and subprogram. In general, the CNC system is actuated by the main program. If an instruction for calling the subprogram is executed in the main program, the CNC system acts by the subprogram. When an instruction for returning to the main program is executed in the subprogram, the CNC system will return to the main program and execute the following blocks. The program execution sequence is shown in Fig.2-2-1.

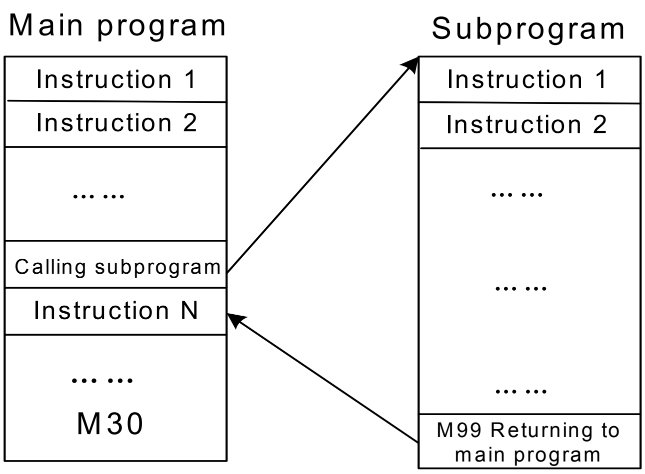


Fig. 2-2-1

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

If a program contains a fixed sequence or frequently repeated pattern, the sequence or pattern can be stored as a subprogram in the memory to simplify the program. The subprogram can be called in Auto mode, usually by M98 in the main program. Besides, the subprogram called can also call another subprogram. The subprogram called from the main program is called the one-level subprogram. Up to 4 levels subprogram can be called in a program (Fig.2-2-2). The last block of a subprogram is the instruction M99 used for returning to the main program. After the return, the blocks following the subprogram calling block are executed. (If the last block of a subprogram is ended with M02 or M03, the system will also return to the main program and proceed to the next block, just as ended with M99.)

When a main program is ended with M99, its execution will be repeated.

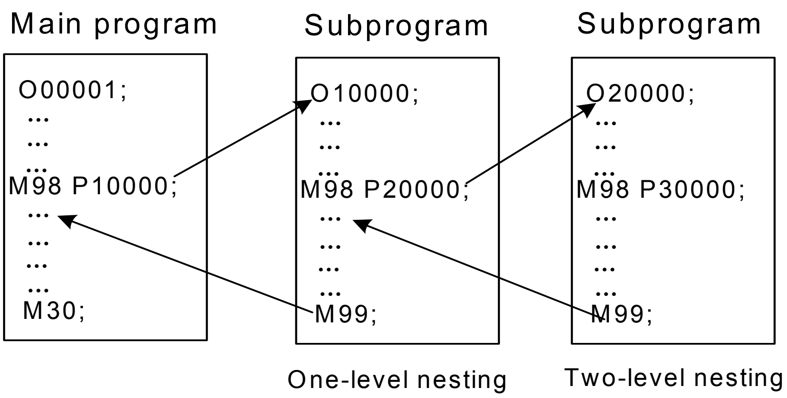


Fig. 2-2-2 Two-level subprogram nesting

The instruction can be called with a subprogram. The same subprogram can be called up to 9999 times consecutively or repeatedly.

2.2.1 Subprogram writing

Write a subprogram following the format below

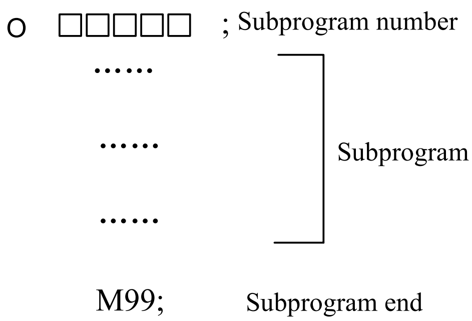


Fig. 2-2-1-1

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

2.2.2 Subprogram call

The subprogram is called by the call instruction of the main program or subprogram. The format of the subprogram is as follows:

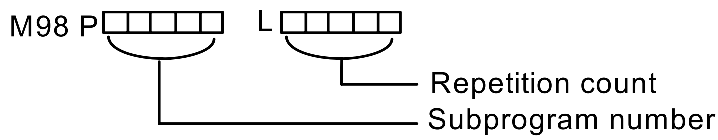


Fig. 2-2-2-1

● If no repetition count is specified, the subprogram is called just once.

(Example) M98 P1002L5 ；(It means a subprogram with number 1002 is repeatedly called 5 times)

● Execution sequence of calling a subprogram from a main program

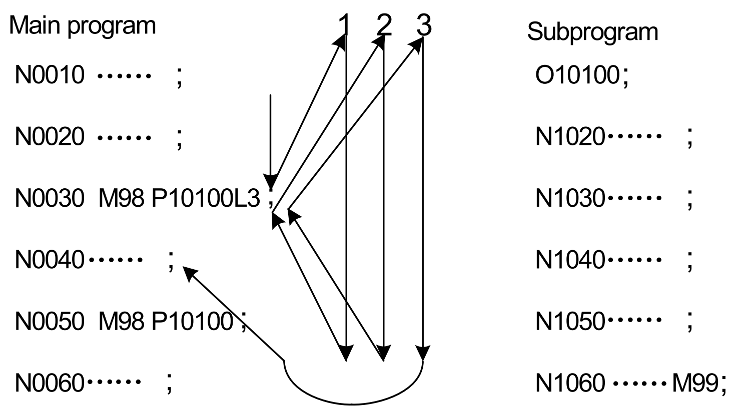


Fig. 2-2-2-2

A subprogram can call another subprogram in the same way as a main program calls a subprogram.

Note 1: An alarm is given when no subprogram number specified with address P is detected.

Note 2: Subprograms with number 90000~99999 are the system reserved programs. When users call such kind of subprograms, the system can execute them but not display them.

2.2.3 Program end

The program begins with a program name, and ends with M02, M30 or M99 (see Fig. **2-2-2-2**). For the end code M02, M30 or M99 detected in program execution: If M02 or M03 is executed in a program, the program is terminated, and the reset state is entered; M30 can be set by bit parameter N0.33#4 to return to the program beginning, and M02 can be set by bit parameter N0.33#2 to return to the program beginning. If M99 is executed in a program, the control returns to the beginning of the program, and then executes the program repeatedly; if M99, M02 or M30 is at the end of the subprogram, the control returns to the program that calls the subprogram and goes on executing the following blocks.

CHAPTER 3 PROGRAMMING BASICS

3.1 Controlled axis

Table 3-1-1

Item	GSK218MC
Basic controlled axes	4 axes （X、Y、Z、4TH）
Extended controlled axes (total)	5 axes at most

3.2 Axis name

The names of the four basic axes are X, Y, Z and A by default.

The number of the controlled axes is set by data parameter **P005,** and the name of each additional axis, such as A, B and C, is set by **P175-P179**.

Note：If two or more axis names are the same, the system initializes them to X, Y, Z, A and B automatically.

3.3 Coordinate system

3.3.1 Machine tool coordinate system

A special point on a machine used as machine benchmark is called machine zero, which is set by the machine tool builder. The coordinate system with machine zero point set as its origin is called the machine coordinate system. It is set up by manual machine zero return after the power is turned 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 in spindle direction is defined as Z axis motion. Viewed from spindle to the workpiece, the motion of the spindle box approaching the workpiece is defined as negative Z axis motion, and the one departing the workpiece as positive. The other directions are determined by right-hand Cartesian coordinate system.

3.3.2 Reference point

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

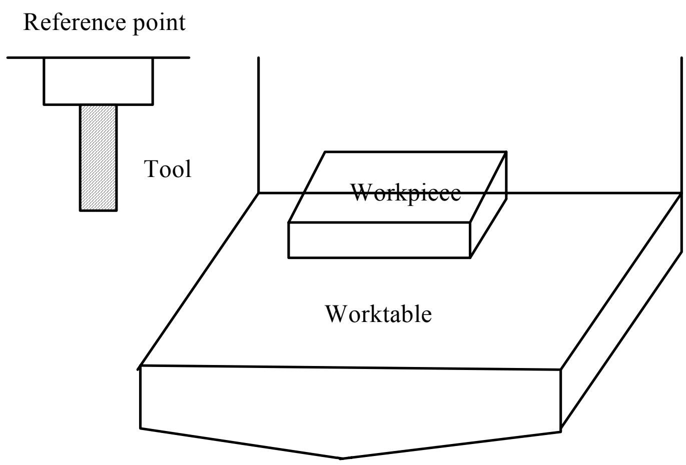


Fig. 3-3-2-1

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

1. Manual reference point return (see “Reference point return” in CHAPTER 9)
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 (set in workpiece coordinate system setting).

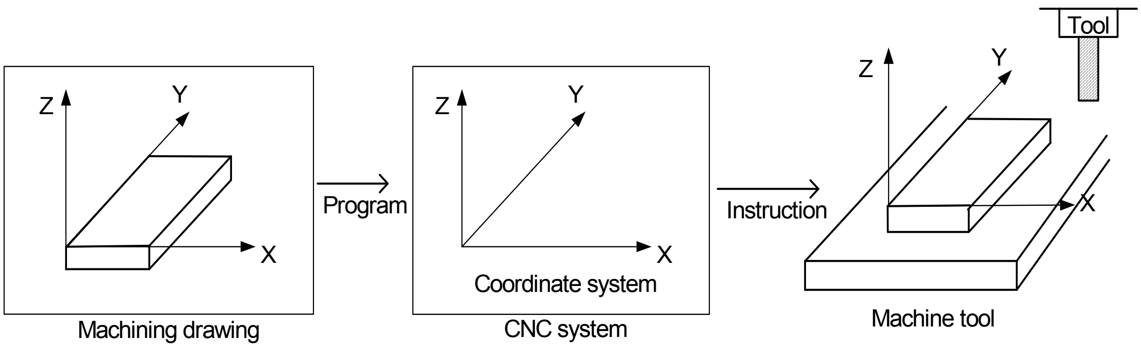


Fig. 3-3-3-1

In the coordinate system specified by CNC, in order to cut the workpiece into the shape on the drawing according to the program of the programming coordinate system on the drawing, the relationship between machine tool coordinate system and workpiece coordinate system must be determined. The method to determine the relationship between these two coordinate systems is called alignment. It can be done by different methods depending on part figure, workpiece quantity, etc.

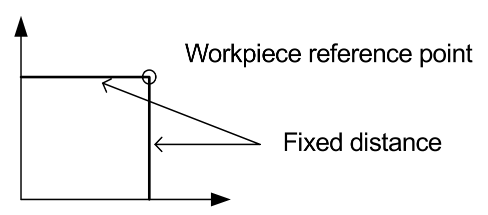
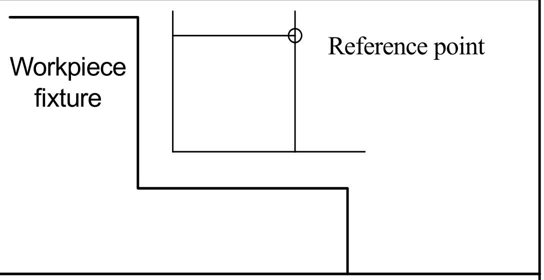
Ⅰ） Using the part reference point	Ⅱ） Fixing the part directly on the fixture
	
Align the tool center to the workpiece reference point, and specify the workpiece coordinate system by CNC instructions at this position. Then the workpiece coordinate system coincides with the programming Coordinate system.	Because the tool center can not be located at the workpiece reference point, the tool is located at a position (can be reference point) the distance of which to the base point is known. Set the workpiece coordinate system using this known distance (e.g. G92).

Fig. 3-3-3-2

A machining program sets a workpiece coordinate system (selecting 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. Using G92, see 4.2.11 for details.
2. Using G54-G59, 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 amount. The absolute definition is a method to program by the coordinate of the end point of the axis movement, which is called absolute programming. Relative definition is method to program directly by the axis moving amount, which is call relative programming (also called incremental programming).

1) Absolute coordinate value

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

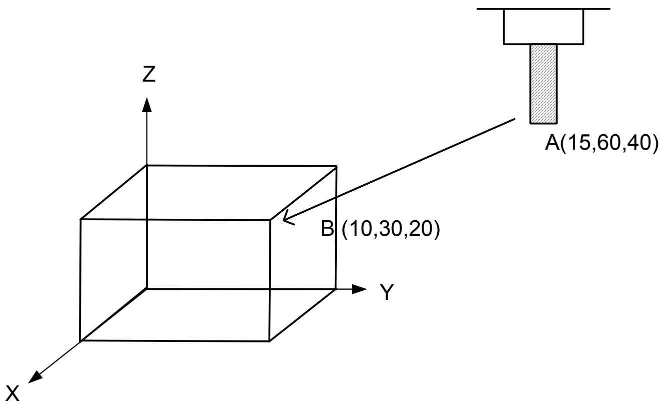


Fig. 3-3-4-1

Move the tool from point A to point B using the point B coordinate in G54 workpiece coordinate system. The instruction is as follows:

G90 G54X10 Y30 Z20 ；

2) Incremental coordinate

It is the target position coordinate relative to the current position with the current position set as the origin.

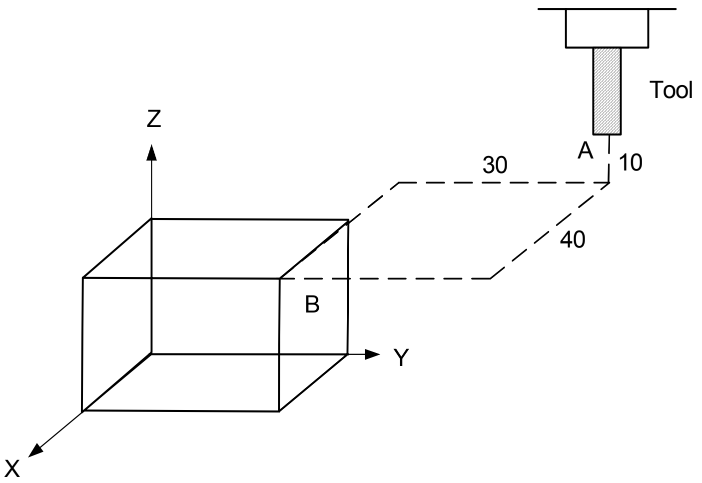


Fig. 3-3-4-2

The tool traverses rapidly to point B from point A. The instruction is as follows:

G0 G91 X-40 Y-30 Z-10；

3.4 Modal and non-modal

The modal means that the set address value keeps effective until it is reset. The other meaning of it is that if a functional word is set, it is unnecessary to input it again in the following blocks which use the same function.

- Example:

G0 X100 Y100； （Rapid positioning to X100 Y100)
X20 Y30；（Rapid positioning to X20 Y30, the modal G0 can be omitted）
G1 X50 Y50 F300；（Linear interpolation to X50 Y50 at a federate of 300mm/min G0→G1）
X100； （Linear interpolation to X100 Y50 at a federate of 300mm/min，the modal G1, Y50 and F300 can be omitted）
G0 X0 Y0； （Rapid positioning to X0 Y0）

The initialized state is the default state after the system Power On. See table 4-1-2.

- Example:

O00001
X100 Y100；（Rapid positioning to X100 Y100，G0 is the initialized state）
G1 X0 Y0 F100；（Linear interpolation to X0 Y0，feed per minute，feedrate: 100mm/min）

Non-modal indicates that the address value is effective only in the block using it. If it is used in the following blocks, it must be specified again. E.g. the functional instructions of group 00 shown in table 4-1-2.

See table 3-4-1 for the modal and non-modal description for the function word.

Table 3-4-1　Modal and non-modal of functional instructions

Modal	Modal G function	A group of G functions that can cancel each other. Once executed, these functions keep effective until they are cancelled by the other G functions in the same group.
	Modal M function	A group of M functions that can cancel each other. These functions keep effective until they are cancelled by the other M functions in the same group.
Non-modal	Non-modal G function	These functions are only effective in the block specifying them. They are cancelled at the end of the block.
	Non-modal M function	These functions are only effective in the block containing them.

3.5　Decimal point programming

Numerical values can be entered with a decimal point. A decimal point can be used when a distance, time, or speed is input. Therefore, it can be specified for the following addresses: **X, Y, Z, A, B, C, I, J, K, R, P, Q and F.**

Explanation:

1. The least moving unit is set by bit parameter **N0：5#1.**
2. The decimal part that is less than the least input incremental unit is rounded off.

Example:

For X9.87654, if the least input incremental unit is 0.001mm, it is processed as X 9.877;
If the least input incremental unit is 0.0001mm, it is processed as X 9.8765.

CHAPTER 4 PREPARATORY FUNCTION: G CODE

4.1 Types of G codes

Preparatory function, represented by a G code with a number behind it, defines the meaning of the block where it is located. G codes are divided into the following two types:

Table 4-1-1

Type	Meaning
Non-modal G code	Only effective in the block in which it is specified
Modal G code	Keep effective until another G code in the same group is specified.

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

G01 X __ ；
Z ______ ； G01 effective
X ______ ； G01 effective
G00 Z____； G00 effective

It is the normal machining mode when the system bit parameter NO：0#7 is set to 0, and the high-speed and high-precision machining mode when NO：0#7 set to 1.

Note 1: F: indicates the normal machining mode; T: indicates high-speed and high-precision machining mode

Note 2: Refer to System Parameter List for details.

Table 4-1-2 G codes and their functions

G code	Group	Format	Whether high-speed and high-precision mode is valid (true or false)	Function
*G00	01	G00 X_Y_Z_	T	Positioning (rapid traverse)
G01		G01 X_Y_Z_F_	T	Linear interpolation (cutting feed)
G02		G02 G03 X_Y_ R_ I_J_ F_;	T	Circular interpolation CW (clockwise)
G03			T	Circular interpolation CCW (counter clockwise)

G code	Group	Format		Whether high-speed and high-precision mode is valid (true or false)	Function
G04	00	G04 P_ or G04 X_		F	Dwell, exact stop
G10		G10 L_N_P_R_		F	Programmable data input
*G11		G11		F	Programmable data input cancel
*G12	16	G12 X_Y_Z_ I_J_K_		F	Stored stroke detection ON
G13		G13		F	Stored stroke detection OFF
*G15	11	G15		F	Polar coordinate instruction cancel
G16		G16		F	Polar coordinate instruction
*G17 G18 G19	02	Written in blocks, used for circular interpolation and tool radius compensation		F	XY plane selection ZX plane selection YZ plane selection
G20	06	Must be specified in a single block		F	Input in inch
*G21					Input in metric
G22	09	G22 X_Y_Z_R_I_L_W_Q_V_D_F_ K		F	CCW inner circular groove rough milling
G23		G23 X_Y_Z_R_I_L_W_Q_V_D_F_K		F	CW inner circular groove rough milling
G24		G24 X_Y_Z_R_I_J_D_F_K_		F	CCW fine milling cycle within a circle
G25		G25 X_Y_Z_R_I_J_D_F_K_		F	CW fine milling cycle within a circle
G26		G26 X_Y_Z_R_I_J_D_F_K_		F	CCW outer circle finishing cycle
G27	00	G27	X_Y_Z_	T	Reference point return detection
G28		G28		T	Reference point return
G29		G29		T	Return from reference point
G30		G30Pn		T	2nd, 3rd and 4th reference point return
G31		G31		F	Skip function
G32	09	G32 X_Y_Z_R_I_J_D_F_K_		F	CW outer circle finishing cycle
G33		G33X_Y_Z_R_I_J_L_W_Q_V_U_D_F_K		F	CCW rectangular groove rough milling
G34		G33X_Y_Z_R_I_J_L_W_Q_V_U_D_F_K		F	CW rectangular groove rough milling
G35		G35 X_Y_Z_R_I_J_L_ U_D_F_K_		F	CCW rectangular groove rough milling cycle

G code	Group	Format			Whether high-speed and high-precision mode is valid (true or false)	Function
G36		G36 X_Y_Z_R_I_J_L_ U_D_F_K_			F	CW rectangular groove rough milling cycle
G37		G37 X_Y_Z_R_I_J_L_ U_D_F_K_			F	CCW rectangular outside groove finishing cycle
G38		G38 X_Y_Z_R_I_J_L_ U_D_F_K_			F	CW rectangular outside groove finishing cycle
G39	00	G39			F	Corner offset circular interpolation
*G40	07	G17	G40 G41 G42	D_X_Y_	T	Tool radius compensation cancel
G41		G18		D_X_Z_	T	Left-hand tool radius compensation
G42		G19		D_Y_Z_	T	Right-hand tool radius compensation
G43	08	G43		H_Z_	T	Tool length compensation in positive direction
G44		G44			T	Tool length compensation in negative direction
*G49		G49			T	Tool length compensation cancel
*G50	12	G50			T	Scaling cancel
G51		G51 X_ Y_ Z_ P_			T	Scaling
G53	00	Written in a program			T	Machine coordinate system selection
*G54	05	Written in a block, usually placed at the program beginning			T	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	00/01	G60 X_ Y_ Z_			T	Unidirectional positioning
G61	14	G61			T	Exact stop mode
G62		G62			T	Automatic corner override
G63		G63			T	Tapping mode
*G64		G64			T	Cutting mode

G code	Group	Format	Whether high-speed and high-precision mode is valid (true or false)	Function
G65	00	G65 H_P# i Q# j R# k	T	Macro program instruction
G68	13	G68 X_ Y_ R_	T	Coordinate rotation
*G69		G69	T	Coordinate rotation cancel
G73	09	G73 X_Y_Z_R_Q_F_;	F	Peck drilling cycle
G74		G74 X_Y_Z_R_P_F_;	F	Left-hand tapping cycle
G76		G76 X_Y_Z_Q_R_P_F_K_;	F	Fine boring cycle
*G80		Written in a block with other programs	F	Canned cycle cancel
G81		G81 X_Y_Z_R_F_;	F	Drilling cycle (spot drilling cycle)
G82		G82 X_Y_Z_R_P_F_;	F	Drilling cycle (counter boring cycle)
G83		G83 X_Y_Z_R_Q_F_;	F	Peck drilling cycle
G84		G84 X_Y_Z_R_P_F_;	F	Right-hand tapping cycle
G85		G85 X_Y_Z_R_F_;	F	Boring cycle
G86		G86 X_Y_Z_R_F_;	F	Boring cycle
G87		G87 X_Y_Z_R_Q_P_F_;	F	Back boring cycle
G88		G88 X_Y_Z_R_P_F_;	F	Boring cycle
G89		G89 X_Y_Z_R_P_F_;	F	Boring cycle
*G90	03	Written into blocks	T	Absolute programming
G91				Incremental programming
G92	00	G92 X_Y_Z_	T	Floating coordinate system setting
*G94	04	G94	T	Feed per minute
G95		G95	T	Feed per revolution
G96	15	G96S_	T	Constant surface speed control （cutting speed）
*G97		G97S_	T	Constant surface speed control cancel（cutting speed）
*G98	10	Written into blocks	T	Return to initial plane in canned cycle
G99				Return to point R plane in canned cycle

Note 1: If modal instructions and non-modal instructions are in the same block, the non-modal instructions take precedence. At the same time, the corresponding modes are changed according to the other modal instructions in the same block, but not executed**.**

Note 2: For the G code with sign *, when the power is switched on, the system is in the state of this G code (some G codes are determined by bit parameter NO：31#0~7).

Note 3: The G codes of group 00 are all non-modal G codes except G10, G11, G92.

Note 4: An alarm occurs if G codes not listed in this table are used or G codes that cannot be selected are specified.

Note 5: G codes from different groups can be specified in a block, but 2 or more G codes from the same group can not be specified in a block by principle. If no alarm occurs when two or more G codes in the same group are in a block after parameter setting, the latter G code functions.

Note 6: If a G code of group 01 is in the same block with a G code of group 09, the G code of group 01 prevails. In canned cycle mode, if G codes from 01 group are specified, the canned cycle will be cancelled automatically and the system turns into G80 state.

Note 7: G codes are represented by group numbers respectively based on their types. Whether the G codes of each group are cleared after reset or emergency stop is determined by bit parameter NO：35#0~7 and NO：36#0~7.

Note 8: If the rotation scaling instruction and the instruction of group 01 or that of group 09 share the same block, the rotation scaling instruction will be taken, and the modes of group 01 or group 09 are changed. If the rotation scaling instruction and the instruction of group 00 share the same block, an alarm occurs.

4.2 Simple G codes

4.2.1 Rapid positioning G00

Code format: G00 X_Y_Z_

Function: G00 instruction moves the tool to the position in the workpiece system specified with the absolute or an incremental instruction at a rapid traverse speed. Whether the absolute or incremental instruction is used is set by bit parameter **NO:12#1**. Select one of the following two tool paths (Fig. 4-2-1-1).

1. Linear interpolation positioning: The tool path is the same as linear interpolation (G01). The tool is positioned within the shortest time at a speed not more than the rapid traverse speed of each axis.
2. Nonlinear interpolation positioning: The tool is positioned at the rapid traverse speed of each axis respectively. The tool path is usually not straight.

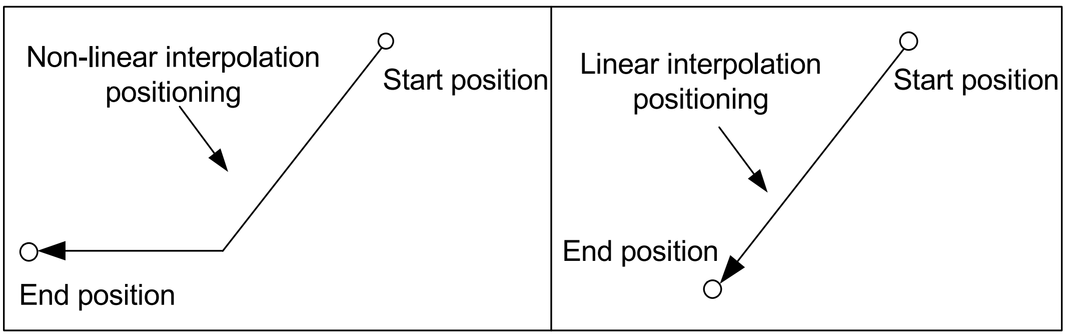


Fig. 4-2-1-1

Explanation:

1. After G00 is executed, the system changes the current tool move mode for G00 mode. Whether the default mode is G00 (parameter value is 0) or G01 (parameter value is 1) after power-on is set by bit parameter **No.031#0.**
2. With no positioning parameter specified, the tool does not move and the system only

changes the mode of the current tool movement for G00.

3. G00 is the same as G0.
4. The G0 speed of axes X, Y, Z and 4th is set by data parameters **P88~P91**.

Limitations:

The rapid traverse speed is set by parameter. The speed F specified in the G0 instruction is the cutting speed of the following machining blocks.

Example:

G0 X0 Y10 F800; Feeding at the speed set by system parameter

G1 X20 Y50; Using the feedrate of F800

The rapid positioning speed is adjusted by the keys F0， 25， 50， 100% on the operation panel (see fig. **4-2-1-2**). The speed to which F0 corresponds is set by data parameter **P93** and it is common to all axes.

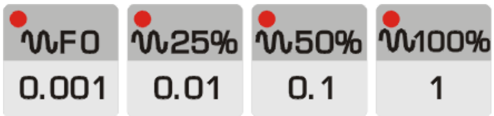


Fig. 4-2-1-2 Keys for rapid feedrate override

Note: Note the position of the worktable and workpiece to prevent tool collision.

4.2.2 Linear interpolation G01

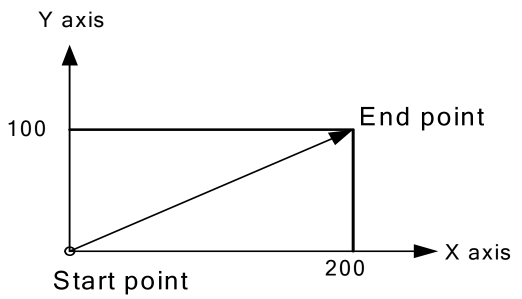
Code format: G01 X_ Y_ Z_ F_

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

Explanation:

1. X_ Y_ Z_ are the coordinates of the end point. Since they are related to the coordinate system, please see sections 3.3.1～3.3.3.
2. The feedrate specified by F keeps effective till a new F value is specified. The federate specified by F code is calculated by an interpolation along a straight line. If F code is not specified in a program, the default F value at system Power On is used (see data parameter P87 for details).

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

<table>
<tr><td rowspan="2"></td><td>G01 X200 Y100 F200 ;</td></tr>
<tr><td>Note: federate of each axis is as follows:
G01 Xα Yβ ZγFf;
In this program:</td></tr>
<tr><td>Feedrate of X axis</td><td>$F_X = \frac{\alpha}{L} \times f$</td></tr>
</table>

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

Federate of Z axis： $F_Z = \frac{\Delta\gamma}{L} \times f$

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

Fig. 4-2-2-1

Note:

1. All code parameters are positioning parameters except for F code. The upper limit of federate F is set by data parameter P96. If the actual cutting federate (after using federate override) exceeds the upper limit, it is clamped to the upper limit (unit: mm/min). The lower limit of the federate F is set by data parameter P97. If the actual cutting federate (after using federate override) exceeds the lower limit, it is clamped to the lower limit (unit: mm/min).

2. The tool does not move when no positioning parameter is specified behind G01, and the system only changes the mode of the current tool movement mode for G01. By altering the system bit parameter NO:31#0, the system default mode at power-on can be set to G00 (value is 0) or G01 (value is 1).

4.2.3 Circular (helical) interpolation G02/G03

A. Circular interpolation G02/G03

Prescriptions for G02 and G03:

The plane circular interpolation means that the arc path is finished according to the specified rotation direction and radius (or circle center) from the start point to end point in the specified plane. Since the arc path can not be determined only by the start point and the end point, other conditions are required:

- Arc rotation direction（G02，G03）
- Circular interpolation plane（G17, G18, G19）
- Circle center coordinate or radius, which thus leads to two instruction formats: Circle center coordinate I, J ,K or radius R programming.

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

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

Arc in XY plane

$$G17 \begin{Bmatrix} G02 \\ G03 \end{Bmatrix} X\_Y\_ \begin{Bmatrix} R\_ \\ I\_J\_ \end{Bmatrix} F\_;$$

Arc in ZX plane

$$G18 \begin{Bmatrix} G02 \\ G03 \end{Bmatrix} X\_Z\_ \begin{Bmatrix} R\_ \\ I\_K\_ \end{Bmatrix} F\_;$$

Arc in YZ plane

$$G19 \begin{array}{c} G02 \\ G03 \end{array} Y\_Z\_ \begin{array}{c} R\_ \\ J\_K\_ \end{array} F\_;$$

Table 4-2-3-1

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

CW and CCW on XY plane (ZX plane or YZ plane) refer to the directions viewed in the positive-to-negative direction of the Z axis (Y axis or X axis) in the right-hand Cartesian coordinate system, as is shown in Fig. 4-2-3-1.

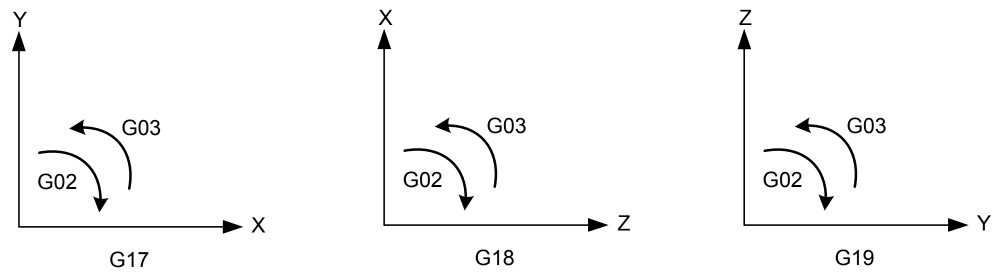


Fig. 4-2-3-1

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

The end point of an arc can be specified by parameter words X, Y and Z. It is expressed as absolute values in G90, and incremental values in G91. The incremental values are the coordinates of the end point relative to the start point. The arc 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 the coordinates of the circle center relative to the arc start point (for simplicity, the circle center coordinates with the start point taken as the origin temporarily). They are the incremental values with signs. See Fig. 4-2-3-2.

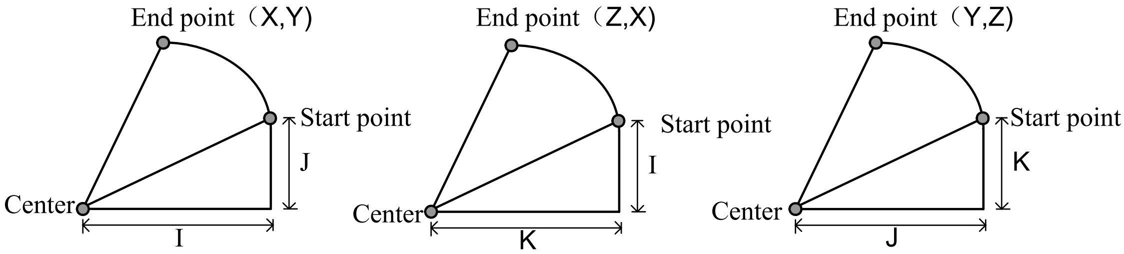


Fig. 4-2-3-2

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

G02 X_ Y_ R_ ；

G03 X_ Y_ R_ ；

1. Two arcs can be drawn as follows; one arc is more than 180°, and the other one is less than 180°. For the arc more than 180°, its radius is specified by a negative value.

(E.g. Fig. **4-2-3-3)** ① As arc is less than 180°,
G91 G02 X60 Y20 R50 F300 ；

② As arc is more than 180°,
G91 G02 X60 Y20 R-50 F300 ；

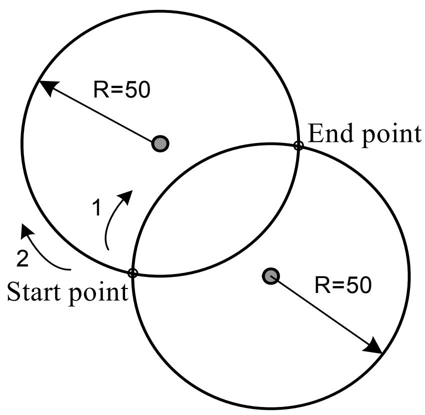


Fig. 4-2-3-3

2. The arc equal to 180° can be programmed either by I, J and 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 of 180°, the arc path is not affected whether the value of R is positive or negative.

3. For the arc equal to 360°, only I, J and K can be used for programming.
(Program example):

Book 1 Programming

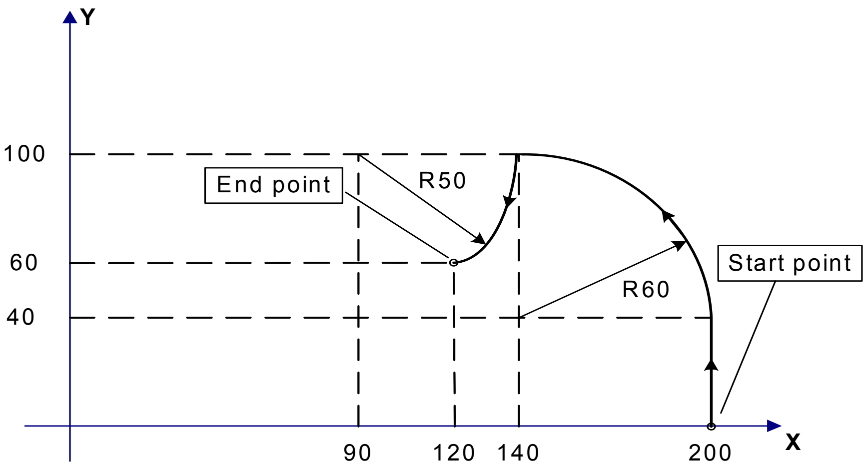


Fig. 4-2-3-4

The tool path programming for Fig. 4-2-3-4 is as follows:

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；

Restrictions:

1. If addresses I, J, K and R are specified simultaneously in a program, the arc specified by R takes precedence, and others are ignored.
2. If neither arc radius parameter or the parameter from the start point to the circle center are specified, an alarm is issued in the system.
3. A full circle can only be interpolated by parameters I, J, K from start point to circle center rather than parameter R.
4. Pay attention to the setting for selecting the coordinate plane when the helical interpolation is being done.
5. If X, Y, Z are all omitted (i.e., the start point and the final point coincides), and R is specified (e.g. G02R50), the tool does not move.

B. Helical interpolation

Code format: G02/G03

Arc on XY plane

$$G17\ \left.\begin{matrix}G02\\ G03\end{matrix}\right\}\ X_p\_Y_p\_Z_p\_\_\quad \left.\begin{matrix}I\_J\_\\ R\_\end{matrix}\right\}\ F\_$$

Arc on ZX plane

$$G18\ \left.\begin{matrix}G02\\ G03\end{matrix}\right\}\ X_p\_Y_p\_Z_p\_\_\quad \left.\begin{matrix}I\_K\_\\ R\_\end{matrix}\right\}\ F\_$$

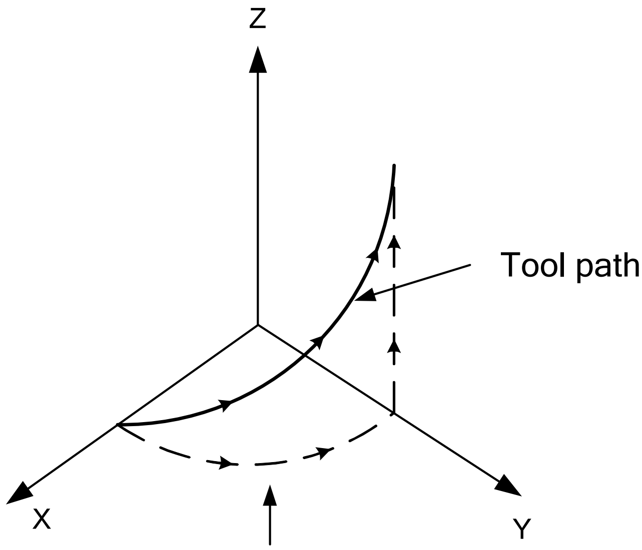
Arc on YZ plane

$$G19\ \left.\begin{matrix}G02\\ G03\end{matrix}\right\}\ X_p\_Y_p\_Z_p\_\_\quad \left.\begin{matrix}J\_K\_\\ R\_\end{matrix}\right\}\ F\_$$

Fig. 4-2-3-5

Function: It is used to move the tool to a specified position from the current position at 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.

Fig. 4-2-3-6

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

If the circle can not be machined according to the specified instruction parameter, the system will give error message. After the execution, the system changes the current tool moving mode for G02/G03 mode.

The federate along the circumference of two circular interpolation axes is specified. The specification method is to simply add a moving axis which is not a circular interpolation axis. The federate along a circular arc is specified by F instruction. Thus the feedrate of the linear axis is as follows:

$$F_C = F * \frac{\text{Length of liner axis}}{\text{Length of circular arc}}$$

Determine the federate to make the linear axis federate not exceed any limit.

Restrictions:

Pay attention to the setting for selecting the coordinate plane when the helical interpolation is being done.

4.2.4 Absolute/incremental programming G90/G91

Instruction format: G90/G91

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

The absolute instruction is a method of programming by the axis moving end point coordinates. The end position involves the concept of coordinate system, please refer to sections 3.3.1～3.3.4.

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

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

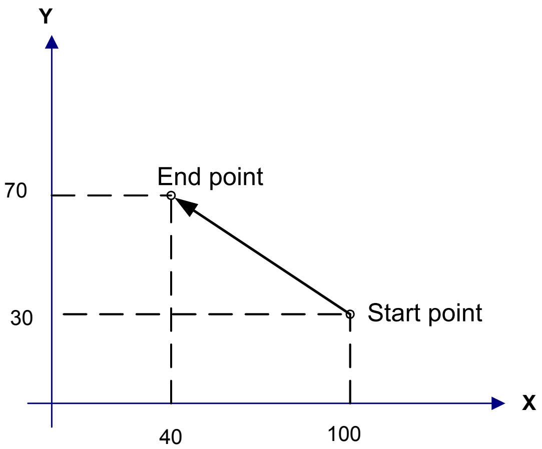


Fig. 4-2-4-1

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

G90 G0 X40 Y70;

Or G91 G0 X－60 Y40 ；

The same action can be performed with the two methods, users thus can choose either one of them as required.

Explanation:

- With no instruction parameter. It can be written into the block with other instructions.
- G90 and G91 are the modal values in the same group, i.e., if G90 is specified, the mode is always G90 (default) till G91 is specified. If G91 specified, the mode is always G91 till G90 specified.

System parameters:

Whether the default positioning parameter is G90 mode **(**parameter is 0**)** or G91 mode**(** parameter is 1**)** at Power On is set by bit parameter **N0：31#4.**

4.2.5 Dwell (G04)

Format: G04 X_ or P_

Function: G40 is for dwell operation. It delays the specified time before executing the next block. In cutting mode G64, it is used for exact stop check. The dwell per revolution in Feed per Revolution mode G95 can be specified by bit parameter No.34#0.

Table 4-2-5-1 Value range of dwell time (instructed with X)

Least moving unit	Value range	Unit of dwell time
No.5#1=0	0.001～9999.999	S or rev
No.5#1=1	0.0001~9999.9999	

Table 4-2-5-2 Value range of dwell time (instructed with P)

Least moving unit	Value range	Unit of dwell time
No.5#1=0	1~99999.999	0.001s or rev
No.5#1=1	1~99999.9999	0.0001s or rev

Explanation:

1. G04 is non-modal instruction, which is only effective in the current block.
2. If parameters X and P appear simultaneously, parameter X is effective.
3. An alarm occurs if the values of X and P are negative.
4. Dwell is not executed if neither X nor P is specified.

4.2.6 Single-direction positioning （G60）

Format: G60 X_ Y_ Z_

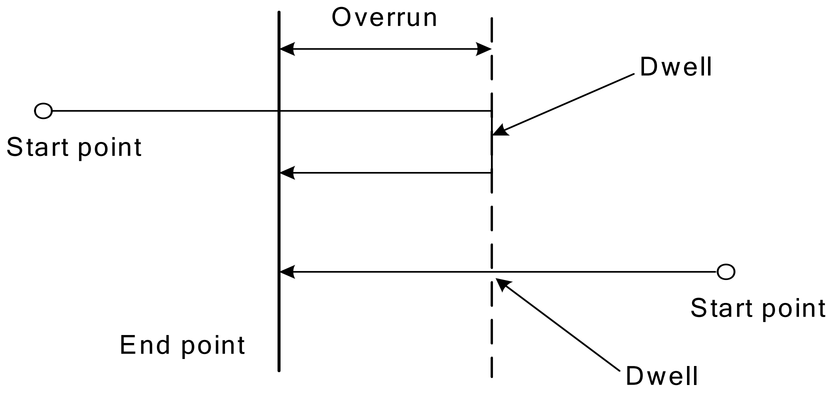


Fig. 4-2-6-1

Function: For accurate positioning without machine backlash, G60 can be used for accurate positioning in a single direction.

Explanation:

G60 is a non-modal G instruction (it can be set to a modal value by bit parameter **NO:48#0**), which is only effective in a specified block.

Parameters X, Y and Z represent the coordinates of the end point in absolute programming; and the moving distance of the tool in incremental programming. In tool offset mode, the path of single-direction positioning is the one after tool compensation when G60 is used.

The overrun marked in above figure can be set by system parameters P335，P336，P337 and P338, and the dwell time can be set by parameter P334. The positioning direction can be determined by setting 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；for statement (1), the tool path is AB→dwell for 1s→BC

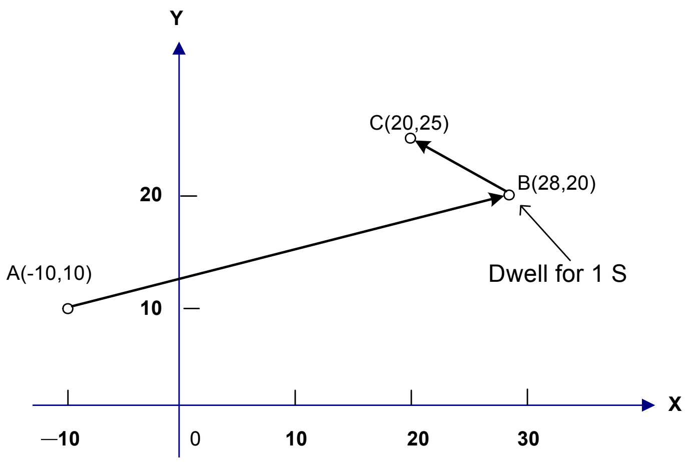


Fig. 4-2-6-2

System parameter:

Table 4-2-6-1

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

Note 1: The signs of parameters P335～P338 are for the direction of single-direction positioning, and their values for the overrun.
Note 2: If overrun>0, the positioning direction is positive.
Note 3: If overrun<0, the positioning direction is negative.
Note 4: If overrun=0, no single-direction positioning is available.

4.2.7 On-line modification for system parameters (G10)

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

Format

G10 L50 N_P _R_； Setting or modifying bit parameter
G10 L51 N_ R_； Setting or modifying data parameter
G11； Canceling parameter input mode

Parameter definition

N： Parameter number. Sequence number to be modified.
P： Parameter bit number. Bit number to be modified.
R： Value. Parameter value after being modified.

The values can also be modified by following instructions. Refer to relative sections for details:

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

Note 1: In parameter input mode, no NC statement can be specified except annotation statement.
Note 2: G10 must be specified in a separate block or an alarm occurs. Please note that the parameter input mode must be cancelled by G11 after G10 is used.

Note 3: The parameter value modified by G10 must within the range of system parameter, otherwise, an alarm occurs.
Note 4: Modal instructions of canned cycle must be cancelled prior to G10 execution, otherwise an alarm occurs.
Note 5: Those parameters which take effect after Power OFF and then On are unavailable to be modified by G10.
Note 6: On line modification for G20 and G21 is unavailable by G10.
Note 7: When G10 modifies external zero offset, workpiece offset, additional workpiece zero offset or tool offset on line in G91 mode, the system adds the instruction offset to the current offset, when modifying them in G90 mode, it modifies by the instruction offset.
Note 8: Cancel G10 mode when executing M00, M01, M02, M30, M99, M98 and M06.
Note 9: Bit parameter No.0#7 (Selection mode: 0 for normal mode, 1 for high speed and high precision mode) does not support G10 on-line modification.

4.2.8 Workpiece coordinate system G54～G59

Format: G54～G59

Function: for specifying the current workpiece coordinate system. The workpiece coordinate system is selected by specifying G instructions of workpiece coordinate system in a program.

Explanation:

1. With no instruction parameter
2. The system itself is capable of setting 6 workpiece coordinate systems, any one of which can be selected by instructions G54~G59.

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

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

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

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

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

   G59 ---------------- Workpiece coordinate system 6
3. At Power On, the system displays the workpiece coordinate instructions G54～G59, G92 or additional workpiece coordinate system ever executed before Power Off.
4. When different workpiece coordinate systems are called in a block, the axis to move is positioned to the coordinate of the new coordinate system; for the axis not to move, its coordinate shifts to the corresponding coordinate in the new coordinate system, with its actual position on the machine tool unchanged.

Example:

The corresponding machine tool 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 in order, the absolute coordinates and machine coordinates of the end point I are displayed as follows:

Table 4-2-8-1

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

5. The offset value of external workpiece zero or the one of workpiece zero can be modified by G10, which is shown as follows:

Using instruction G10 L2 Pp X_Y_Z_

P=0 ： External workpiece zero offset value (reference offset amount)

P=1 to 6 ： Workpiece zero offset values of workpiece coordinate systems 1 to 6

X_Y_Z_ ： For absolute instruction（G90）, it is workpiece zero offset of each axis；For incremental instruction（G91）, it is the offset to be added to the set workpiece zero of each axis (the result of addition is the new workpiece zero offset).

Using G10, each workpiece coordinate can be changed respectively.

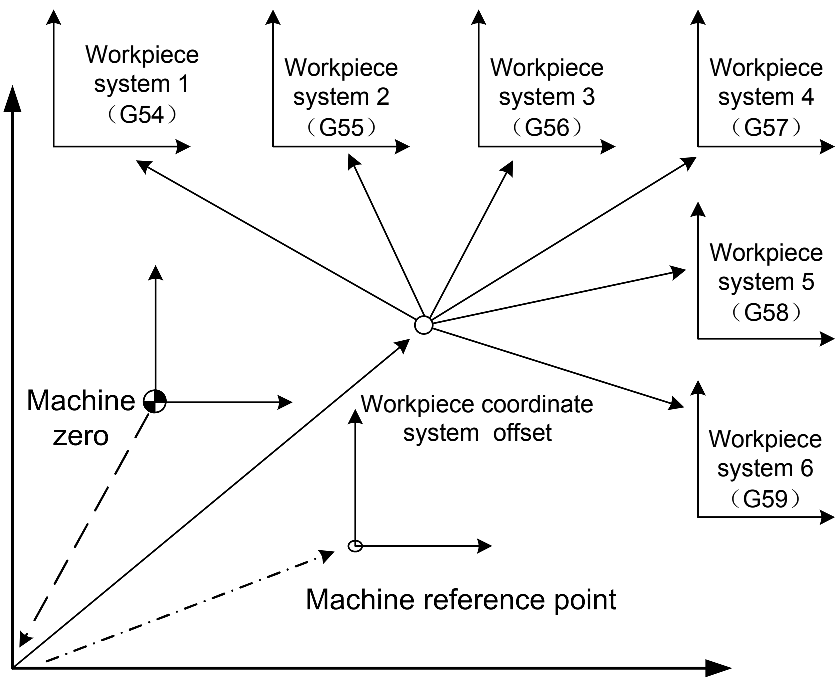


Fig. 4-2-8-1

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 the machine zero, which thus generates the machine reference point and determines the workpiece coordinate system. The corresponding values of offset data parameter **P10～13** in workpiece coordinate system are the integral offset of the 6 workpiece coordinate systems. The origins of these workpiece coordinate systems can be specified by inputting the coordinate offset in MDI mode or by setting data parameters **P15～P43**. These 6 workpiece coordinate systems are set up by the distances from machine zero to their respective coordinate system origins.

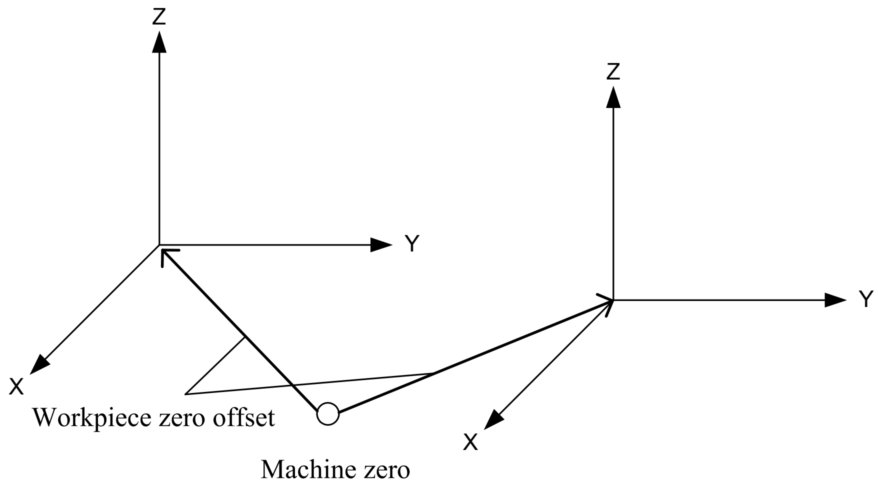


Fig. 4-2-8-2

Example: N10 G55 G90 G00 X100 Y20；

N20 G56 X80.5 Z25.5；

In the above example, when block N10 is executed, the tool traverses rapidly to the position in workpiece coordinate system G55 (X=100，Y=20). When block N20 is executed , the tool traverses rapidly to the position in workpiece coordinate system G56, and the absolute coordinates shifts to the coordinates (X=80.5，Z=25.5) in workpiece coordinate system G55 automatically.

4.2.9 Additional workpiece coordinate system

Another 50 additional workpiece coordinate systems can be used besides the 6 workpiece coordinate systems (G54 to G59).

Format: G54 Pn

Pn：A code to specify the additional coordinate system with a range of 1~50.

The setting and restrictions of the additional workpiece coordinate system are the same as those of workpiece coordinate systems G54~G59.

G10 can be used to set the offset value of the workpiece zero in the additional workpiece system, as shown below:

By instruction G10 L20 Pn X_Y_Z_;

n=1 to 50： Code of additional workpiece coordinate system

X_Y_Z_ ： For setting axis address and offset value for workpiece zero offset.

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

For incremental instruction (G91), the specified value is added to the current offset value to produce a new offset value.

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

When the address P of the additional workpiece coordinate system is in the same block with other instructions containing address P, they share this P address together.

4.2.10 Selecting machine coordinate system G53

Format: G53 X_ Y_ Z_

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

Explanations:

1. While G53 is used in the program, the instruction coordinates behind it should be the ones in the machine coordinate system and the machine will rapidly position to the specified location.
2. G53 is a non-modal instruction, which is only effective in the current block. It does not affect the coordinate system defined before.

Restrictions:

Selecting current coordinate system G53

When the position on the machine is specified, the tool traverses to the position rapidly. G53 used for selecting the machine coordinate system is a non-modal instruction, i.e., it is effective only in the block specifying the machine coordinate system. Absolute value G90 should be specified for G53. If G53 is specified in incremental mode (G91), the instruction G91 will be ignored (i.e., G53 is still in G90 mode without changing G91 mode). The tool can be specified to move to a special position on the machine, e.g. using G53 to write a moving program to move the tool to the tool changing position.

.

Note: When G53 is specified, the tool radius compensation and tool length offset are cancelled temporarily. They will resume in the next compensation axis block buffered.

4.2.11 Floating coordinate system G92

Format: G92 X_ Y_ Z_

Function: for setting the floating workpiece coordinate system. The current tool absolute coordinate values in the new floating workpiece coordinate system are specified by 3 instruction parameters. This instruction does not cause the movement axis to move.

Explanation:

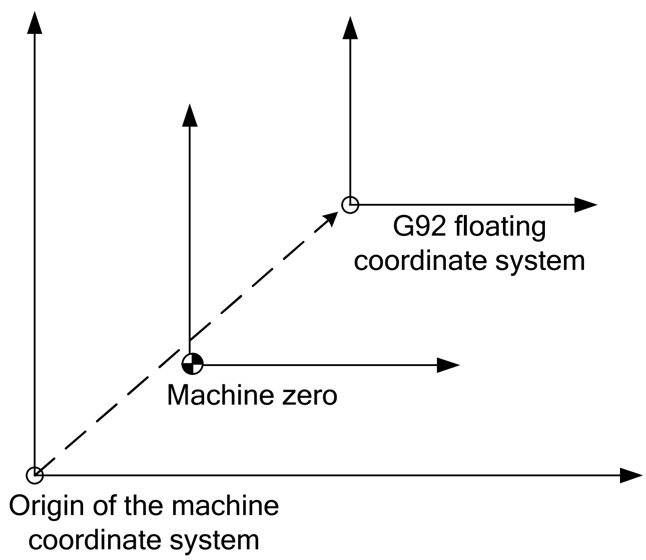


Fig. 4-2-11-1

1. As shown in Fig. 4-2-11-1, the corresponding origin of the G92 floating coordinate system is the value in machine coordinate system, which is not related to the workpiece coordinate system.

   G92 setting is effective in the following conditions:

   1) Before the workpiece coordinate system is called
   2) Before the machine zero return

   The G92 floating coordinate system is often used for the alignment for temporary workpiece machining. It is usually specified at the beginning of the program or in MDI mode before the program auto run.

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

   1） Determining the coordinate system with tool nose

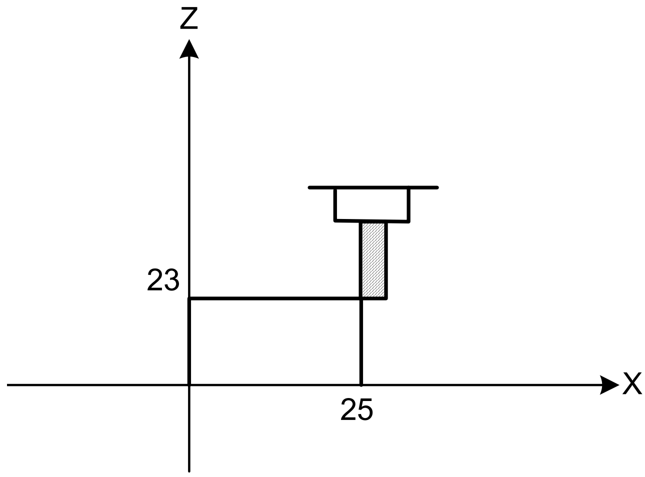


Fig. 4-2-11-2

As shown in Fig. 4-2-11-2, G92 X25 Z23, the tool nose position is taken as point（X25， Z23） in the floating coordinate system.

2） Taking a fixed point on the tool holder as the reference point of the coordinate system

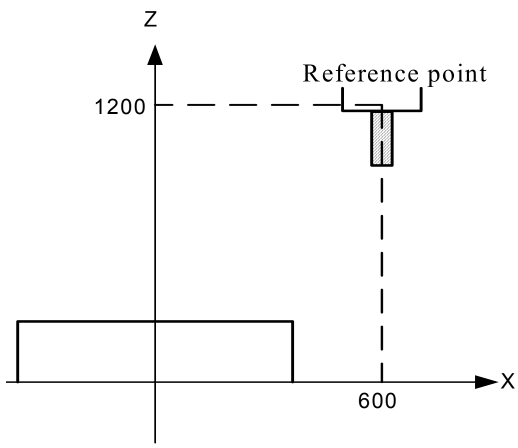


Fig. 4-2-11-3

As Fig. 4-2-11-3 shows, specify the workpiece coordinate system by instruction “G92 X600 Y1200” (taking a certain reference point on the tool holder as the tool start point). Taking a reference point on the tool holder as the start point, if the tool moves by the absolute value instruction in the program, the specified position to which the reference point is moved must adds the tool length compensation, the value of which is the difference between reference point and tool nose.

Note 1: If G92 is used to set the coordinate system in the tool offset, the coordinate system for tool length compensation is the one set by G92 before the tool offset is added.

Note 2: For tool radius compensation, the tool offset is cancelled with G92.

4.2.12 Plane selection G17/G18/G19

Format: G17/G18/G19

Function: Select planes for circular interpolation, tool radius compensation, drilling or boring with G17/G18/G19.

Explanation: It has no instruction parameter. G17 is the default plane at Power On. The default plane at Power On can also be determined by bit parameters **N0：31#1, #2 and #3.** The relation between instruction and plane is as follows:

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

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

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

The plane keeps unchanged in the block in which G17，G18 or G19 is not specified.

Example: G18 X_ Z_；ZX plane

G0 X_ Y_； Plane is unchanged (ZX plane)

In addition, the movement instruction is irrelevant to the plane selection. For example, in the following instruction, Y is not on the ZX plane, and its movement is irrelevant to the ZX plane.

G18Y_；

Note: Only the canned cycle in G17 plane is supported at present. For criterion or astringency, it is strongly recommended that the plane be clearly specified in corresponding blocks when programming, especially in the case that a system is used by different operators. In this way, accidents or abnormality caused by program errors can be avoided.

4.2.13 Polar coordinate start/cancel G16/G15

Format: G16/G15

Function:

G16: Starts the polar coordinate mode of the positioning parameter

G15: Cancels the polar coordinate mode of the positioning parameter

Explanation:

With no instruction parameter.

By setting G16, the coordinate value can be input with polar coordinate radius and angle. The positive direction of the angle is the counterclockwise direction of the 1^{st} axis in the selected plane, and the negative direction is the clockwise direction. Both the radius and angle can use either absolute instruction or incremental instruction (G90 or G91). After G16 appears, the 1^{st} axis of the positioning parameter of the tool movement instruction is the polar radius in the polar coordinate system, and the 2^{nd} axis is the polar angle in the polar coordinate system.

G15 can cancel the polar coordinate mode and thus return the coordinate value to the rectangular coordinate mode.

Specifying polar coordinate origin

1. In G90 absolute mode, when G16 is specified, the zero point of the workpiece coordinate system is set as the origin of the polar coordinate system.

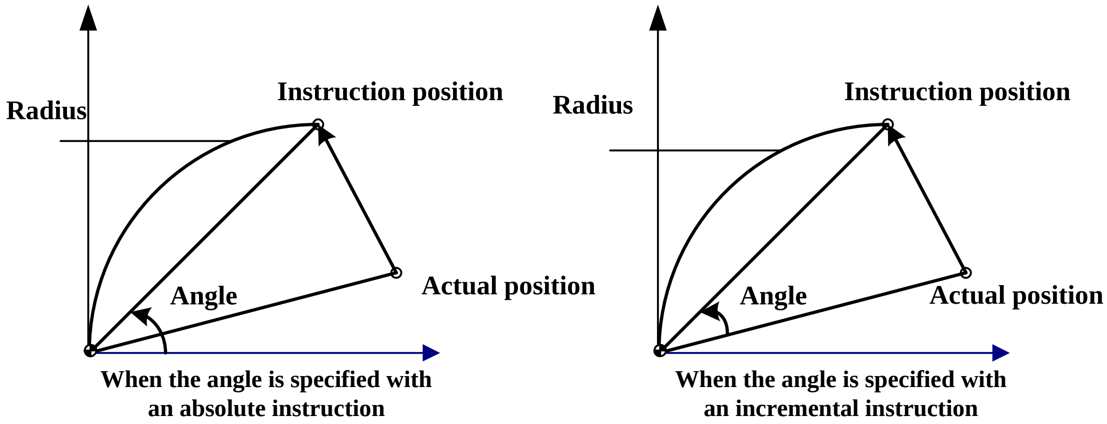


Fig. 4-2-13-1

2. In G91 absolute mode, when G16 is specified, the current point is set as the origin of the polar coordinate system.

Example: bole hole circle (the zero point of the workpiece coordinate system is set as the origin of the polar coordinate system, and X—Y plane is selected**)**

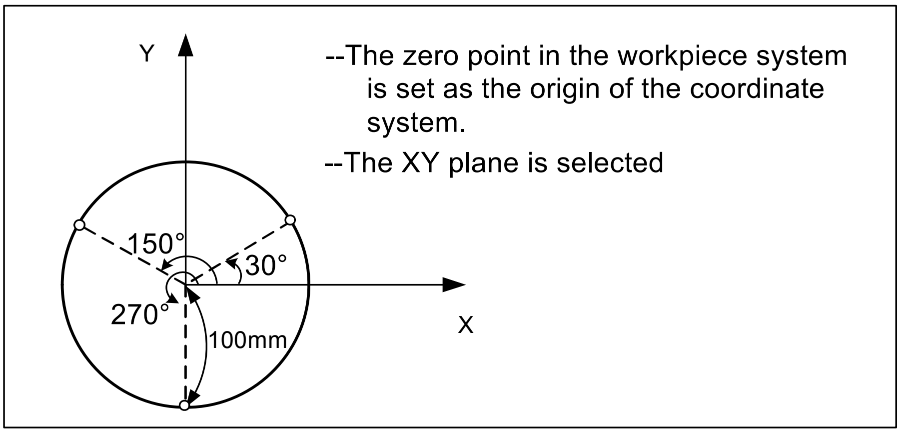


Fig. 4-2-13-2

- Specifying angles and a radius with absolute value

G17 G90 G16; Specifying the polar coordinate instruction and selecting XY plane, setting the zero point of the workpiece coordinate system as the origin of the polar coordinate system.

G81 X100 Y30 Z-20 R -5 F200; Specifying a distance of 100mm and an angle of 30°.

Y150; Specifying a distance of 100mm and an angle of 150°

Y270; Specifying a distance of 100mm and an angle of 270°

G15 G80; Cancelling the polar coordinate instruction

- Specifying angles with incremental value and a polar radius with absolute value

G17 G90 G16; Specifying the polar coordinate instruction and selecting XY plane, setting the zero point of the workpiece coordinate system as the origin of the polar coordinate system.

G81 X100 Y30 Z-20 R -5 F200; Specifying a distance of 100mm and an angle of 30°.

G91 Y120; Specifying a distance of 100mm and an angle of 150°.

Y120; Specifying a distance of 100mm and an angle of 270°.

G15 G80; Cancelling the polar coordinate instruction

Moreover, when programming by polar coordinate system, the current coordinate plane setting should be considered. The polar coordinate plane is related to the current coordinate plane. E.g. in G91 mode, if the current coordinate plane is specified by G17, the components of X axis and Y axis of the current tool position are taken as the origin. If the current coordinate plane is specified by G18, the components of Z axis and X axis of the current tool position are taken as the origin.

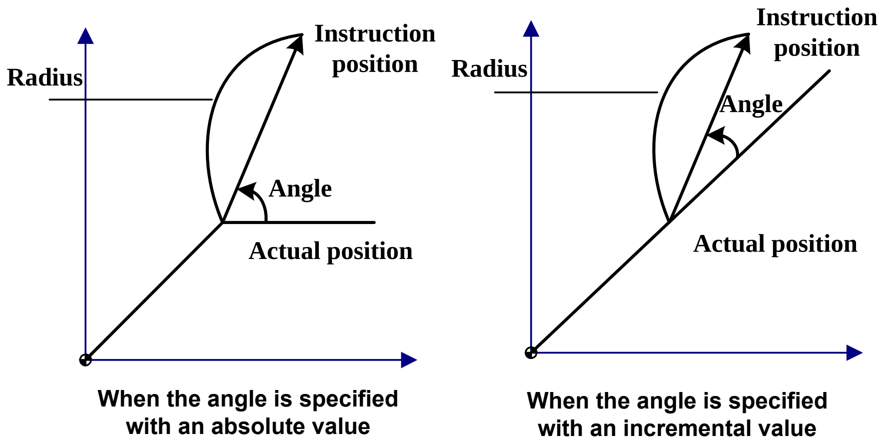


Fig. 4-2-13-3

If the positioning parameter of the first hole cycle instruction behind G16 is not specified, the system takes the current tool position as the default positioning parameter of the hole cycle. At present, the first canned cycle instruction behind the polar coordinate must be complete, or the tool movement is incorrect.

The positioning words of the positioning parameters of the tool movement instructions behind G16, except for the hole cycle, are relevant to the actual plane selection mode. After the polar coordinate is cancelled with G15, if there is a movement instruction following it, the default current tool position is the start point of this movement instruction.

4.2.14 Scaling in a plane G51/G50

Format:

G51 X_ Y_ Z_ P_ （X.Y.Z: absolute instruction for the scaling center coordinates, P: each axis is scaled up or down at the same rate of magnification)

… Scaled machining blocks

G50 Scaling cancelled

Or G51 X_ Y_Z_ I_ J_ K_（Each axis is scaled up and down at different rates（I、J、K）of magnification）

… Scaled machining blocks

G50 Scaling cancelled

Function:

G51 scales up and down the programmed figure in the same or different rate taking a specified position as its center. It is suggested that the G51 be specified in a separate block (or unexpected results may occur, resulting in workpiece damage and personal injury) and cancelled with G50.

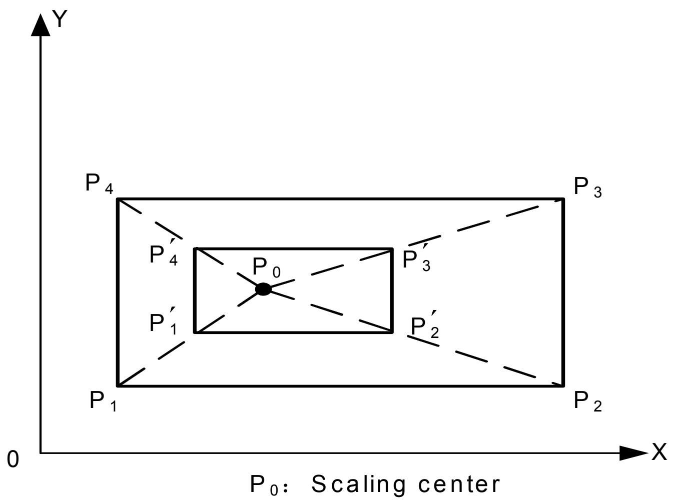


Fig. 4-2-14-1 Scaling up and down (P1P2P3P4→ P1'P2'P3'P4')

Explanation:

1. Scaling center: G51 can be specified with three positioning parameters X_Y_Z_, all of which are optional parameters. These positioning parameters are for specifying the scaling center of G51. If they are not specified, the system assumes the tool current position as the scaling center. Whether the current positioning mode is in absolute or incremental mode, the scaling center is always specified with the absolute positioning mode. Moreover, the parameters of instruction G51 are also expressed with rectangular coordinate system in polar coordinate G16 mode.

Example: G17 G91 G54 G0 X10 Y10；

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

G1 Y90； Parameter Y is still in incremental mode.

2. Scaling: Either in G90 mode or G91 mode, the rate of magnification is always expressed with absolute mode.

   The rate of magnification can be set either in parameters or in programs. Data parameters P331~P333 correspond to the magnifications of X, Y and Z respectively.

   If there is no scaling instruction specified, the setting value of data parameter P330 is used for scaling.

   If the parameter values of parameter P or I, J and K are negative, the mirror image is applied for the corresponding axis.

3. Scaling setting: The effectiveness of scaling is set by parameter **No：60#5,** The effectiveness of the X axis scaling is set by bit parameter NO：47#3, the effectiveness of the Y axis scaling is set by bit parameter NO：47#4, the effectiveness of the Z axis scaling is set by bit parameter NO：47#5, and the scaling rate of each axis is set by bit parameter NO：47#6 (0: instructed with P, 1: instructed with I, J, K.).

4. Scaling cancel: After the scaling followed by a movement instruction is cancelled by G50, the current tool position is regarded as the start point of this movement instruction by default.

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

6. Even if different magnifications are specified for circular interpolation and each axis, the tool will not trace an ellipse.

   When the magnification for each axis is different and the circular interpolation is programmed with radius R, the interpolation figure is shown in fig. **4-2-14-2** (in the example below, the magnification for X axis is 2, for Y axis is 1)**.**

```
G90 G0 X0 Y100;
G51 X0 Y0 Z0 I2 J1;
G02 X100 Y0 R100 F500;
```

Above instructions are equivalent to the following ones:

```
G90 G0 X0 Y100;
G02 X200 Y0 R200 F500;
```

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

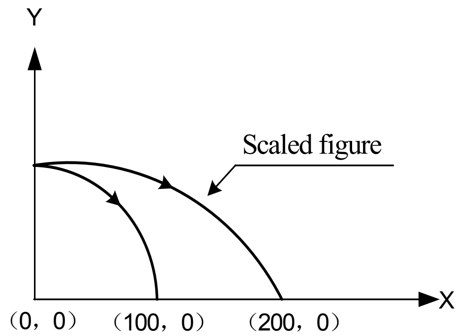


Fig. 4-2-14-2 Scaling for circular interpolation 1

When the magnifications of the axes are different and the circular interpolation is programmed with I, J and K, an alarm is given if the arc does not exist.

7. Scaling has no effect on the tool offset value, see Fig. **4-2-14-3**.

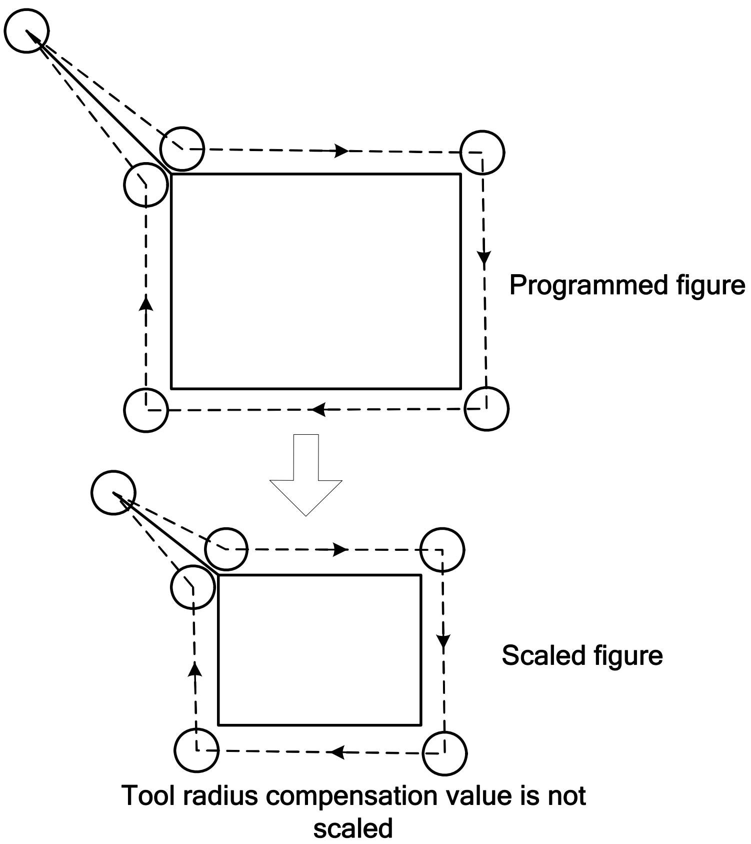


Fig. 4-2-14-3 Scaling for tool radius compensation

Example of a mirror image program:

Main program:

G00 G90；

M98 P9000；

G51 X50.0 Y50.0 I-1 J1；
M98 P9000；

G51 X50.0 Y50.0 I-1 J-1；

M98 P9000；

G51 X50.0 Y50.0 I1 J-1；

M98 P9000；

G50；

M30；

Subprogram:

O9000；

G00 G90 X60.0 Y60.0；

G01 X100.0 F100；

G01 Y100；

G01 X60.0 Y60.0；

M99；

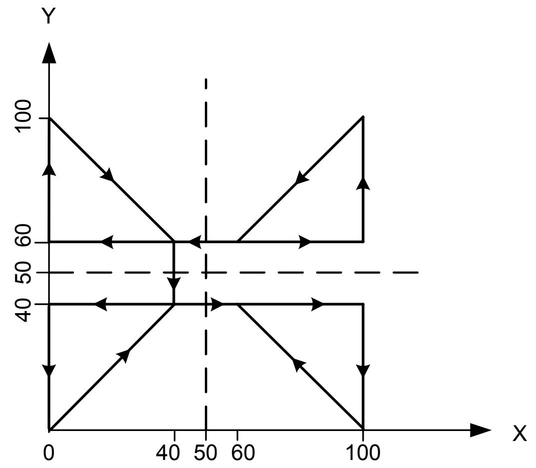


Fig. 4-2-14-4

Restrictions:

1. When the canned cycle is executed in scaling mode, the system only scales up or down the hole positioning data rather than point R, value Q, point Z at hole bottom and dwell time P at hole bottom. For example:

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

   2）Fine boring cycle（G76）.

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

2. In MANUAL mode, the traverse distance cannot be increased or decreased by scaling.

Note 1: The position displays the coordinate values after scaling.

Note 2: The results are as follows when a mirror image is applied to one axis of a specified plane:

1）Circular instruction………………. Direction of rotation is reversed

2）Tool radius compensation C………. Direction of offset is reversed

3）Coordinate system rotation……………Rotation angle is reversed

4.2.15 Coordinate system rotation G68/G69

For the workpiece which consists of many figures with the same shapes, users can program using the coordinate rotation function, i.e., write a subprogram to the figure unit, and then call the subprogram using rotation function.

Format: G17 G68 X_ Y_ R_；

Or G18 G68 X_ Z_ R_；

Or G19 G68 Y_ Z_R_ ；

G69；

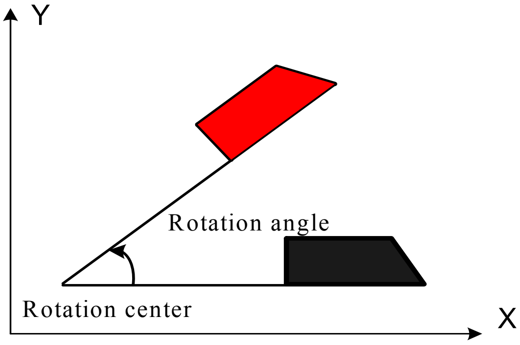


Fig. 4-2-15-1

Function: G68 rotates the programmed shape in a plane taking a specified center as its origin. G69 is used for cancelling the coordinate system rotation.

Explanation:

1. G68 has two positioning parameters, both of which are optional ones. They are used for specifying the rotation center. If the rotation center is not specified, the system assumes the current tool position as the rotation center. The positioning parameters are relative to the current coordinate plane, e.g., X and Y for G17; X and Z for G18; Y and Z for G19.
2. When the current positioning mode is the absolute mode, the system assumes the specified point as the rotation center. When the positioning mode is the relative mode, the system specifies the current point as the rotation center. G68 can also use an instruction parameter R, of which the value is the rotation angle, with degree as its unit. A positive value of R indicates the counterclock rotation. When there is no rotation angle instruction in the coordinate rotation, the rotation angle to be used is set by data parameter **P329.**
3. In G91 mode, the system takes the current tool position as the rotation center; the rotation angle by increment is set by bit parameter NO: 47#0 (rotation angle of coordinate system, 0: by absolute instruction; 1: by G90/91 instruction).
4. When programming, please note that no plane selection is allowed when the system is in rotation mode, otherwise an alarm occurs.
5. In coordinate system rotation mode, G codes for reference point return （G27～G30 etc.）and coordinate system specification（G52～G59 , G92 etc.）cannot be specified. They should be specified after the scaling is cancelled if needed.
6. After coordinate system rotation, perform operations such as the tool radius compensation, tool length compensation, tool offset and other compensation.
7. If the coordinate system rotation is performed in scaling mode (G51), the rotation center coordinate values will be scaled rather than the rotation angle. When a movement instruction is given, the scaling will be executed first, then the coordinate system rotation.

Example 1: Rotation:

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

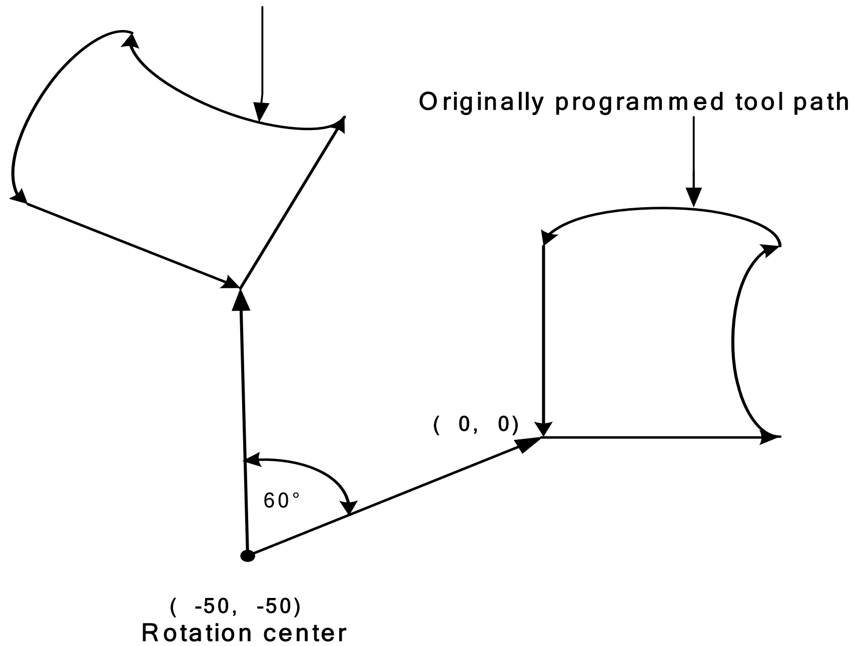


Fig. 4-2-15-2

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；
M30；
```

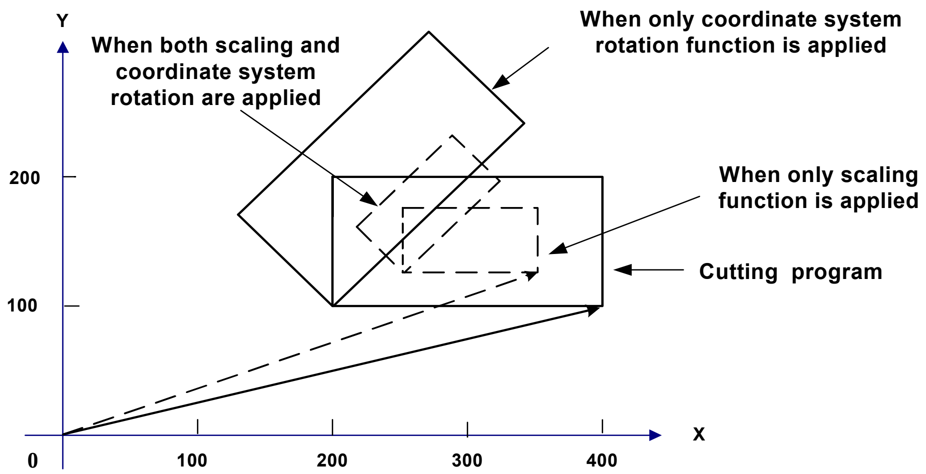


Fig. 4-2-15-3

Example 3: Repetition of G68

By program (main program)

```
G92 X0 Y0 Z20 G69 G17;
M3 S1000;
G0 Z2;
G42 D01;                  (tool offset setting)
M98   P2100(P02100);      (subprogram call)
M98   P2200L7;            （call 7 times）
G40;
G0 G90 Z20;
X0Y0;
M30;
```

Subprogram 2200

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

Subprogram 2100

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

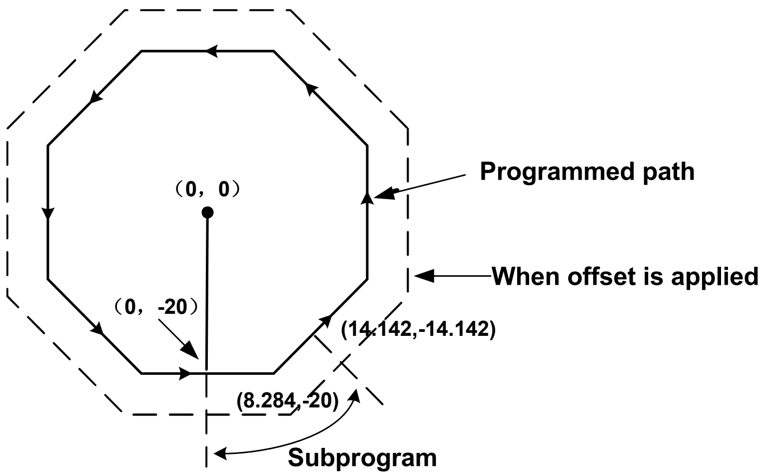


Fig. 4-2-15-4

Book 1 Programming

4.2.16 Skip function G31

Format: G31 X_Y_Z_

Function: Linear interpolation can be specified after G31 in the same way as after G01. During the execution of this instruction, if an external skip signal is input, the execution of the instruction is interrupted and the next block is executed. When the machining end point is not programmed, but it is specified using a signal from the machine, use the skip function. For example, use it for grinding. The function is used for measuring the dimension of a workpiece as well.

Explanation:

1. G31 is a non-modal G instruction only effective in the block in which it is specified.
2. When tool radius compensation is being executed, if G31 is specified, an alarm will occur. Therefore, the tool radius compensation should be cancelled before G31.

Example:

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

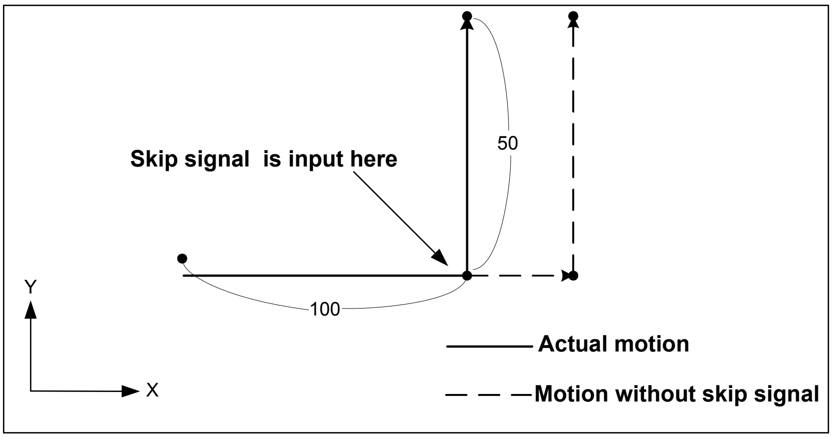


Fig. 4-2-16-1 The next block is the single-axis movement specified by incremental values

The next block after G31 is a single-axis movement specified by absolute values, as shown in fig. **4-2-16-2:**

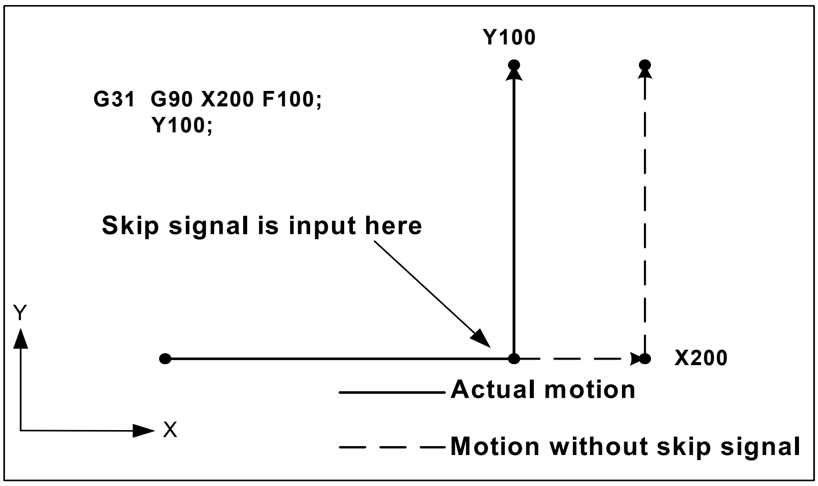


Fig. 4-2-16-2 The next block is a single-axis movement specified by absolute values

The next block after G31 is two-axis movement specified by absolute values, as shown in fig. **4-2-16-3**:

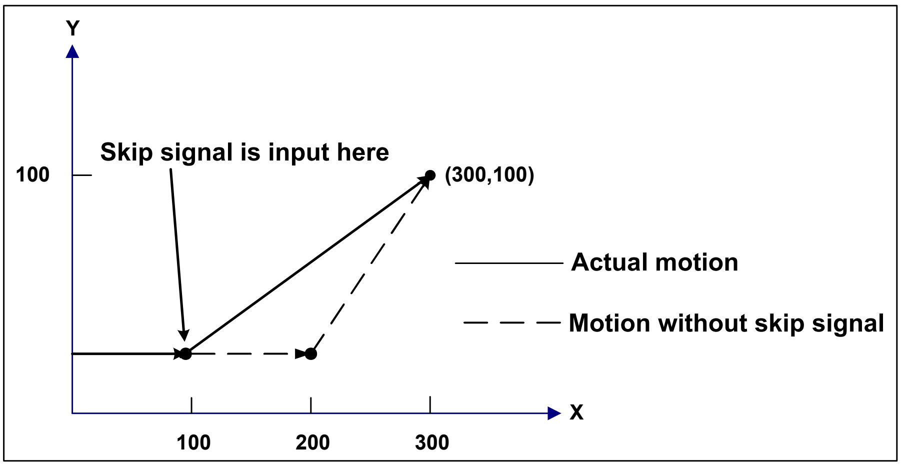


Fig. 4-2-16-3 The next block is two-axis movement specified by absolute values

Note: The setting can be done by bit parameter NO：02#6 [skip signal SKIP, (0:1, 1:0)].

4.2.17 Inch/metric conversion G20/G21

Format: G20: inch input
G21: metric input
Function: They are used for the inch/metric input conversion in a program.

Explanation:

After inch/metric conversion, the units of the following values are changed:

Feedrate specified by F code, position instruction, workpiece zero offset value, tool compensation value, scale unit of MPG and movement distance in incremental feeding.

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

Note: 1. When the inch input is converted to metric input or vice versa, the tool compensation value must be preset according to the lest input incremental unit.

2. After inch input is converted to metric input or vice versa, for the first G28, the operation from the intermediate point is the same as that of manual reference point return.
3. When the least input incremental unit is different from the least instruction incremental unit, the maximum error is half of the least instruction unit and this error is not accumulated.
4. Program inch/metric input can be set by bit parameter N0：00#2
5. Program inch/metric output can be set by bit parameter N0：03#0.
6. G20 or G21 must be specified in a separate block.

4.2.18 Optional angle chamfering/corner rounding

Format: ，**L_**：Chamfering
，**R_**：Corner rounding

Function: When the instructions above are added to the end of the block specifying linear interpolation (G01) or circular interpolation(G02、G03), a chamfering or corner rounding is added automatically outside the corner during machining. Blocks specifying chamfering or corner rounding arc can be specified consecutively.

Explanation:

1. Chamfering: after L, specify the distance from the virtual corner point to the start and the end points of the corner. The virtual corner point is the corner point that exists if chamfering is not performed. As the following figure shows:

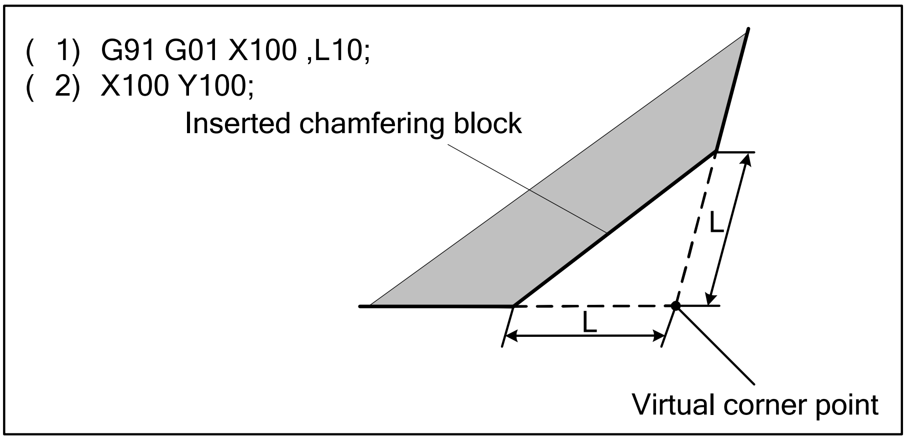


Fig. 4-2-18-1

2. Corner R: after R, specify the radius for the corner rounding, as shown below:

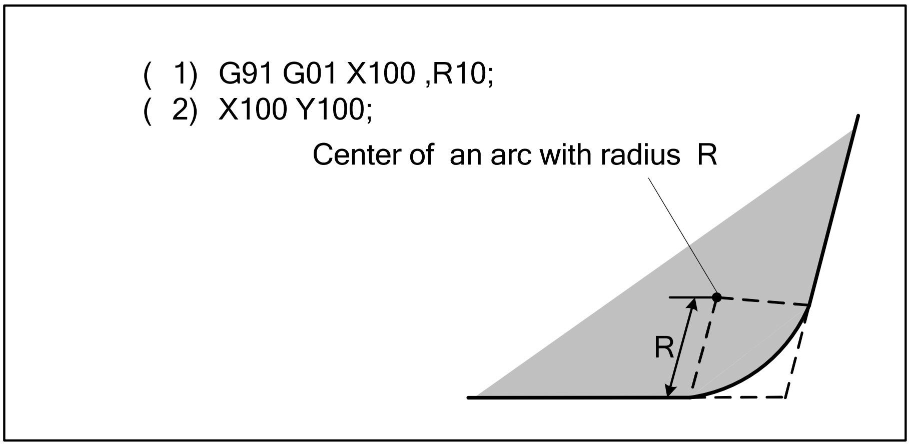


Fig. 4-2-18-2

Restrictions:

1. Chamfering and corner rounding can only be performed in a specified plane, and these functions cannot be performed for parallel axes.
2. If the inserted chamfering or corner rounding block causes the tool to go beyond the original interpolation move range, an alarm is issued.
3. Corner rounding cannot be specified in a threading block.
4. When the values of chamfering and corner rounding are negative, their absolute values are used in the system.

4.3 Reference point G instruction

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

There are 3 instructions for the 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 a specified axis by G28; or be moved automatically from the reference point to a specified point via an intermediate point along a specified axis by G29.

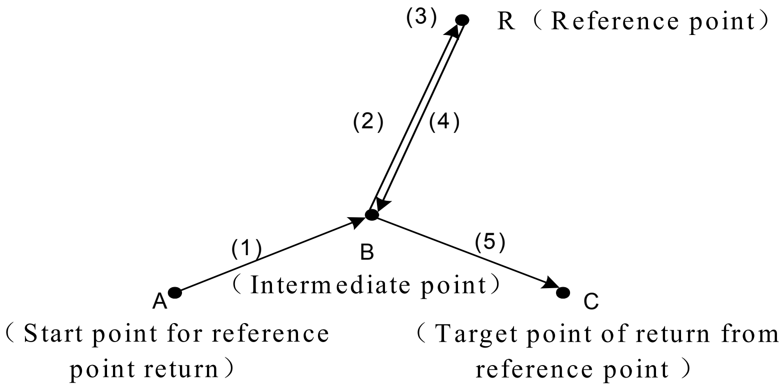


Fig. 4-3-1

4.3.1 Reference point return G28

Format: G28 X_ Y_ Z_

Function: G28 is for the operation of returning to the reference point (a specific point on the machine tool) via intermediate point.

Explanation:

Intermediate point:

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

Note:

The coordinate values of the intermediate point are stored in the CNC system. Only the axis coordinate values specified by G28 are stored each time, for the other axes not specified by G28, the coordinate values specified by G28 before are used. If the current default intermediate point of the system is unknown when G28 is used, it is recommended that each axis be specified with one. Please take a consideration according to block N5 in the following example.

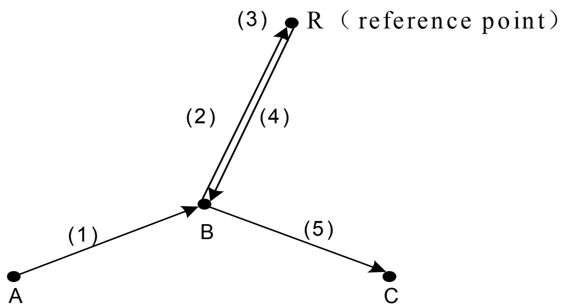


Fig. 4-3-1-1

1. The action of block G28 can be divided as follows: (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 effective only in the current block.
3. Single-axis reference point return and multi-axis reference point return are available. The intermediate point coordinates are saved by the system when the workpiece coordinate system is changed.

Example:

N1 G90 G54 X0 Y10；

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

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

N4 G01 X20；

N5 G28 Y60 ；intermediate point is (X40，Y60). As its coordinate in X axis is not specified, the X40 specified in the previous G28 is used. 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 point (60, 20) via the intermediate point (40，60) in G55 workpiece coordinate system from the reference point.

The G28 instruction can automatically cancel the tool compensation, but this instruction is only used in automatic tool change mode (i.e. changing the tool at the reference point after reference point return). Therefore, the tool radius compensation and tool length compensation, in principle, should be cancelled before the use of this instruction. See data parameters P45～P48 for the 1st reference point setting.

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 2nd, 3rd, 4th 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_；2nd reference point return (P2 can be omitted)

G30 P3 X_ Y_ Z_；3rd reference point return

G30 P4 X_ Y_ Z_；4th reference point return

Function: G30 performs the operation of returning to the specified reference point via the intermediate point specified by G30.

Explanation:

1. X_ Y_ Z_； Instruction for specifying the intermediate point (absolute/ incremental)
2. The setting and restrictions of instruction G30 are the same as those of instruction G28. See data parameter P50～63 for the 2nd, 3rd, 4th reference point setting.
3. The G30 code can also be used together with G29 code (return from reference point), of which the setting and restrictions are identical with those of G28 code.

4.3.3 Automatic return from reference point G29

Format: G29 X_ Y_ Z_

Function: G29 performs the operation of returning to the specified point via the intermediate point specified by G28 or G29 from the reference point (or the current point).

Explanation:

1. The action of block G29 can be divided as follows:（refer to Fig.4-3-1-1）：
   (1) Positioning to the intermediate point (point R→point B) specified by G28 or 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 the current block. In general, the instruction Return from Reference Point should be specified immediately after instruction G28 or G29.
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) of the return from the reference point, all of which can be expressed by absolute or incremental instruction. The instruction specifies the incremental value departed from the intermediate point in incremental programming. If the value is not specified for an axis, it means the axis has no movement relative to the intermediate point. The G29 instruction followed by only one axis means the single axis return with no action performed to other axes.

Example:

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 that the component in X axis should be 60 in incremental programming.

The values of the intermediate point specified by G29 are assigned by G28 or G30. See the explanation of instruction G28 for the definition, specification and system default of the intermediate point.

4.3.4 Reference point return check G27

Format: G27 X_ Y_ Z_

Function: G27 performs the reference point return check, with the reference point specified by X_ Y_

Explanation:

1. G27 instruction positions the tool at the rapid traverse speed. If the tool reaches the reference point, the indicator for reference point return lights up. However, if the position the tool reaches is not the reference point, an alarm is issued.
2. In machine lock mode, even if G27 is specified and the tool has automatically returned to the reference point, the indicator for return completion does not light up.
3. In the offset mode, the position to be reached by the tool specified with G27 instruction is the position obtained after the offset is added. Therefore, if the position with the offset added to it is not the reference point, the indicator does not light up, and an alarm is issued. Usually the tool offset should be cancelled before the use of G27 instruction.
4. The coordinate position of X, Y and Z specified by G27 is the position in the machine coordinate system.

4.4　Canned cycle G code

The canned cycle uses a single block containing G functions to achieve the machining action which needs to be done with multiple blocks to simply the programming, making the programming easier for programmers (in this system only the canned cycle in G17 plane is available).

General process of canned cycle:

A canned cycle consists of a sequence of 6 operations, as shown in fig. **4-4-1.**

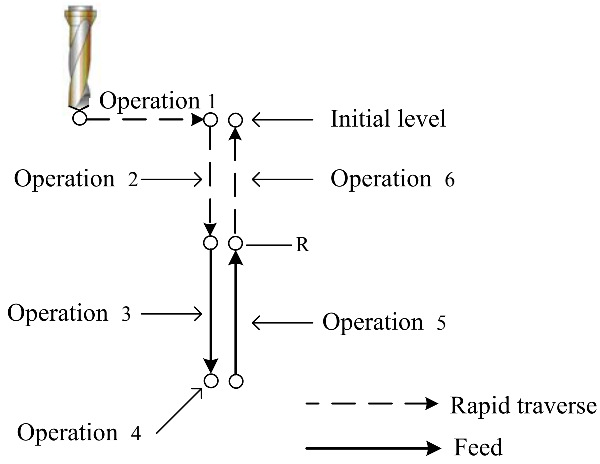


Fig. 4-4-1

Operation 1: Positioning of axes X and Y (another axis can be included)
Operation 2: Rapid 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: Rapid traverse to the initial point

Positioning is performed in XY plane, and hole machining is performed along Z axis. It is defined that a canned cycle operation is determined by 3 types, which are specified by G codes respectively.

1) Data type
   G90 absolute mode; G91 incremental mode
2) Return point plane
   G98 initial level; G99 point R level
3) Groove machining type
   G22, G23, G24, G25, G26, G32, G33, G34, G35, G36, G37, G38.
4) Hole machining type
   G73, G74, G76, G81～G89.

Initial point Z level and point R level

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

Point R level: It is also called safety level. It is the position in Z axis which is generally located a certain distance above the workpiece surface to prevent the tool from colliding with the workpiece and ensure an enough distance for deceleration when the rapid traverse is switched to cutting feed in canned cycle.

G73/G74 /G76/G81～G89 specifies all the data of canned cycle (hole position data, hole machining data and number of repeats) into a single block.

Z, R: If either of hole bottom parameter Z and R is missing when the first hole drilling is executed, the system only changes the mode, with no Z axis action executed.

The format of hole machining is as follows:

（G17）G__ X__ Y__ Z__ R__ Q__ P__ F__ K__;

Hole machining data

Hole position data

Hole machining type

The meanings of hole position data and hole machining data are shown in table **4-4-1.**

Table 4-4-1

Designation	Parameter word	Explanation
Hole machining	G	Refer to table **4-4-3**, and note the restrictions above.
Hole position data	X，Y	The hole position is specified by either absolute value or incremental value and the control is identical to that of G00 positioning.
Hole machining data	Z	As Fig. 4-4-2 shows, the distance from point R level to the hole bottom is specified by incremental values, or the hole bottom coordinates are specified by absolute values. As shown in fig. 4-4-1, the federate is the speed specified by F in operation 3, while it is the rapid traverse speed or the speed specified F instruction in operation 5 depending on different hole machining types.
	R	In Fig. 4-4-2, the distance from the initial level to point R level is specified by incremental value, or point R level coordinates are specified by absolute values. The federates, shown in fig. 4-4-1, are both rapid traverse in operations 2 and 6.
	Q	It is used to specify the cut-in value each time in G73 or G83, or the parallel movement value (incremental 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 with unit of ms. The min. value of the parameter can be set by number parameter P281, and the max. value by data parameter P282.
	F	It is used for specifying the cutting federate.

Designation	Parameter word	Explanation
	K	The number of repeats is specified in K_, which is only effective in the block in which it is specified. If it is omitted, the default is 1 time. The maximum drilling times are 99999. When the value is negative, its absolute value is executed. When the value is 0, only the mode is changed, with no drilling operation executed.

Restrictions:

- The canned cycle G instructions are modal ones, which remain effective till they are cancelled by a G code for cancelling it.
- G80 and G codes in group 01 are used for cancelling the canned cycle.
- Once the hole machining data in canned cycle is specified, it is retained till the cycle is cancelled. All the required hole machining data should be specified at the beginning of the canned cycle, and only the updated data needs to be specified in the subsequent canned cycle.

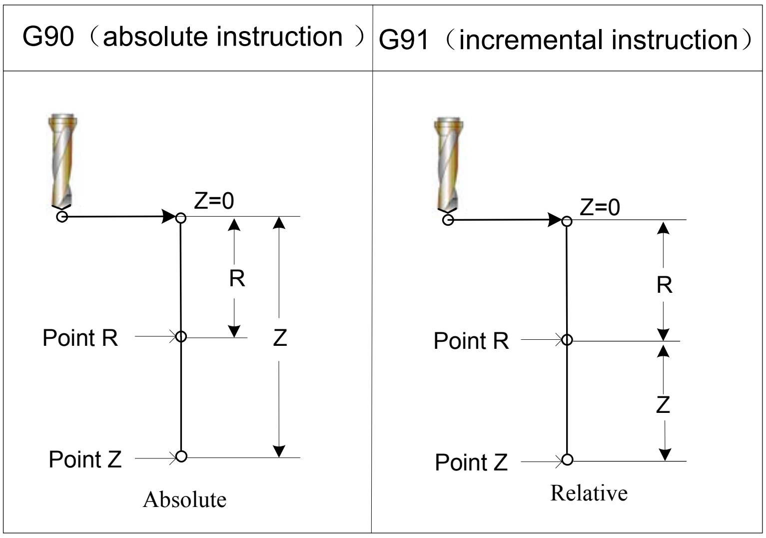
Note 1: The feedrate specified by F remains effective even if the canned cycle is cancelled.
Note 2: The scaling for Z axis (cutting axis direction) is invalid in the canned cycle.
Note 3: In single block mode, the canned cycle uses the 3-stage machining type, i.e. positioning→R level→initial level.
Note 4: In the canned cycle, when the system bit parameter NO：36#1 is 1, if reset or emergency stop is performed, both the hole machining data and hole position data will be cleared. Examples for data remaining and data clearing above are shown in the following table:

Table 4-4-2

Sequence	Data designation	Explanation
①	G00X_M3；	
②	G81X_Y_Z_R_F_；	Specify values for Z, R and F in the beginning.
③	Y_；	G81，Z-R-F- can all be omitted since the hole machining mode and data are the same as those specified in ②. Drill the hole for the length Y once by G81.
④	G82X_P_；	Move only in X axis direction relative to the position of hole ③. Perform hole machining by G82 using the hole machining data Z, R and F specified in ② and P in ④.
⑤	G80X_ Y_	Hole machining is not performed. Cancel all the hole machining data.
⑥	G85X_Z_R_P_；	Since all the data are cancelled in ⑤, Z and R need to be re-specified. F is identical with that in ② , so it can be omitted. P is not required in this block and it is saved.
⑦	X_Z_；	It is the hole machining identical with that in ⑥ except for Z value. And there is movement only in X axis at the hole position.
⑧	G89X_Y_；	Perform G89 hole machining using Z specified in ⑦, R and P in ⑥, F in ② as the machining data.
⑨	G01X_Y_；	Cancel hole machining mode and clear hole machining 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 processed according to the specified signs + and -)



(A) (B)

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 for the last drilling operation. The initial level does not change even if the drilling is performed in G99 mode. The following figure illustrates the operations of G98 and G99.

G98 is the system default mode.

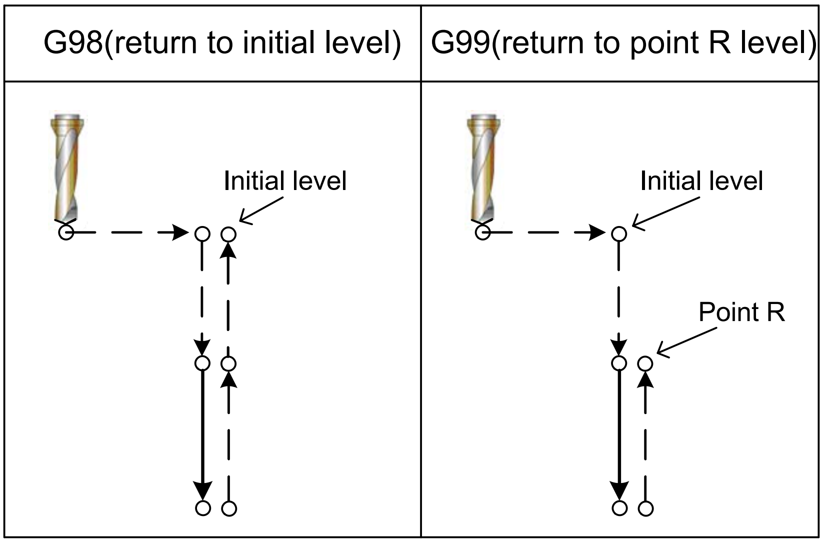


Fig. 4-4-3

The following symbols are used for the canned cycle illustration:

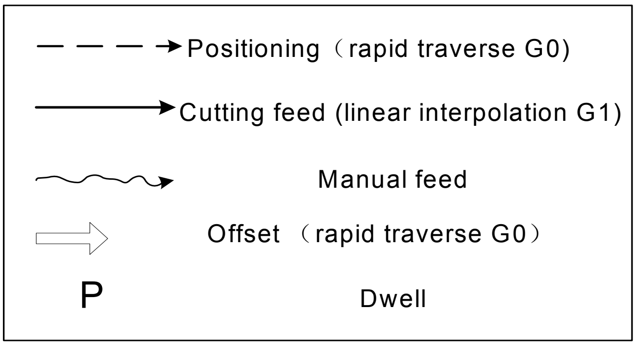


Fig. 4-4-4

Canned cycle comparison table（G22～G89）

Table 4-4-3

G code	Driling (-Z direction)	Operation at hole bottom	Retraction operation (+Z direction)	Application
G22	Cutting feed		Rapid traverse	CCW inner circular groove rough milling
G23	Cutting feed		Rapid traverse	CW inner circular groove rough milling
G24	Cutting feed		Rapid traverse	CCW finish-milling cycle within a circle
G25	Cutting feed		Rapid traverse	CW finish-milling cycle within a circle
G26	Cutting feed		Rapid traverse	CCW outer circle finish-milling cycle
G32	Cutting feed		Rapid traverse	CW outer circle finish-milling cycle
G33	Cutting feed		Rapid traverse	CCW rectangle groove rough milling
G34	Cutting feed		Rapid traverse	CW rectangle groove rough milling
G35	Cutting feed		Rapid traverse	CCW rectangle groove inner finish-milling cycle
G36	Cutting feed		Rapid traverse	CW rectangle groove inner finish-milling cycle
G37	Cutting feed		Rapid traverse	CCW rectangle outside finish-milling cycle
G38	Cutting feed		Rapid traverse	CW rectangle outside finish

				milling cycle
G73	Intermittent feed		Rapid traverse	High-speed peck drilling cycle
G74	Cutting feed	Dwell→spindle CCW	Rapid traverse	Counter taping cycle
G76	Cutting feed	Oriented spindle stop	Rapid traverse	Fine boring
G80				Cancel
G81	Cutting feed		Rapid traverse	Drilling, spot drilling
G82	Cutting feed	Stop	Rapid traverse	Drilling, counter boring
G83	Intermittent feed		Rapid traverse	Peck drilling cycle
G84	Cutting feed	Stop→spindle CCW	Cutting feed	Taping
G85	Cutting feed		Cutting feed	Boring
G86	Cutting feed	Spindle stop	Rapid traverse	Boring
G87	Cutting feed	Spindle CCW	Rapid traverse	Boring
G88	Cutting feed	Stop→spindle CCW	Manual	Boring
G89	Cutting feed	Dwell	Cutting feed	Boring

Restrictions:

Tool radius offset (D) is ignored during the canned cycle positioning

4.4.1 Inner circular groove rough milling G22/G23

Format:

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

Function: it is used for performing circular interpolations from the circle center by helical line till the programmed figure of the circle groove is machined.

Explanation:

G22: CCW inner circular groove rough milling

G23: CW inner circular groove rough milling

X、Y：The start point in X, Y plane；

Z：Machining depth, which is the absolute position in G90, and the position relative to R level in G91；

R：R reference level, which is the absolute position in G90, and the position relative to the start point of this block in G91;

I：Circular groove radius, which should be greater than the current tool radius；

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

W：First cutting depth in Z axis direction. It is the distance below the R level, which should be greater than 0 (if the first cutting depth exceeds the groove bottom, then the machining is performed at the groove bottom)；

Q：Cutting depth for each cutting feed；

V： Distance (greater than 0) to the end surface to be machined at rapid tool traverse;

D： Tool compensation number, ranging from 1～256. D0 is 0 by default. The current tool diameter value is obtained by the specified sequence number;

K：Number of repeats

Cycle process:

(1) Rapid positioning to the position in XY plane;
(2) Rapid down to point R level;
(3) Cut a depth W downward at the cutting speed by helical mode→feed to the circle center;
(4) Mill the circle surface with a radius of I helically outward from the center by an increment of L each time;
(5) Return to R reference level along Z axis;
(6) Axes X and Y rapidly position to the start point;
(7) Down to the position at which the distance to the end surface to be machined is V along Z axis;
(8) Cut a depth (Q+V) downward along Z axis;
(9) Repeat the operations (4)~(8) till the total depth of circle surface is finished;
(10) Return to initial level or point R level depending on G98 or G99.

Instruction path:

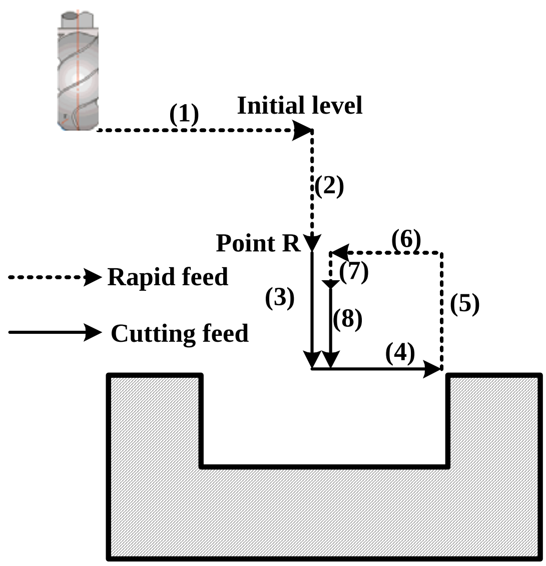


Fig. 4-4-1-1

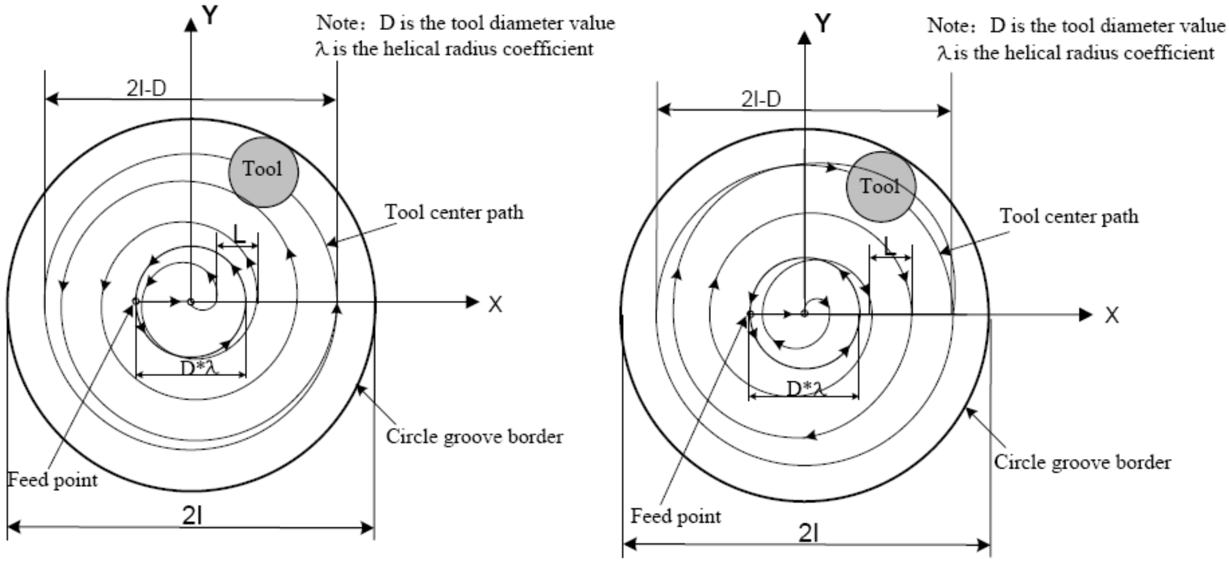


Fig. 4-4-1-2

Note:

1. It is suggested that the NO：12#1 be set to 1 when this instruction is used.
2. The helical radius coefficient in the groove cycle must be greater than 0. The coefficient is set by data parameter P269.

Example: Rough milling an inner circle groove using the canned cycle instruction G22, as shown in the figure below:

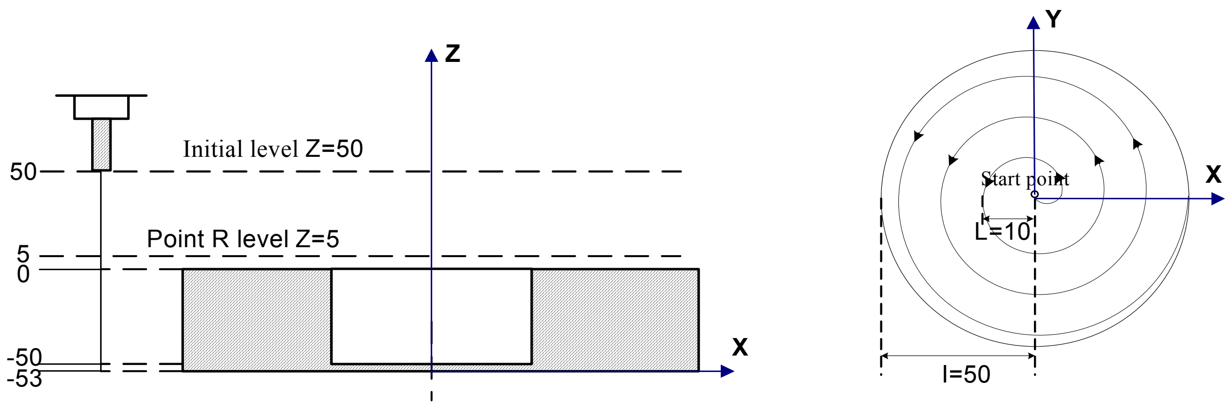


Fig. 4-4-1-3

G90 G00 X50 Y50 Z50；　（G00 Rapid positioning）

G99 G22 X25 Y25 Z-50 R5 I50 L10 W20 Q10 V10 D1 F800；（Groove rough milling within a circle）

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

M30；

Cancel：

G codes in group 01 (G00 to G03), G60 modal G code (bit parameter NO: 48#0 is set to 1) and G22/G23 cannot be specified in the same block, or G22/G23 will be cancelled.

Tool offset: The tool radius offset is ignored during the canned cycle positioning.

4.4.2 Fine milling cycle within a full circle G24/G25

Format:

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

Function: The tool fine mills a full circle within a circle by the specified radius I and the specified direction, and it returns after finishing the fine milling.

Explanation:

G24： CCW fine milling inside a circle

G25： CW fine milling inside a circle

X、Y：The start point position within X, Y plane

Z：Machining depth, which is absolute position in G90 and position relative to R reference level in G91

R：R reference level which is the absolute position in G90 and the position relative to start point of this block in G91

I： Fine milling circle radius, ranging from 0.0001mm~99999.9999mm. Its absolute value is used if it is negative;

J：Distance from fine milling start point to circle center, ranging from 0~99999.9999mm. Its absolute value is used if it is negative;

D：Tool diameter number, ranging from 1~256. D0 is 0 by default. The tool diameter value is obtained by the given number.

K：Number of repeats

Cycle process:

⑴ Rapid positioning to a location within XY plane;
⑵ Rapid down to point R level;
⑶ Feed to the machining start point at hole bottom;
⑷ To make circular interpolation by the transition arc 1 from the start point;
⑸ To make circular interpolation for the whole circle by inner arc path of finish-milling.
⑹ To make circular interpolation by transition arc 4 and return to the start point;
⑺ Return to the initial level or R level according to instruction G98 or G99.

Instruction path:

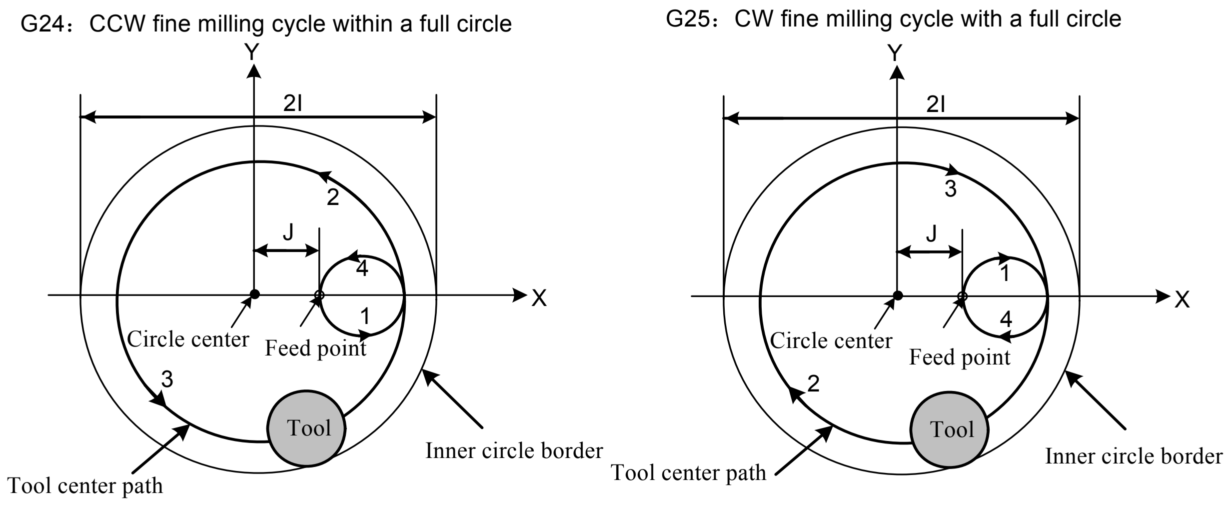


Fig. 4-4-2-1

Note：The NO: 12#1 should be set to 1 when this instruction is used.

Example: Fine milling a circular groove that has been rough milled as follows by canned cycle instruction G24:

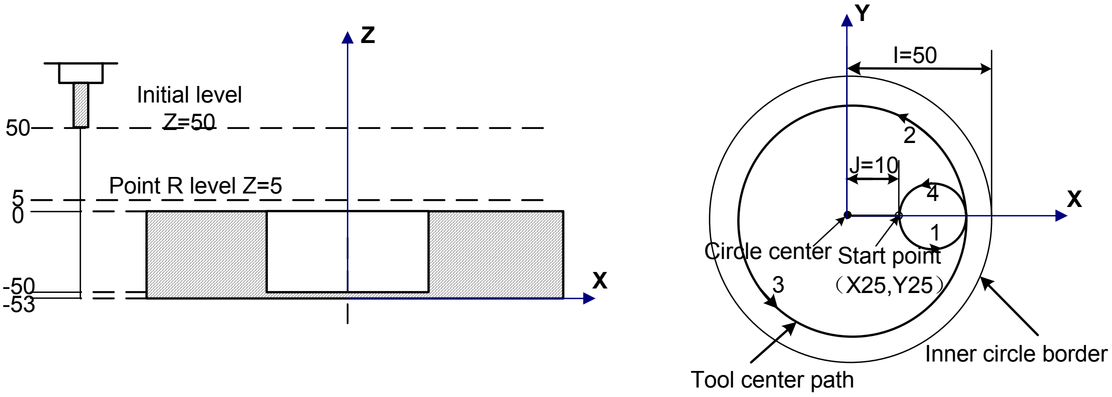


Fig. 4-4-2-2

G90 G00 X50 Y50 Z50； (G00 rapid positioning)
G99 G24 X25 Y25 Z-50 R5 I50 J10 D1 F800； (Canned cycle starts, and goes down to the bottom to perform the inner circle finish milling)
G80 X50 Y50 Z50； (To cancel canned cycle and return from R level)
M30；

Cancellation：G codes in 01 group (G00 to G03), G60 modal G code (bit parameter **NO: 48#0** is set to 1) and G24/G25 cannot be specified in the same block, or G24/G25 will be cancelled.

Tool offset：The tool radius offset is ignored during canned cycle positioning.

4.4.3 Outer circle finish milling cycle G26/G32

Format:

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

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

Explanation:

G26： CCW outer circle fine milling cycle
G32： CW outer circle fine milling cycle
X、Y：The start point within X, Y plane
Z： Machining depth, which is absolute position in G90 and position relative to R reference level in G91;
R：R reference level, which is absolute position in G90 and position relative to the start point of this block in G91;
I： Fine milling circle radius, ranging from 0.0001mm~99999.9999mm mm. Its absolute value is

used if it is a negative one;

J： Distance from the milling start point to the milling circle center, ranging from 0.0001mm~99999.9999mm. Its absolute value is used if it is a negative one;

D：Tool radius number, ranging from 0 ~256, D0 is defaulted for 0. The current tool radius value is obtained by the given number;

K：Number of repeats.

Cycle process:

⑴ Rapid positioning to a location within XY plane;
⑵ Rapid down to R level;
⑶ Feed to the hole bottom;
⑷ To make circular interpolation by the transition arc 1 from the start point;
⑸ To make circular interpolation for the whole circle by the path of arc2 and arc 3;
⑹ To make circular interpolation by transition arc 4 and return to the start point;
⑺ Return to the initial level or R level according to instruction G98 or G99.

Instruction path:

G26: CCW outer circle finish milling cycle   G32: CW outer circle finish milling cycle

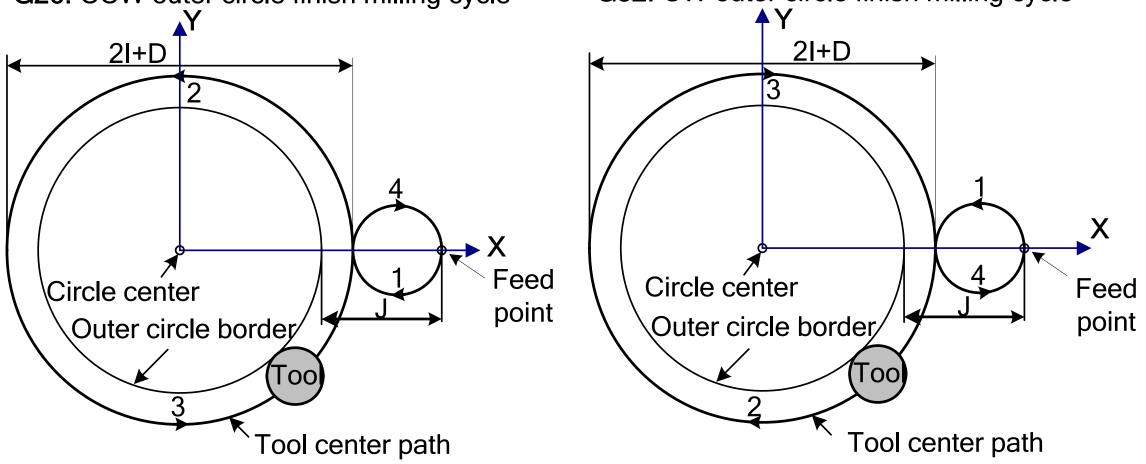


Fig. 4-4-3-1

Explanation:

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

Example: Fine milling a circular groove that has been rough milled as follows by the canned cycle instruction G26:

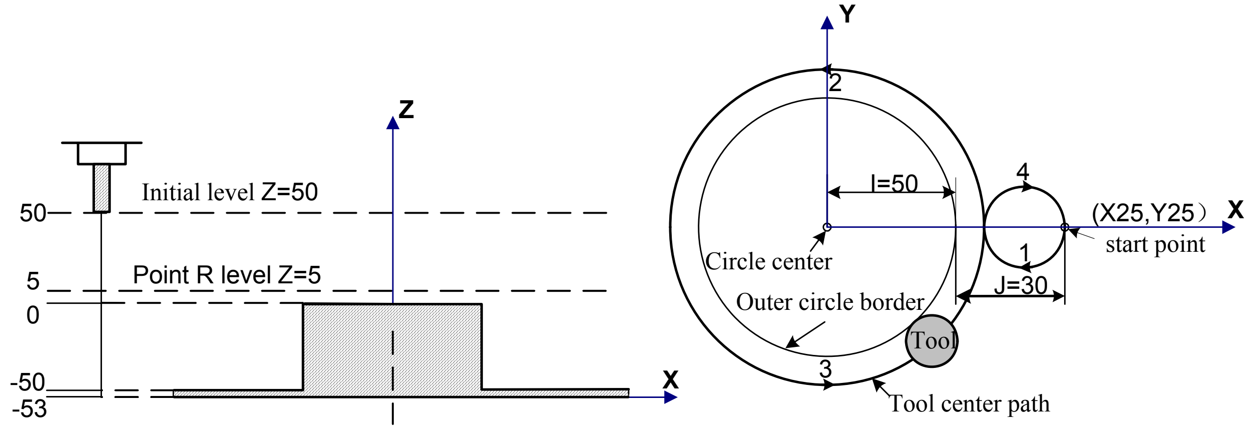


Fig. 4-4-3-2

G90 G00 X50 Y50 Z50； （G00 rapid positioning)
G99 G26 X25 Y25 Z-50 R5 I50 J30 D1 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；

Cancel：G codes in 01 group (G00 to G03), G60 modal G code (bit parameter NO: 48#0 is set to 1) and G26/G32 cannot be specified in a same block, or G26/G32 will be cancelled.
Tool offset：The tool radius offset is ignored during the canned cycle positioning.

4.4.4 Rectangular groove rough milling G33/G34

Format:

G98/G99 **G33** / **G34** **X_ Y_ Z_ R_ I_ J_ L_ W_ Q_ V_ U_ D_ F_ K_**

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

G33： CCW rectangular groove rough milling
G34： CW rectangular groove rough milling
X、Y：The start point within X, Y plane
Z：Machining depth, which is absolute position in G90 and position relative to R reference plane in G91
R：R reference plane, which is absolute position in G90 and position relative to the start point of this block in G91
I：Rectangular groove width in X axis, which should be greater than { (The setting value of data parameter **P269** ∗ tool radius)＋ tool radius} ∗ 2, and the helical feed radius should be smaller than {(I/2) - tool radius}.
J：Rectangular groove width in Y axis, which should be greater than { (The setting value of data parameter **P269** ∗ tool radius)＋ tool radius} ∗ 2, and helical feed radius should be smaller than {(J/2) - tool radius} .
L：Cutting width increment within a specified plane, which should be less than the tool diameter but greater than 0. Its absolute value is used if it is a negative one.
W：First cut depth in Z axis, which is a downward distance from R level and is greater than 0 (if

the first cut exceeds the groove bottom, it will cut at the bottom position). Its absolute value is used if it is a negative one.

Q：Cut depth of each cutting feed

V：Distance to the end surface to be machined in rapid feed, which is greater than 0. Its absolute value is used if it is negative.

U：Corner arc radius. No corner arc transition if it is omitted. The range of U is |U|, which is greater than or equal to D/2, and smaller than I/2 or J/2 whichever is smaller.

D：Tool diameter number, ranging from 1 ~ 256, D0 is 0 by default. The current tool diameter value is given by the specified number.

K：Number of repeats.

Cycle process

(1) Rapid positioning to the start point of helical feed within XY plane;
(2) Rapid down to R level;
(3) The diameter helical feed W width is obtained by radius compensation value multiplying the parameter N0. 269 value;
(4) Feed to the rectangle center;
(5) To mill a rectangular surface helically by an increment L from center outward each time;
(6) Rapid return to R level along Z axis;
(7) Rapid positioning to star point of the helical feed in XY plane;
(8) Rapid down to a position at which the distance to the end surface is V along Z axis;
(9) Z axis cuts downward for a（Q+V）depth;
(10) Repeat the actions of（4）~（8）till the rectangular surface with the total depth machined;
(11) Return to the initial level or R level according to instruction G98 or G99.

Instruction path:

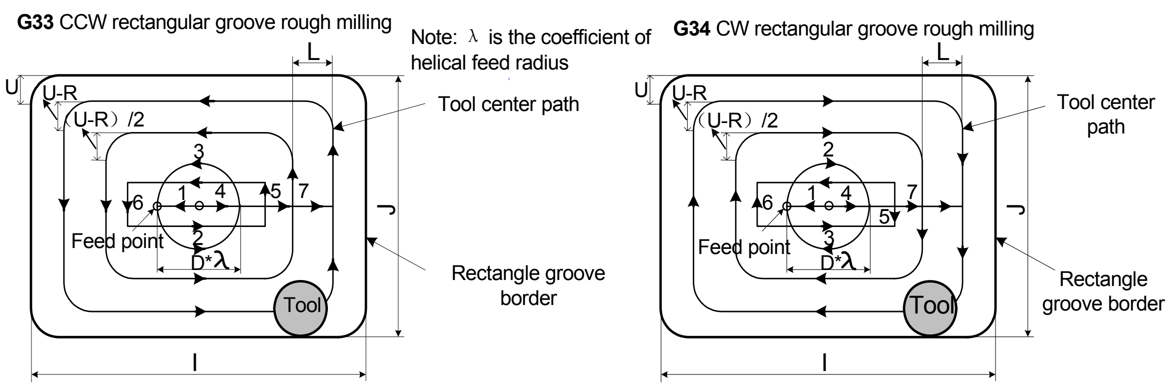


Fig. 4-4-4-1

Note: The NO:12#1 should be set to 1 when this instruction is used.

Example: Rough milling an inner rectangular groove by the canned cycle instruction G33, as shown in the following figure:

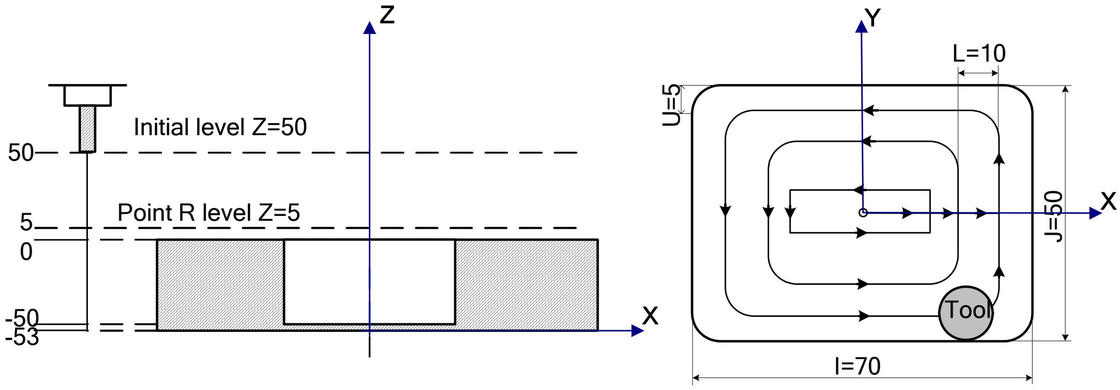


Fig. 4-4-4-2

G90 G00 X50 Y50 Z50； （G00 rapid positioning）

G99 G33 X25 Y25 Z-50 R5 I70 J50 L10 W20 Q10 V10 U5 D1 F800；

（To perform inner rectangular grove rough milling cycle）

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

M30；

Cancel：G codes in 01 group (G00 to G03), G60 modal G code (bit parameter **NO: 48#0** is set to 1) and G33/G34 cannot be specified in the same block, or G33/G34 will be cancelled.

Tool offset：The tool radius offset is ignored during the canned cycle positioning.

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 specified width and direction, and the tool returns after finishing the fine milling.

Explanation:

G35： CCW inner rectangular groove finish milling cycle.

G36： CW inner rectangular groove finish milling cycle.

X、Y：The start point within X, Y plane;

Z： Machining depth, which is absolute position in G90 and position relative to R reference plane in G91;

R： R reference plane, which is absolute position in G90 and position relative to the start point of this block in G91;

I： Rectangular width in X axis, ranging from tool diameter~99999.9999mm. Its absolute value is used if it is negative;

J： Rectangular width in Y axis, ranging from tool diameter~99999.9999mm. Its absolute value is used if it is negative;

L：Distance from milling start point to rectangular side in X axis, ranging from tool radius~99999.9999mm. Its absolute value is used if it is negative;

U：Corner arc radius. No corner transition if it is omitted. Alarm is issued if 0＜U＜tool radius;

D：Tool diameter number, ranging from 1 ~ 256, D0 is 0 by default. The current tool diameter value is given by the specified number;

K：Number of repeats.

Cycle process:

(1) Rapid positioning to the start point within XY plane;
(2) Rapid down to R level;
(3) Feed to the hole bottom;
(4) Perform circular interpolation by the path of transition arc 1 from the start point;
(5) Perform linear and circular interpolation by the path 2-3-4-5-6;
(6) perform circular interpolation by the path of transition arc 7 and return to the start point;
(7) Return to the initial level or R level according to G98 or G99 instruction.

Instruction path:

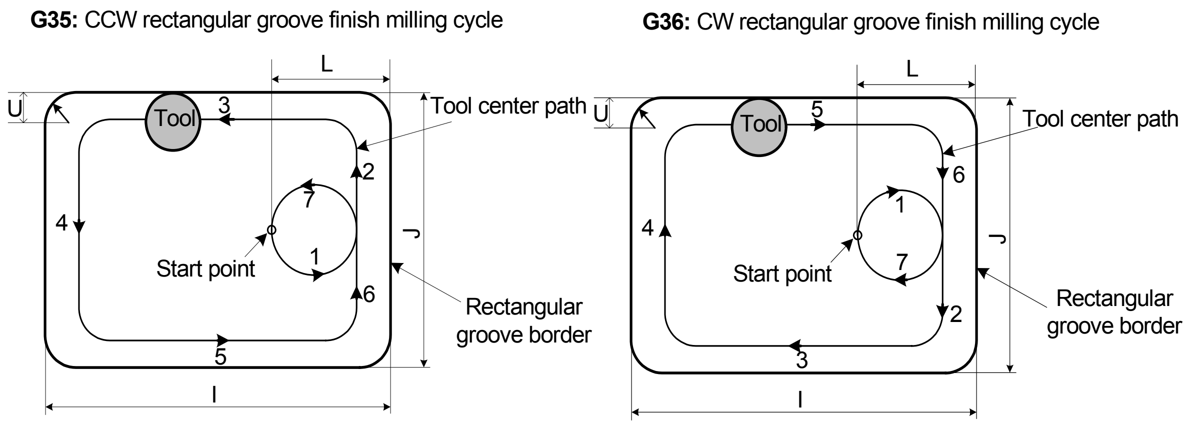


Fig. 4-4-5-1

Note: The NO:12#1 should be set to 1 when this instruction is used.

Example: Fine milling a circular groove that has been rough milled in the figure below by canned cycle G35 instruction:

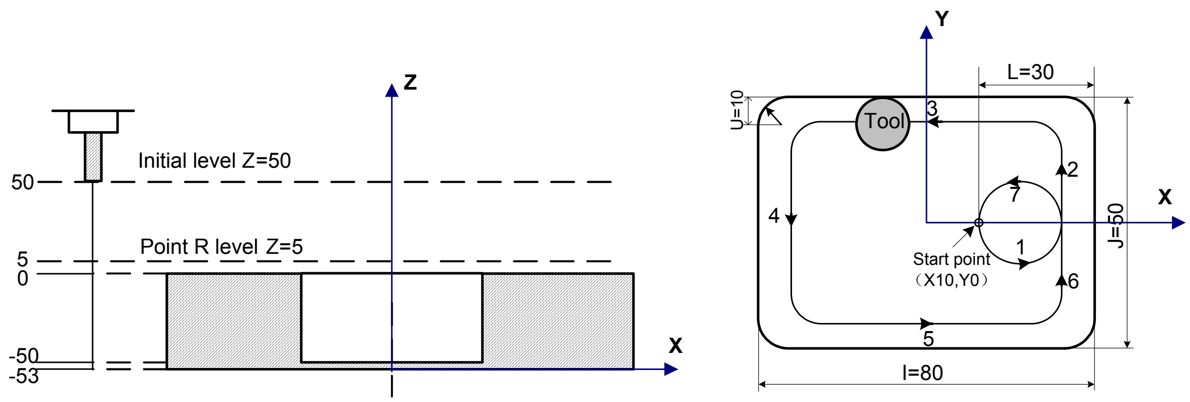


Fig. 4-4-5-2

G90 G00 X50 Y50 Z50； (G00 rapid positioning)

G99 G35 X10 Y0 Z-50 R5 I80 J50 L30 U10 D1 F800；(Performing inner rectangular groove milling at hole bottom in the canned cycle）

G80 X50 Y50 Z50；　　(Cancelling the canned cycle, and returning from point R level)

M30；

Cancel: G codes in 01 group (G00 to G03), G60 modal G code (bit parameter NO: 48#0 is set to 1) and G35/G36 cannot be specified in the same block, or G35/G36 will be cancelled.
Tool offset：The tool radius offset is ignored during the canned cycle positioning.

4.4.6 Rectangle outside fine milling cycle G37/G38

Format:

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

Function: The tool performs fine milling outside the rectangle by the specified width and direction, and then returns after finishing the fine milling.

Explanation:

G37：CCW fine milling cycle outside a rectangle.

G38：CW fine milling cycle outside a rectangle.

X、Y：The start point within X, Y plane；

Z：Machining depth，which is absolute position in G90 and position relative to R reference plane in G91;

R：R reference plane, which is absolute position in G90 and position relative to the start point of this block in G91；

I：Rectangular width in X axis, ranging from 0 mm ~99999.9999mm. Its absolute value is used if it is negative;

J：Rectangular width in Y axis, ranging from 0 mm ~99999.9999mm. Its absolute value is used if it is negative;

L：Distance from the milling start point to rectangular side in X axis, ranging from 0 mm ~99999.9999mm. Its absolute value is used if it is negative;

U： Corner arc radius. There is no corner transition arc if it is omitted;

D：Tool diameter number, ranging from 1 ~ 256, D0 is 0 by default. The current tool diameter value is given by the specified number;

K：Number of repeats.

Cycle process:

⑴ Rapid positioning to the start point within XY plane;
⑵ Rapid down to R level;
⑶ Feed to the hole bottom;
⑷ Perform circular interpolation by the path of transition arc 1 from the start point;
⑸ Perform linear and circular interpolation by the path 2-3-4-5-6
⑹; Perform circular interpolation by the path of transition arc 7 and return to the start point;
⑺ Return to the initial level or R level according to G98 or G99 instruction.

Instruction path:

G37 CCW fine milling cycle outside a rectangle G38 CW fine milling cycle outside a rectangle

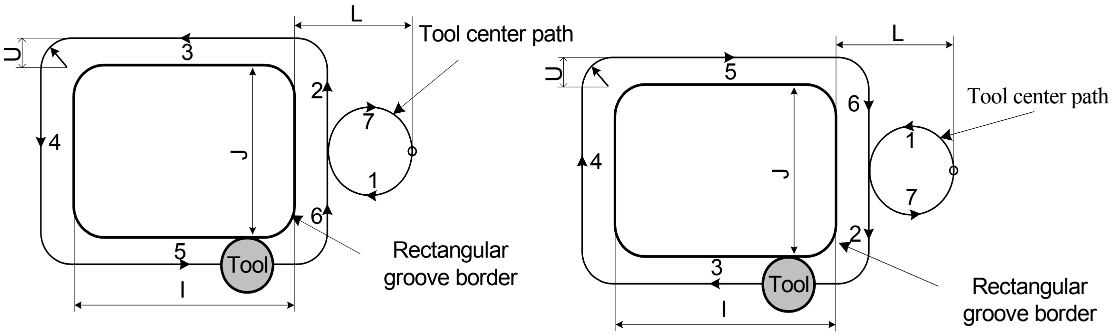


Fig. 4-4-6-1

Explanation:

For the rectangle outside fine milling, if the interpolation directions of the transition arc and fine milling arc are inconsistent, the interpolation direction in the instruction is the one of the fine milling arc.

Example: Performing fine milling outside a rectangle by the canned cycle instruction G37.

G90 G00 X50 Y50 Z50； （G00 rapid positioning）

G99 G37 X25 Y25 Z-50 R5 I80 J50 L30 U10 D1 F800；（Performing fine milling outside a rectangle at the hole bottom in the canned cycle）

G80 X50 Y50 Z50； (Cancelling the canned cycle, returning from the Point R level)

M30；

Cancel: G codes in group 01 (G00 to G03), G60 modal G code (bit parameter **NO: 48#0** is set to 1) and G37/G38 cannot be specified in the same block, or G37/G38 will be cancelled.

Tool offset：The tool radius offset is ignored during the canned cycle positioning.

4.4.7 High-speed peck drilling cycle G37

Format: G73 X_Y_Z_R_Q_F_K_

Function: The cycle is specially set for the high-speed peck drilling. It performs intermittent cutting feed to the bottom of a hole while removing chips from the hole. The operation illustration is shown as Fig. **4-4-7-1.**

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 coordinates 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 coordinates of point R;

Q_：Cut depth of each cutting feed;

F_：Cutting federate;

K_：Number of repeats.

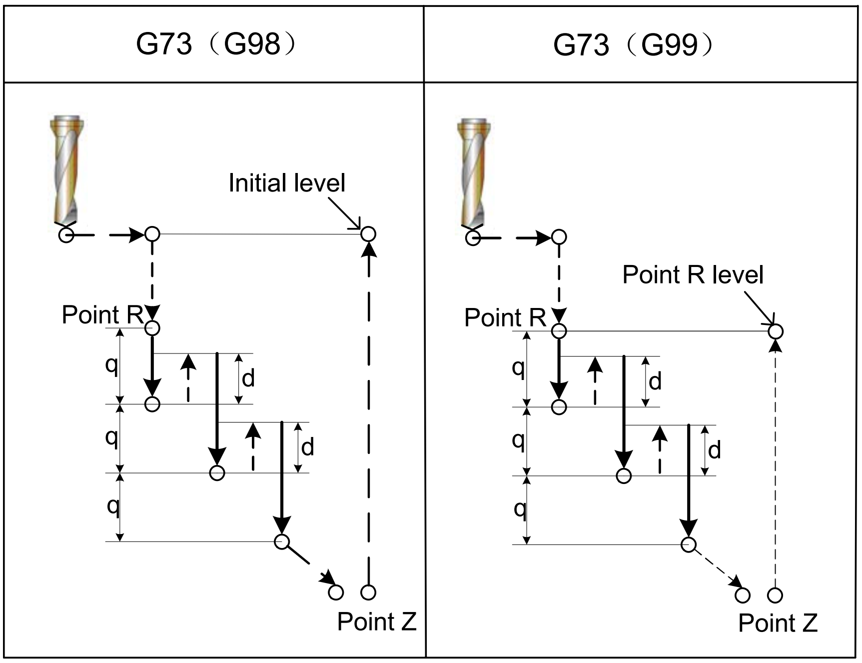


Fig. 4-4-7-1

Z, R： If either of hole bottom parameter Z and R is missing when the first drilling is being executed, the system only changes the mode, with no Z axis action executed.

Q： If parameter Q is specified, the intermittent feed shown in the figure above is performed. Here, the system retracts the tool by the retraction d (Fig.4-4-1-1) specified by data parameter **p270**, and the tool performs rapid retraction for distance d intermittently each feeding.

If G73 and an M code are specified in the same block, the M code is executed at the time of the 1st hole positioning operation, then the system proceeds to the next drilling operation.

If the number of repeats K is specified, M code is only executed for the first hole, not for the other holes.

Note 1: In the current version, M00, M01, M02, M06, M30, M98 and M99 are the M codes executed after the other instructions in a block, i.e. these M codes are executed after the execution of the current statement block.

Note 2: When the bit parameter NO：43# 1=0, no alarm will be issued if there is no cut-in value specified in the peck drilling (G73，G83). At this moment, if the instruction parameter Q is not specified or it is 0, the system performs the hole positioning in XY plane, but does not perform the drilling operation. When the bit parameter NO：43#1=1, an alarm will be issued if no cut-in value is specified in the peck drilling (G73，G83), i.e., an alarm "0045：Address Q not found or set to 0（G73/G83）" occurs when the instruction parameter Q is not specified or it is 0. If the Q value is negative, the system takes its absolute value to perform intermittent feed.

Note 3: Tool length compensation: If the tool length compensation instruction G43, G44 or G49 is specified in the same block with a canned cycle instruction, the offset is added or cancelled when the tool is positioned to point R; If the tool compensation instruction G43, G44 or G49 is specified in a separate block in the canned cycle mode, the system can add or cancel the offset in real time.

Cancel: G codes in 01 group (G00 to G03), G60 modal G code (bit parameter NO: 48#0 is set to 1) and G73 cannot be specified in the same block, otherwise G73 will be cancelled.

Tool offset: The tool radius offset is ignored during the canned cycle positioning.

Example:

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

Note: In the example above, the chip removal operation is still performed though Q is omitted during the machining for the holes 2 to 6.

4.4.8 Drilling cycle, spot drilling cycle G81

Format: G81 X_ Y_ Z_ R_ F_ K_

Function: This cycle is used for normal drilling. Cutting feed is performed to the bottom of the hole, and then the tool is retracted from the bottom in rapid traverse.

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 coordinates 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 coordinates of point R level.

F_: Cutting feedrate

K_: Number of repeats (if needed)

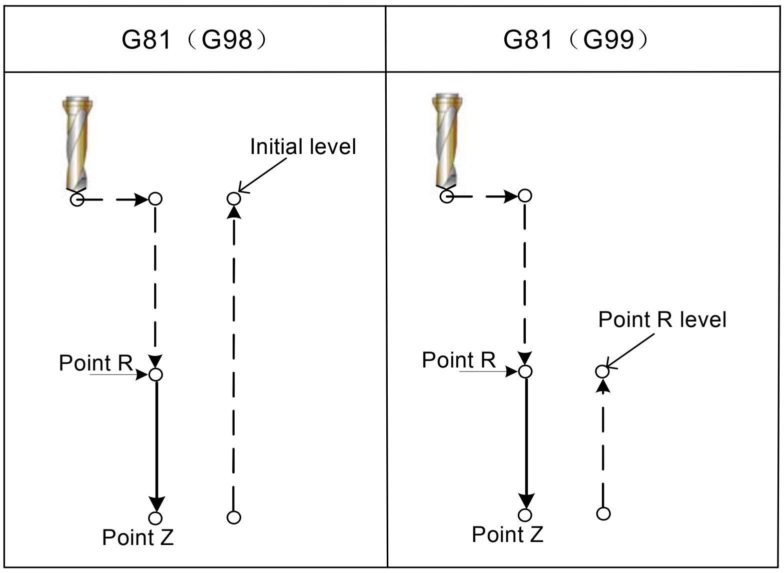


Fig. 4-4-8-1

Z, R：If either of hole bottom parameter Z and R is missing when the first drilling is executed, the system only changes the mode, with no Z axis action executed. After positioning along X axis and Y axis, rapid traverse is performed to point R. Drilling from point R to point Z is performed, the tool is then retracted in the rapid traverse. Miscellaneous function M codes are used to rotate the spindle before G81 is specified.

When G81 and an M code are specified in the same block, the M code is executed at the time of the first hole positioning, the system then proceeds to the next drilling operation.

When the number of repeats K is specified, the M code is only performed for the first hole. For the other holes, it is not performed.

Note: In the current version, M00, M01, M02, M06, M30, M98 and M99 are the M codes executed after the other instructions in a block, i.e. these M codes are executed after the execution of the current statement block.

Tool length compensation: If the tool length compensation instruction G43, G44 or G49 is specified in the same block with a canned cycle instructioin, the offset is added or cancelled at the time of positioning to point R level; If the tool compensation instruction G43, G44 or G49 is specified in a separate block in the canned cycle mode, the system can add or cancel the offset in real time.

Example:

M3 S2000	Spindle starts to rotate
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, drilling 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；	

Cancel: G codes in 01 group (G00 to G03), G60 modal G code (bit parameter **NO: 48#0** is set to 1) and G81 cannot be specified in the same block,

Tool offset：The tool radius offset is ignored at the time of the canned cycle positioning.

4.4.9 Drilling cycle, counterboring cycle G82

Format: G82 X_ Y_ Z_ R_ P_ F_ K_；

Function: This cycle is used for normal drilling. Cutting feed is performed to the bottom of the hole. At the bottom, a dwell is performed, and the tool is then retracted from the bottom of the hole in rapid traverse.

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 coordinates 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 coordinates of point R;

F_：Cutting federate;

P_：The minimum dwell time at the hole bottom, with its absolute value used if it is negative;

K_：Number of repeats.

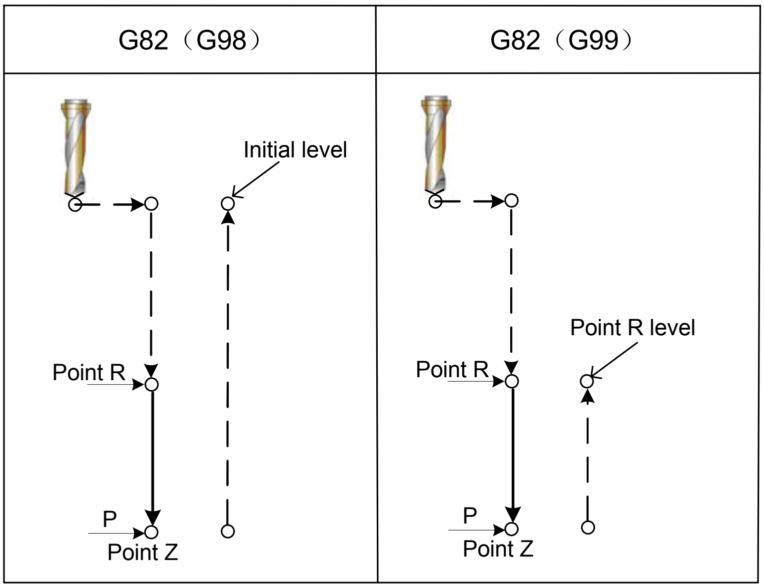


Fig. 4-4-9-1

After positioning along axes X and Y, rapid traverse is performed to point R, and drilling is then performed from point R to point Z. When the tool reaches the bottom of the hole, a dwell is performed and the tool is then retracted in rapid traverse.

Miscellaneous function M codes are used to rotate the spindle before G82 is specified.

When G82 and an M code are specified in the same block, the M code is executed at the time of the first hole positioning, and the system then proceeds to the next drilling operation.

When the number of repeats K is specified, the M code is only executed for the first hole. It is not executed for the other holes.

Note: In the current version, M00, M01, M02, M06, M30, M98 and M99 are the M codes executed after the other instructions in a block, i.e. these M codes are executed after the execution of the current statement block.

Tool length compensation: If the tool length compensation instruction G43, G44 or G49 is specified in the same block with the canned cycle instruction, the offset is added or cancelled at the time of positioning to point R level; If the tool compensation instruction G43, G44 or G49 is specified in a separate block in the canned cycle mode, the system can add or cancel the offset in real time.

P is a modal instruction, with its min. value set by data parameter P281 and its max. value by P282. If P value is less than the value set by P281, the min. value takes effect; if P value is more than the value set by P282, the max. value takes effect. P cannot be stored as modal data f it is specified in a block that does not perform drilling.

Example:

M3 S2000　Spindle starts to rotate

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

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

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

X1000.； Positioning, drill hole 4, dwell for 1s at the hole bottom, then return to point R
Y-550； Positioning, drill hole 5, dwell for 1s at the hole bottom, then return to point R
G98 Y-750； Positioning, drill hole 6, dwell for 1s at the hole bottom, then return to initial level
G80； Cancel the canned cycle
G28 G91 X0 Y0 Z0 ； Return to the reference point
M5； Spindle stops
M30；

Cancel: G codes in 01 group (G00 to G03), G60 modal G code (bit parameter NO: 48#0 is set to 1) and G82 cannot be specified in the same block, otherwise G82 will be cancelled.
Tool offset：The tool radius offset is ignored at the time of the canned cycle positioning.

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. It performs intermittent cutting feed to the bottom of the hole while removing the chips from the 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 coordinates 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 coordinates of point R;
Q_：Cut depth for each cutting feed;
F_：Cutting federate;
K_：Number of repeats.

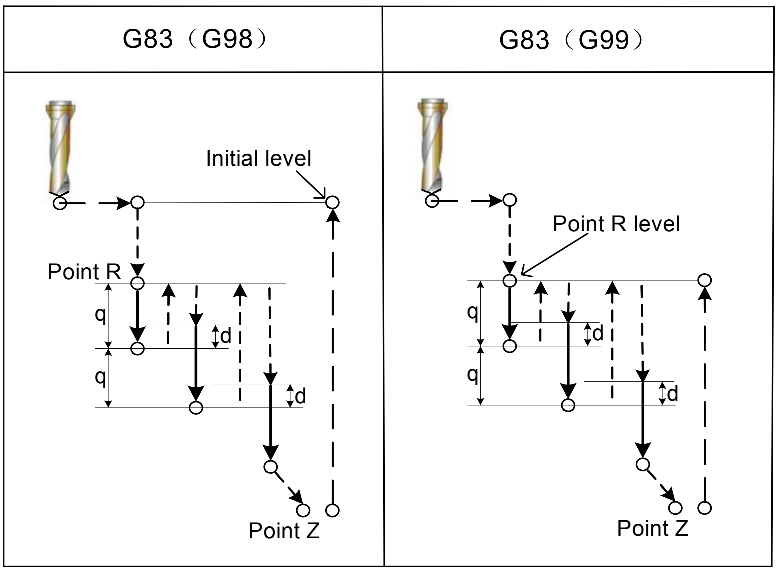


Fig. 4-4-10-1

Q：It specifies the cut depth for each cutting feed, which must be specified as an incremental value. In the second and the subsequent cutting feed, the tool rapidly traverses to the position which has a distance d to the end position of the last drilling and then performs the cutting feed again. d is set by parameter P271, as is shown in Fig. 4-4-10-1.

Specify a positive value for Q, and a negative one will be processed as its absolute value.

Specify Q in a drilling block. If it is specified in the block containing no drilling, it is stored as modal data.

Miscellaneous function M codes are used to rotate the spindle before G83 is specified.

When G83 and an M code are specified in the same block, the M code is executed at the time of the first hole positioning, and the system then proceeds to the next drilling operation.

When the number of repeats K is specified, the M code is only executed for the first hole, but not for the other holes.

Note 1: In the current version, M00, M01, M02, M06, M30, M98 and M99 are the M codes executed after the other instructions in a block, i.e. these M codes are executed after the execution of the current statement block.

Note 2: When the bit parameter NO：43# 1=0, no alarm will be issued if there is no cut-in value specified in the peck drilling (G73，G83). At this moment, if the instruction parameter Q is not specified or it is 0, the system performs the hole positioning in XY plane, but it does not perform the drilling operation. When the bit parameter NO：43#1=1, an alarm will be issued if no cut-in value is specified in the peck drilling (G73，G83), i.e. an alarm "0045：Address Q not found or set to 0（G73/G83）" occurs when the instruction parameter Q is not specified or it is 0. If the Q value is negative, the system uses its absolute value to perform intermittent feeding.

Tool length compensation: If the tool length compensation instruction G43, G44 or G49 is specified in the same block with the canned cycle instruction, the offset is added or cancelled at the time of positioning to point R level; If the tool compensation instruction G43, G44 or G49 is specified in a separate block in the canned cycle mode, the system can add or cancel the offset in real time.

Example:

M3 S2000；	Spindle starts to rotate
G90 G99 G83 X300 Y-250 Z-150 R-100 Q15 F120；	Positioning, drill hole 1, then return to point R
Y-550；	Positioning, drill hole 2, then return to point R
Y-750；	Positioning, drill hole 3, then return to point R
X1000；	Positioning, drill hole 4, then return to point R
Y-550；	Positioning, drill hole 5, then return to point R
G98 Y-750；	Positioning, drill hole 6, then return to initial level
G80；	
G28 G91 X0 Y0 Z0 ；	Return to the reference point
M5；	Spindle stops
M30；	

Cancel：G codes in 01 group (G00 to G03), G60 modal G code (bit parameter NO: 48#0 is set to 1) and G83 cannot be specified in the same block, otherwise G83 will be cancelled.

Tool offset：The tool radius offset is ignored at the time of the canned cycle positioning.

4.4.11 Right-hand taping cycle G84

Format: G84 X_ Y_ Z_ R_ P_ F_

Function: This cycle performs taping. The spindle is rotated in the reverse direction when the bottom of the hole is reached in this taping 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 coordinates 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 coordinates of point R.

P_: Minimum Dwell time at the hole bottom. The absolute value is used if it is a negative one.

F_: Cutting feedrate.

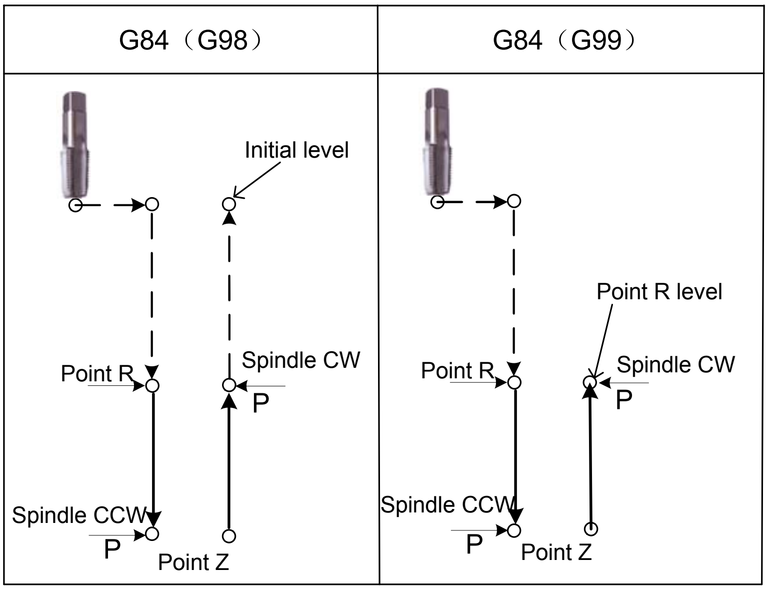


Fig. 4-4-11-1

Taping is performed by rotating the spindle counterclockwise. When the bottom of the hole is reached, the spindle is rotated in the reverse direction for retraction. This operation creates threads.

Feedrate override and spindle override 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 CCW rotation is not specified, the system will adjust the rotation to the CCW rotation automatically in R level by the current specified spindle speed.

If G84 and M code are specified in the same block, the M code is executed at the time of the 1st hole positioning operation, then the system proceeds to the next drilling operation.

When the number of repeats K is specified, the M code is only executed for the first hole, not for the other holes.

Note: In the current version, M00, M01, M02, M06, M30, M98 and M99 are the M codes executed after the other instructions in a block, i.e. these M codes are executed after the execution of the current statement block.

P is a modal instruction, with its min. value set by number parameter P281 and max. value by P282. If P value is less than the value set by P281, the min. value takes effect; if P value is more than the value set by P282, the max. value takes effect. P cannot be stored as modal data if it is specified in a block that does not perform drilling.

Tool length compensation: If the tool length compensation instruction G43, G44 or G49 is specified in the same block with the canned cycle instruction, the offset is added or cancelled at the time of positioning to point R level; If the tool compensation instruction G43, G44 or G49 is specified in a separate block in the canned cycle mode, the system can add or cancel the offset in real time.

In feeding per minute, the relationship among thread lead and feedrate as well as spindle speed is as follows:

Feedrate F=tap pitch×spindle speed S
For example: for the M12×1.5 thread hole on the workpiece, the following parameters can be used:
S500=500 r /min F=1.5×500=750mm/min
For multiple thread, F value can be obtained by multiplying the thread number.

Example:

M29 S100 ；	Spindle starts to rotate
G90 G99 G74 X300 Y-250 Z-150 R-120 P300 F120；	Positioning, drill hole 1, then return to point R
Y-550；	Positioning, drill hole 2, then return to point R
Y-750；	Positioning, drill hole 3, then return to point R
X1000；	Positioning, drill hole 4, then return to point R
Y-550；	Positioning, drill hole 5, then return to point R
G98 Y-750；	Positioning, drill hole 6, then return to initial level
G80；	
G28 G91 X0 Y0 Z0 ；	Return to the reference point
M5；	Spindle stops
M30；	

Cancel: G codes in 01 group (G00 to G03), G60 modal G code (bit parameter NO: 48#0 is set to 1) and G84 cannot be specified in the same block, otherwise G84 will be cancelled.

Tool offset：The tool radius offset is ignored at the time of the canned cycle positioning.

4.4.12 Left-hand taping cycle G74

Format: G74 X_ Y_ Z_ R_ P_ F_

Function: This cycle performs taping. In this cycle, the spindle is rotated in the reverse direction when the bottom of the hole is reached.

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 coordinates 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 coordinates of point R.
P_: Minimum dwell time at the hole bottom. The absolute value is used if it is a negative one.
F_: Cutting feedrate.

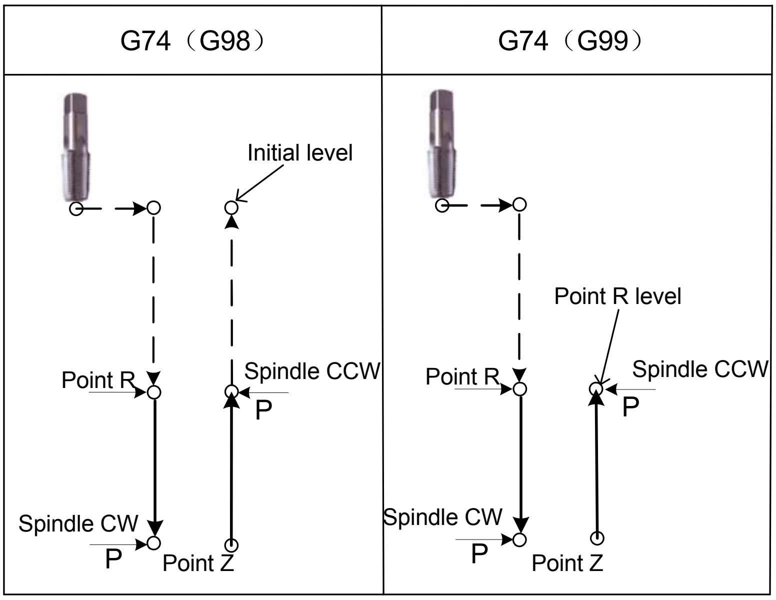


Fig. 4-4-12-1

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 override and spindle override 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 CW rotation is not specified, the system will adjust itself to CW rotation in R level automatically by the current specified spindle speed.

If G74 and an M code are specified in the same block, the M code is executed at the time of the 1st hole positioning operation, then the system proceeds to next drilling operation.

When the number of repeats K is specified, the M code is only executed for the first hole. It is not executed for the other holes.

Note: In the current version, M00, M01, M02, M06, M30, M98 and M99 are the M codes executed after the other instructions in a block, i.e. these M codes are executed after the execution of the current statement block.

P is a modal instruction, with its min. value set by number parameter P281 and max. value by P282. If P value is less than the value set by P281, the min. value takes effect; if P value is more than the value set by P282, the max. value takes effect. P cannot be stored as modal data if it is specified in a block that does not perform drilling.

Tool length compensation: If the tool length compensation instruction G43, G44 or G49 is specified in the same block with canned cycle, the offset is added or cancelled at the time of positioning to point R level; If the tool compensation instruction G43, G44 or G49 is specified in a separate block in the canned cycle mode, the system can add or cancel the offset in real time.

Example：

M04 S100；	Spindle starts to rotate
G90 G99 G74 X300 Y-250 Z-150 R-120 P300 F120；	Positioning, drill hole 1, then return to point R
Y-550；	Positioning, drill hole 2, then return to point R
Y-750；	Positioning, drill hole 3, then return to point R
X1000；	Positioning, drill hole 4, then return to point R
Y-550；	Positioning, drill hole 5, then return to point R

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

Cancel: G codes in 01 group (G00 to G03), G60 modal G code (bit parameter NO: 48#0 is set to 1) and G74 cannot be specified in the same block, otherwise G74 will be cancelled.
Tool offset：The tool radius offset is ignored at the time of the canned cycle positioning.

4.4.13 Fine boring cycle G76

Format: G76 X_Y_Z_Q_R_P_F_K_
Function: This cycle is used for boring a hole precisely.
When the tool reaches the hole bottom, the spindle stops and the tool is moved away from the machined surface of the workpiece and retracted.
Prevent the retraction trail from affecting the machined surface smoothness and avoid the tool damage 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 coordinates 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 coordinates of point R level.
Q_: Offset at the hole bottom
P_: Dwell time at the hole bottom, with its absolute value used if it is negative.
F_: Cutting feedrate.
K_: Number of fine boring repeats

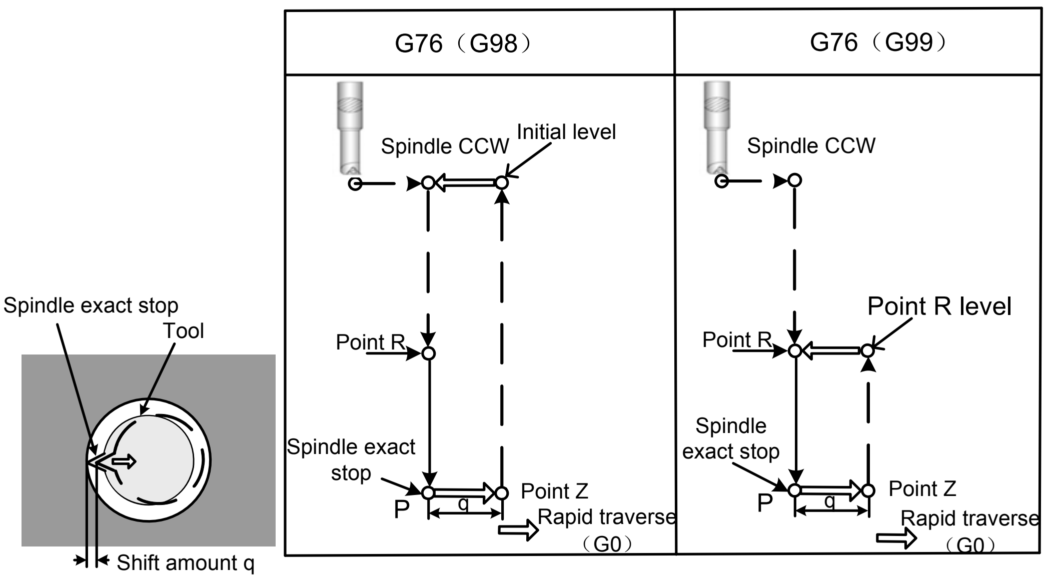


Fig. 4-4-13-1

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

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

If G76 and an M code are specified in the same block, the M code is executed at the time of the 1st hole positioning operation, then the system proceeds to the next boring operation.

If the number of repeats K is specified, the M code is only executed for the 1st hole, for the other holes, the M code is not executed.

Note: In the current version, M00, M01, M02, M06, M30, M98 and M99 are the M codes executed after the other instructions in a block, i.e. these M codes are executed after the execution of the current statement block.

Tool length compensation: If the tool length compensation instruction G43, G44 or G49 is specified in the same block with the canned cycle instruction, the offset is added or cancelled at the time of positioning to point R level; If the tool compensation instruction G43, G44 or G49 is specified in a separate block in the canned cycle mode, the system can add or cancel the offset in real time.

Axis switching: The canned cycle must be canceled before the drilling axis is changed.

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

Example:

```
M3 S500；                      Spindle starts to rotate
G90 G99 G76 X300 Y-250；       Positioning, bore hole 1, then return to point R
  Z-150 R-100 Q5 ；            Orient at the bottom of the hole, then shift by 5mm
P1000 F120；                   Stop at the bottom of the hole for 1s
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 the reference point
M5；                           Spindle stops
```

Cancel: G codes in 01 group (G00 to G03), G60 modal G code (bit parameter NO: 48#0 is set to 1) and G76 cannot be specified in the same block, otherwise G76 will be cancelled.

Tool offset：The tool radius offset is ignored at the time of the canned cycle positioning.

4.4.14 Boring cycle G85

Format: G85 X_ Y_ Z_ R_ F_ K_

Function: This cycle is used for boring 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 coordinates 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 coordinates of point R.

F_： Cutting feedrate.

K_： Number of repeats

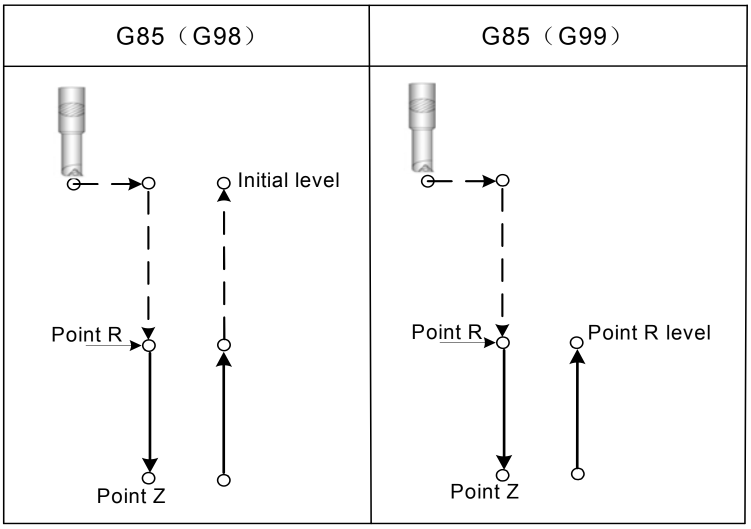


Fig. 4-4-14-1

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

Use a miscellaneous function (M code) to rotate the spindle before specifying G85.

If G85 and an M code are specified in the same block, the M code is executed at the time of the 1st hole positioning operation, then the system proceeds to the next boring operation.

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

Note: In the current version, M00, M01, M02, M06, M30, M98 and M99 are the M codes executed after the other instructions in a block, i.e. these M codes are executed after the execution of the current statement block.

Tool length compensation: If the tool length compensation instruction G43, G44 or G49 is specified in the same block with the canned cycle instruction, the offset is added or cancelled at the time of positioning to point R level; If the tool compensation instruction G43, G44 or G49 is specified in a separate block in the canned cycle mode, the system can add or cancel the offset in real time.

Axis switching: The canned cycle must be cancelled before the drilling axis is changed.

Boring: Boring is not performed in a block which does not contain X, Y, Z or other axes.

Example:

M3 S100 ；	The spindle starts to rotate
G90 G99 G85 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
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 the reference point
M5；	Spindle stops
M30；	

Cancel: G codes in 01 group (G00 to G03), G60 modal G code (bit parameter NO: 48#0 is set to 1) and G85 cannot be specified in a same block, otherwise G85 will be cancelled.
Tool offset：The tool radius offset is ignored at the time of the canned cycle positioning.

4.4.15 Boring cycle G86

Format: G86 X_ Y_ Z_ R_ F_ K_；
Function: This cycle instruction 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 coordinates 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 coordinates of point R;
F_： Cutting federate;
K_： Number of repeats.

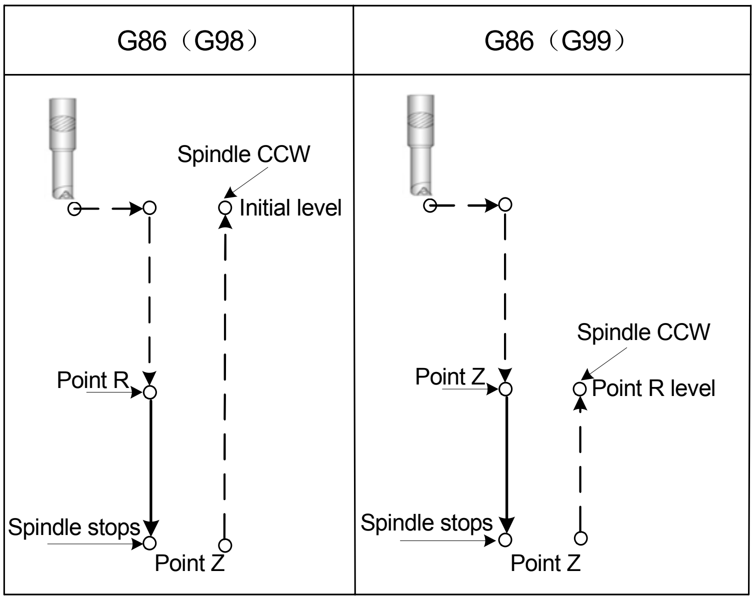


Fig. 4-4-15-1

After positioning along X and Y axes, rapid traverse is performed to point R. And boring is performed from point R to point Z. When the spindle stops at the bottom of the hole, the tool is retracted in rapid traverse.

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

If G86 and an M code are specified in the same block, the M code is executed at the time of the 1st hole positioning operation, then the system proceeds to the next boring operation.

Note: In the current version, M00, M01, M02, M06, M30, M98 and M99 are the M codes executed after the other instructions in a block, i.e. these M codes are executed after the execution of the current statement block.

Tool length compensation: If the tool length compensation instruction G43, G44 or G49 is specified in the same block with the canned cycle instruction, the offset is added or cancelled at the time of positioning to point R level; If the tool compensation instruction G43, G44 or G49 is specified in a separate block in the canned cycle mode, the system can add or cancel the offset in real time.

Axis switching: The canned cycle must be cancelled before the drilling axis is changed.

Boring: Boring is not performed in a block which does not contain X, Y, Z or other axes.

Example:

M3 S2000； Spindle starts to rotate

G90 G99 G86 X300 Y-250 Z-150 R-100 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 the reference point

M5； Spindle stops

M30；

Cancel: G codes in 01 group (G00 to G03), G60 modal G code (bit parameter NO: 48#0 is set to 1) and G86 cannot be specified in the same block, otherwise G86 will be cancelled.

Tool offset：The tool radius offset is ignored at the time of the canned cycle positioning.

4.4.16 Boring cycle, back boring cycle G87

Format: G87 X_Y_Z_R_Q_P_ F_；

Function: This cycle performs accurate boring.

Explanation:

X_Y_: Hole positioning data

Z_: In incremental programming it specifies the distance from point R level to point Z level; in absolute programming it specifies the absolute coordinates of the point Z level.

R_: In incremental programming it specifies the distance from the initial level to point R level; in absolute programming it specifies the absolute coordinates of point R level (hole bottom).

Q_: Shift amount at the bottom of the hole

P_: Minimum dwell time at the hole bottom, with its absolute value used if it is negative.

F_: Cutting feedrate

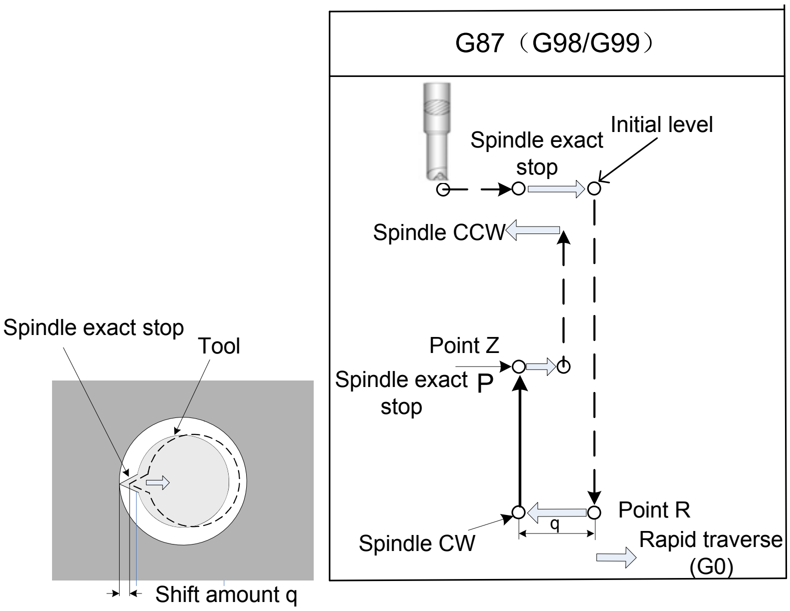


Fig. 4-4-16-1

After positioning along X and Y axes, the tool is stopped after spindle orientation. Then the tool is moved in the direction opposite to the tool nose, and positioning (rapid traverse) is performed to the hole bottom (point R). The tool is then shifted in the direction of the tool nose and the spindle is rotated counterclockwise. 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 in the direction opposite to the tool nose, then it is returned to the initial level. The tool is then shifted in the direction of the tool nose and the spindle is rotated counterclockwise to proceed to the next block operation.

The parameter Q specifies the retraction distance. The retraction direction and retraction axis are set by system parameter NO:42#4 and NO:42#5 respectively. Q must be a positive value, if it is specified with a negative value, the negative sign is ignored. The hole bottom shift amount of Q is a modal value retained in the canned cycle, which must be specified carefully because it is also used as the cutting depth for G73 and G83.

Before specifying G87, use a miscellaneous function (M code) to rotate the spindle.

If G87 and an M code are specified in the same block, the M code is executed at the time of the 1st hole positioning operation, then the system proceeds to the next boring operation.

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

Note: In the current version, M00, M01, M02, M06, M30, M98 and M99 are the M codes executed after the other instructions in a block, i.e. these M codes are executed after the execution of the current statement block.

Tool length compensation: If the tool length compensation instruction G43, G44 or G49 is specified in the block with the canned cycle instruction, the offset is added or cancelled at the time of positioning to point R level; If the tool compensation instruction G43, G44 or G49 is specified in a separate block in the canned cycle mode, the system can add or cancel the offset in real time.

The canned cycle can only be executed in G17 plane.

Boring: In a block which contains no X, Y, Z or other additional axes, boring is not performed.

Note: The values of Z and R must be specified when the back boring cycle is being programmed. In general, point Z is located above point R, otherwise an alarm occurs.

Example:

```
M3 S500;                    Spindle starts to rotate
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 the reference point
M5;                         Spindle stops
```

Cancel: G codes in 01 group (G00 to G03), G60 modal G code (bit parameter NO: 48#0 is set to 1) and G87 cannot be specified in the same block, otherwise G87 will be cancelled.

Tool offset: The tool radius offset is ignored at the time of the canned cycle positioning.

4.4.17 Boring cycle G88

Format: G88 X_Y_Z_R_ P_F_

Function: This cycle is use for boring 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 coordinates 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 coordinates of point R.

P_: Dwell time at the bottom of the hole, with its absolute value used if it is negative.

F_: Cutting feedrate

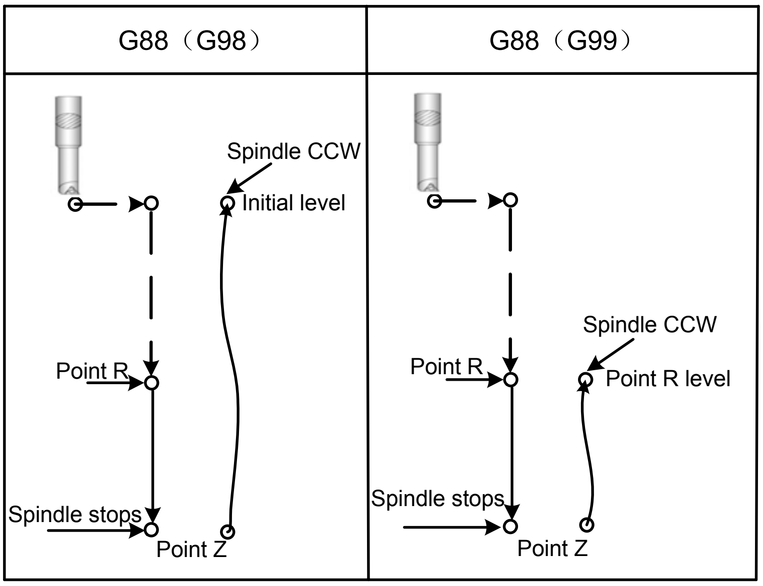


Fig. 4-4-17-1

After positioning along X and Y axes, rapid traverse is performed to point R. Boring is performed from point R to point Z. When boring is completed, a dwell is performed then the spindle is stopped. The tool is manually retracted from point Z at the hole bottom to point R (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 an M code are specified in the same block, the M code is executed at the time of the 1st hole positioning operation, then the system proceeds to the next drilling operation.

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

Note: In the current version, M00, M01, M02, M06, M30, M98 and M99 are the M codes executed after the other instructions in a block, i.e. these M codes are executed after the execution of the current statement block.

P is a modal instruction, with its min. value set by data parameter P281 and max. value by P282. If P value is less than the value set by P281, the min. value takes effect; if P value is more than the value set by P282, the max. value takes effect. P cannot be stored as modal data if it is specified in a block that does not perform drilling.

Tool length compensation: If the tool length compensation instruction G43, G44 or G49 is specified in the same block with the canned cycle instruction, the offset is added or cancelled at the time of positioning to point R level; If the tool compensation instruction G43, G44 or G49 is specified in a separate block in the canned cycle mode, the system can add or cancel the offset in real time.

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

Boring: In a block which contains no X, Y, Z or other additional axes, boring is not performed.

Example:

M3 S2000	Spindle starts to rotate
G90 G99 G88 X300. Y-250. Z-150. R-100. P1000 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 the reference point
M5;	Spindle stops

Cancel: G codes in 01 group (G00 to G03), G60 modal G code (bit parameter NO: 48#0 is set to 1) and G88 cannot be specified in the same block, otherwise G88 will be cancelled.
Tool offset：The tool radius offset is ignored at the time of the canned cycle positioning.

4.4.18 Boring cycle G89

Format: G89 X_ Y_ Z_ R_ P_ F_ K_
Function: This cycle is used for boring 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 coordinates 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 coordinates of point R.
P_： Minimum dwell time at the bottom of the hole, with its absolute value used if it is negative.
F_： Cutting feedrate.
K_： Number of repeats

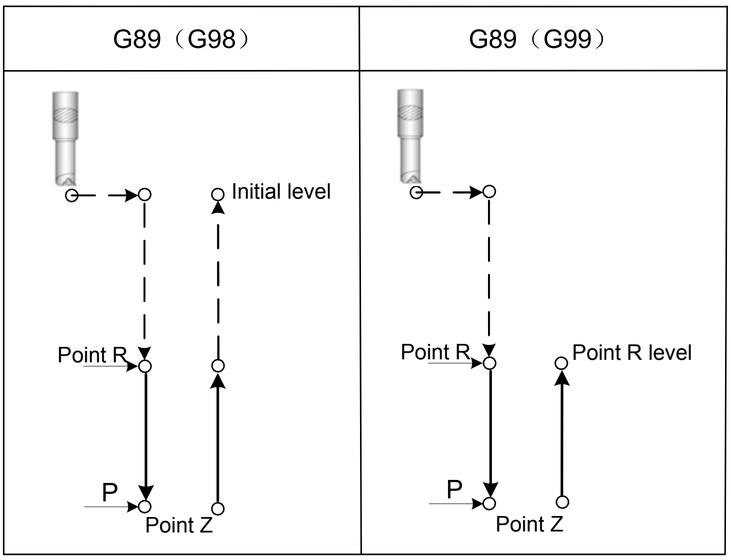


Fig. 4-4-18-1

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

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

If G89 and an M code are specified in the same block, the M code is executed while the 1st hole positioning operation, then the system proceeds to the next drilling operation.

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

Note: In the current version, M00, M01, M02, M06, M30, M98 and M99 are the M codes executed after the other instructions in a block, i.e. these M codes are executed after the execution of the current statement block.

P is a modal instruction, with its min. value set by data parameter P281 and max. value by P282. If P value is less than the value set by P281, the min. value takes effect; if P value is more than the value set by P282, the max. value takes effect. P cannot be stored as modal data if it is specified in a block that does not perform drilling.

Tool length compensation: If the tool length compensation instruction G43, G44 or G49 is specified in the same block with the canned cycle instruction, the offset is added or cancelled at the time of positioning to point R level; If the tool compensation instruction G43, G44 or G49 is specified in a separate block in the canned cycle mode, the system can add or cancel the offset in real time.

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 starts to rotate
G90 G99 G89 X300. Y-250. Z-150. R-120. P1000 F120.
Positioning, bore hole 1, return to point R level, then 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 the reference point
M5； Spindle stops
M30；

Cancel: G codes in 01 group (G00 to G03), G60 modal G code (bit parameter NO: 48#0 is set to 1) and G89 cannot be specified in the same block, otherwise G89 will be cancelled.

Tool offset：The tool radius offset is ignored at the time of the canned cycle positioning.

4.4.19 Left-hand rigid taping G74

Format: G74 X_Y_Z_R_P_F_K_

Function: In the rigid taping, the spindle motor is controlled as it were a servo motor. This instruction is used for left-hand high-speed and high-precision taping.

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 coordinates 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 coordinates of point R.

P_： Dwell time at the bottom of the hole, with its absolute value used if it is negative.

F_： Cutting feedrate.

K_： Number of repeats

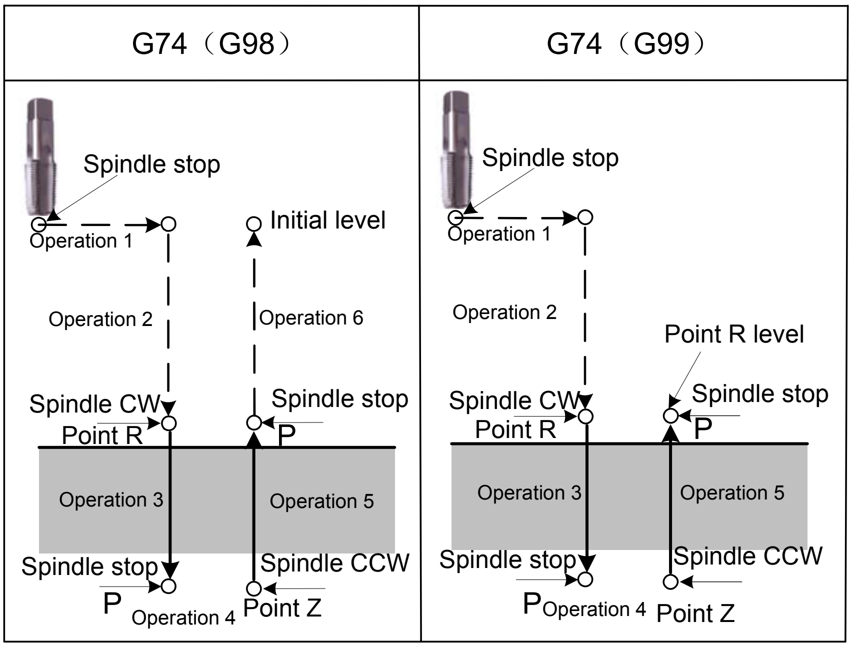


Fig. 4-4-19-1

After positioning along X and Y axes, rapid traverse is performed along 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 completed, 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. Rapid traverse is then performed to initial level. When the tapping is being performed, the feedrate override and the spindle override are assumed to be 100%.

Rigid mode:

Any of the methods below can be used to specify the rigid mode.

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

(2) Specify M29 S***** in the block which contains a taping instruction

If G74 and an M code are specified in the same block, the M code is executed at the time of the 1st hole positioning operation, then the system proceeds to the next tapping operation.

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

Note: In the current version, M00, M01, M02, M06, M30, M98 and M99 are the M codes executed after the other instructions in a block, i.e. these M codes are executed after the execution of the current statement block.

P is a modal instruction, with its min. value set by data parameter P281 and max. value by P282. If P value is less than the value set by P281, the min. value takes effect; if P value is more than the value set by P282, the max. value takes effect. P cannot be stored as modal data if it is specified in a block that does not perform drilling,.

Tool length compensation: If the tool length compensation instruction G43, G44 or G49 is specified in the same block with the canned cycle instruction, the offset is added or cancelled at the time of positioning to point R level; If the tool compensation instruction G43, G44 or G49 is specified in a separate block in the canned cycle mode, the system can add or cancel the offset in real time.

Axis switching: The canned cycle must be cancelled before the taping axis is changed. If the taping axis is changed in the rigid mode, an alarm (No.206) will be issued.

In feed-per-minute mode,

Thread pitch=feedrate/spindle speed.

Federate of Z axis=thread lead×spindle speed.

In feed-per-revolution mode,

Thread lead=feedrate.

Federate of Z axis=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 specification
G74 Z-100 R-20 F1000；       Rigid tapping
```

Restrictions:

F: If the specified F value exceeds the upper limit of the cutting federate, the upper limit is used.

S: If the speed exceeds the maximum speed for a specified gear, its upper limit is used. The speed gear is set by data parameters P294~296.

Cancel: G codes in 01 group (G00 to G03), G60 modal G code (bit parameter NO: 48#0 is set to 1) and G74 cannot be specified in the same block, otherwise G74 will be cancelled.

Tool offset：The tool radius offset is ignored at the time of the canned cycle positioning.

Program restart: It is invalid during the rigid taping.

4.4.20 Right-hand rigid taping G84

Format: G84 X_Y_Z_R_P_F_K_

Function: In rigid taping, the spindle motor is controlled as if it were a servo motor, which is used for high-speed and high-precision taping. It keeps the start positions of the taping unchanged if point R is not changed. Even if taping is performed repeatedly in a position, the threads will not be broken.

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 coordinates 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 coordinates of point R;

P_： Dwell time at the bottom of the hole, with its absolute value used if it is negative;

F_： Cutting feedrate;

K_： Number of repeats.

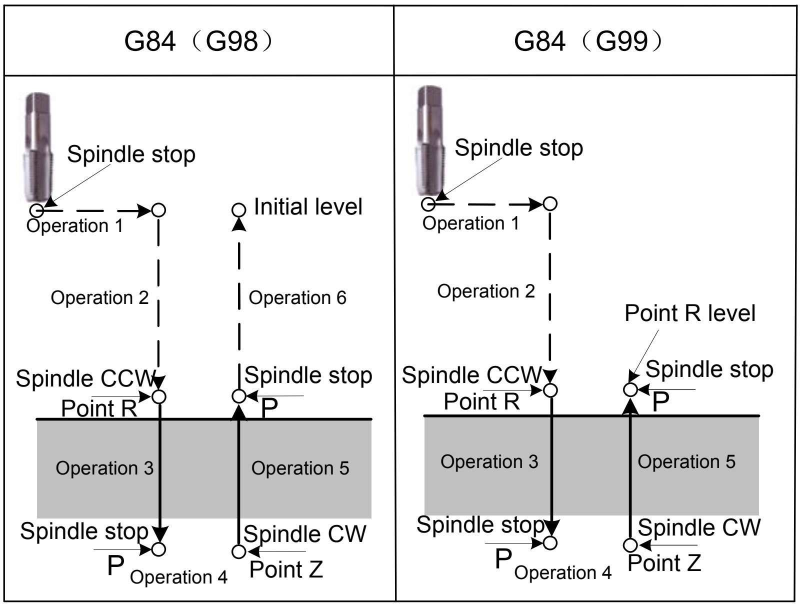


Fig. 4-4-20-1

After positioning along X and Y axes, rapid traverse is performed to point R level along Z axis. The spindle is rotated CCW for tapping from point R level to Z level by G84 instruction. When tapping is completed, 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. Rapid traverse to initial level is then performed.

When taping is being performed, the feedrate override and 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 an M code are specified in the same block, the M code is executed at the time of the 1st hole positioning operation, then the system proceeds to the next tapping operation.

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

Note: In the current version, M00, M01, M02, M06, M30, M98 and M99 are the M codes executed after the other instructions in a block, i.e. these M codes are executed after the execution of the current statement block.

P is a modal instruction, with its min. value set by data parameter P281 and max. value by P282. If P value is less than the value set by P281, the min. value takes effect; if P value is more than the value set by P282, the max. value takes effect. If it is specified in a block that does not perform drilling, P cannot be stored as modal data.

Tool length compensation: If the tool length compensation instruction G43, G44 or G49 is specified in the same block with the canned cycle instruction, the offset is added or cancelled at the time of positioning to point R level; If the tool compensation instruction G43, G44 or G49 is specified in a separate block in the canned cycle mode, the system can add or cancel the offset in real time.

Axis switching: The canned cycle must be cancelled before the taping axis is changed. If the taping axis is changed in the rigid mode, an alarm (No.206) will be issued.

In feed-per-minute mode,

Thread lead =feedrate/spindle speed.

Feedrate of Z axis=spindle speed×thread lead.

In feed-per-revolution mode,

Thread lead=federate.

Federate of Z axis=thread lead

Example: Spindle speed 1000r/min;

Thread lead 1.0mm

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

G00 X120 Y100； Positioning

M29 S1000； Rigid mode specification

G84 Z-100 R-20 F1000； Rigid tapping

Restrictions:

F: If the specified F value exceeds the upper limit of the cutting federate, the upper limit is used.

S: If the speed exceeds the maximum speed for a specified gear, an alarm is issued. The speed gear is set by data parameters P294~296.

Cancel: G codes in 01 group (G00 to G03), G60 modal G code (bit parameter NO: 48#0 is set to 1) and G84 cannot be specified in the same block, otherwise G84 will be cancelled.

Tool offset：The tool radius offset is ignored at the time of the canned cycle positioning.

Program restart: It is invalid during the rigid taping.

4.4.21 Peck rigid taping (chip removal) cycle

Format: G84（or G74） X_Y_Z_R_P_Q_F_K_

Function: In peck rigid taping, cutting is performed several times until the bottom of the hole is reached.

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 coordinates 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 coordinates of point R.

P_： Minimum dwell time at the bottom of the hole or at point R when a return is made. Its absolute value is used if it is negative.

Q_： Cut depth for each cutting feed

F_： Cutting feedrate.

K_： Number of repeats

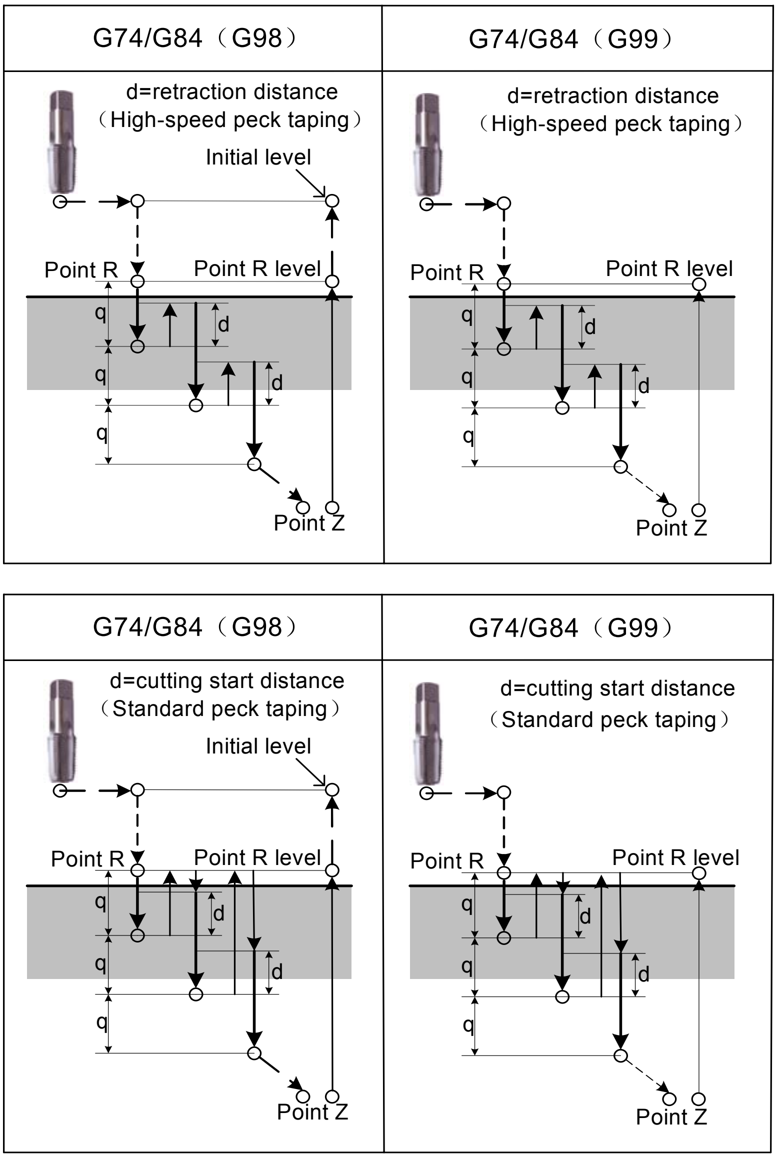


Fig. 4-4-21-1

There are two types of peck rigid tapping cycles: high-speed peck tapping cycle and standard peck tapping cycle, both of which are set by bit parameter NO: 44#5.

When bit parameter NO: 44#5 is 1, the type is high-speed peck tapping cycle: After positioning along X and Y axes, rapid traverse is performed to point R level. The cutting is performed with feed depth Q (cutting depth for each cutting feed) from point R. and then the tool is retracted by a distance d (set by number parameter P284). Whether the override is valid in rigid tapping retraction is set by bit parameter NO:44#4. The retraction speed override is set by bit parameter NO:45#3. Whether the same time constant is used for rigid taping feed and retraction is set by bit parameter NO:45#2; whether the signals for feedrate override selection and feedrate override cancel are valid in rigid taping is set by bit parameter NP:45#4.When point Z has been reached, the spindle is stopped, and then rotated in the reverse direction for retraction.

When bit parameter NO;44#5 is 0, the type is standard peck tapping cycle: After positioning along X and Y axes, rapid traverse to point R level is performed. The cutting is performed with feed depth Q (cutting depth for each cutting feed) from point R, and then a return is performed to point R. Whether the override in rigid tapping retraction is valid is set by bit parameter NO:44#4, and the retraction speed override is set by bit parameter NO:45#3.The moving of cutting feedrate F is performed from point R to a position distance d (set by data parameter P284) to the end point of the last cutting, which is where the cutting is restarted. Whether the same time constant is used in rigid tapping feed and retraction is set by bit parameter NO:45#2. When point Z is reached, the spindle is stopped, and then rotated in the reverse direction for retraction.

Restrictions:

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

S： An alarm is issued if the rotation speed exceeds the max. speed for the gear used. The speed gear is set by number parameter P294~296.

Cancel: G codes in 01 group (G00 to G03), and G84（or G74）cannot be specified in the same block, otherwise G84（or G74）will be cancelled.

Tool offset：The tool radius offset is ignored at the time of the canned cycle positioning.

Program restart: It is invalid during the rigid taping.

4.4.22 Canned cycle cancel G80

Format: G80

Function: It is used for cancelling the canned cycle.

Explanation:

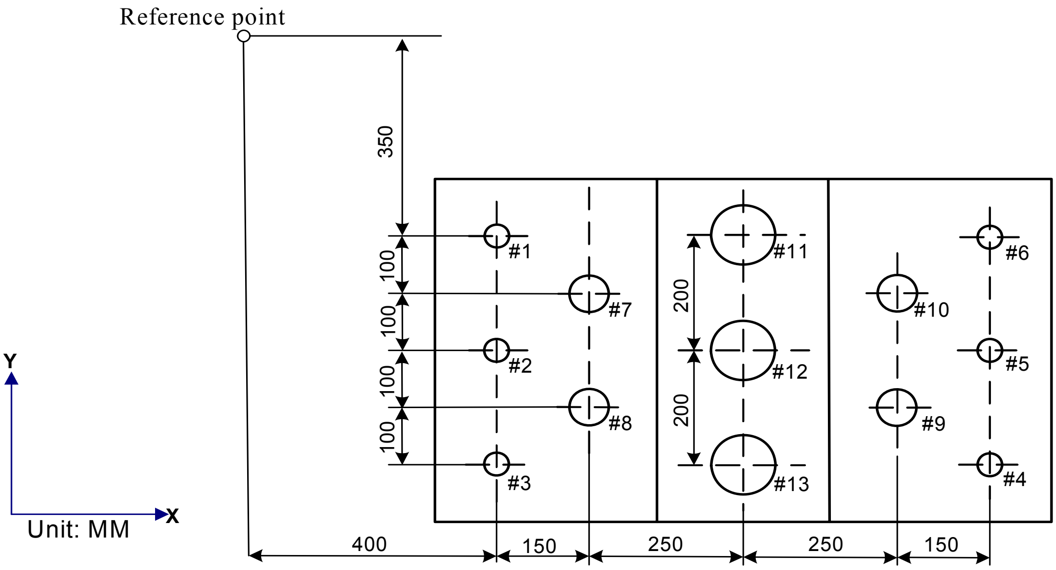
All the canned cycles are cancelled to perform normal operation. Point R, point Z are also cancelled, and the other drilling and boring data is cleared as well.

Example:

```
M3 S100；                       Spindle starts to rotate
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 the initial level
G80；
G28 G91 X0 Y0 Z0；              Return to the reference point and cancel the canned cycle
M5；                            Spindle stop
```

Example:

Explanation for the usage of the canned cycle using the tool length compensation:



 # 1～ 6... drilling of a Φ10 hole
 # 7～10... drilling of a Φ20 hole
#11～13.. boring of a Φ95 hole

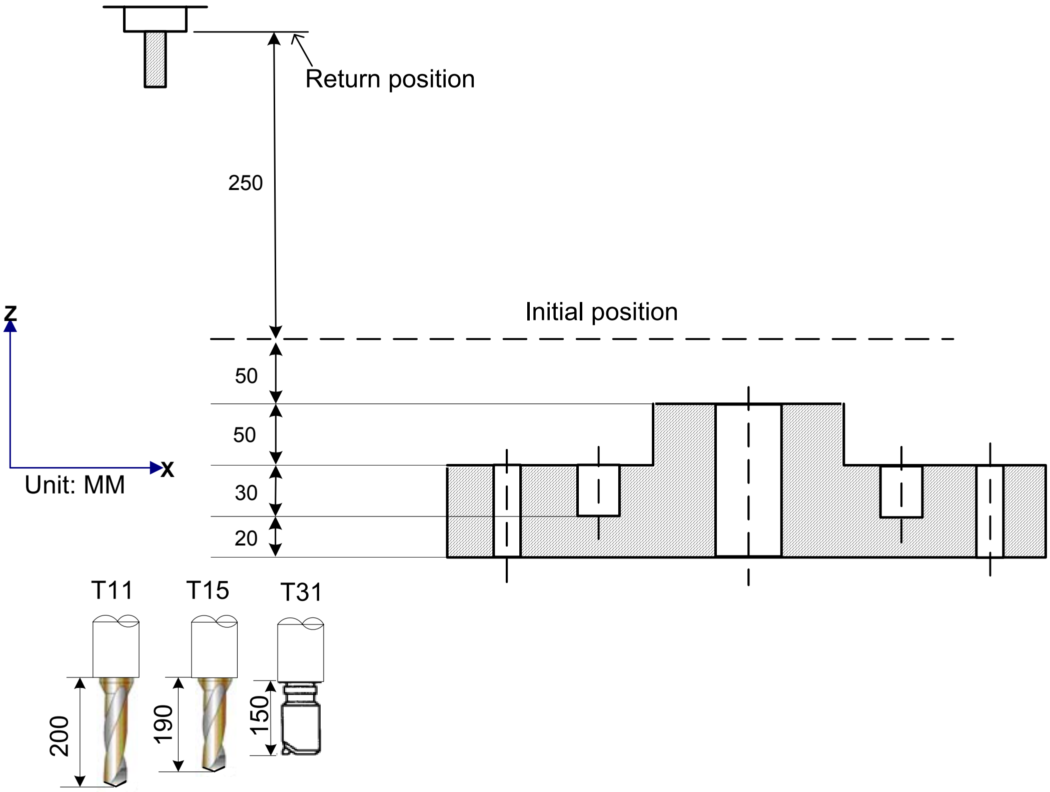


Fig. 4-4-22-1

The values of offset No.11, No. 15 and No. 31 are set to 200, 190 and 150 respectively. The program is as follows:

N001 G92 X0 Y0 Z0 ；	Coordinate system set at reference point
N002 G90 G00 Z250 T11 M6 ；	Tool change
N003 G43 Z0 H11 ；	Tool length compensation at the initial point

N004 S300 M3 ;	Spindle start
N005 G99 G81 X400 Y-350 ; Z-153 R-97 F120 ;	Positioning, then hole #1 drilling
N006 Y-550 ;	Positioning, then hole #2 drilling and point R level return
N007 G98 Y-750 ;	Positioning, then hole #3 drilling and initial level return
N008 G99 X1200 ;	Positioning, then hole #4 drilling and point R level return
N009 Y-550 ;	Positioning, then hole #5 drilling and point R level return
N010 G98 Y-350 ;	Positioning, then hole #6 drilling and initial level return
N011 G00 X0 Y0 M5 ;	Reference point return, then spindle stop
N012 G49 Z250 T15 M6 ;	Tool length compensation cancel, then 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 hole #7 drilling and point R level return
N016 G98 Y-650 ;	Positioning, then hole #8 drilling and initial level return
N017 G99 X1050 ;	Positioning, then hole #9 drilling and point R level return
N018 G98 Y-450 ;	Positioning, then hole #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 hole #11 drilling and point R level return
N024 G91 Y-200 ; Y-200 ;	Positioning, then holes #12 and #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:

$\left.\begin{matrix}G43\\G44\end{matrix}\right\}$ 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 instructions above are used to shift an offset value for the end point of the specified axis. The difference between assumed tool length (usually the 1st tool) and actual tool length used is saved into the offset memory, tools of different length thus can be used to machine the workpiece only by changing the tool length offset values instead of the program.

G43 and G44 specify the different offset directions, and H code specifies the offset number.

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 coordinates of the moving end point specified by an instruction in the program. When G44 is specified, the offset value specified by H code is subtracted from the coordinates of the end position, and the resulting value obtained is taken as the final coordinates of the end position.

G43，G44 are modal G codes, 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. The offset value assigned to the offset number is added to or subtracted from the moving instruction value of Z axis, which obtains the new instruction value of Z axis. H00～H256 can be specified as the offset number as required.

The range of the offset value is as follows:

Table 4-5-1-1

	Range
Offset value H(input in mm)	-999.999 mm～+999.999mm
Offset value H (input in inch)	-99.9998 inch～+99.9998 inch

The offset value assigned to offset number 00 (H00) is 0, which cannot be set in the system.

Note: When the offset value is changed due to the change of the offset number, the new offset value replaces the old one directly rather than being added to the old compensation value. For example:

H01.......................... Offset value 20
H02.......................... Offset value 30
G90 G43 Z100 H01 ； .... Z moves to 120
G90 G43 Z100 H02 ； .... Z moves to 130

3. Sequence of the offset number

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 is cancelled at the end of the block
    M30；
```

4. Tool length compensation cancel

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

Note: 1. After B mode of tool length offset is executed along two or more axes, all the axis offsets are cancelled by specifying G49, however, only the axis offset perpendicular to a specified plane is cancelled by specifying H00.

2. It is suggested that a moving instruction of Z axis be added for the set-up and cancel of the tool length offset, otherwise, the length offset will be set up or cancelled at the current point. Therefore, please ensure a safe height in the Z axis when using G49 to prevent tool collision and workpiece damage.

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 the tool is moved to a specified position (cancelled at the specified position in G53; cancelled at the reference point in G28, G30), but the modal code display 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 length offsets of the axes are cancelled after the specified position is moved to; if G28 or G30 is in the same block with G49, the length offsets of all the axes are cancelled after the reference point is moved to. The cancelled tool length offset vectors will be restored in the next buffered block containing compensation axes.

6. Example for tool length compensation

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

(B) H01= offset value – 4

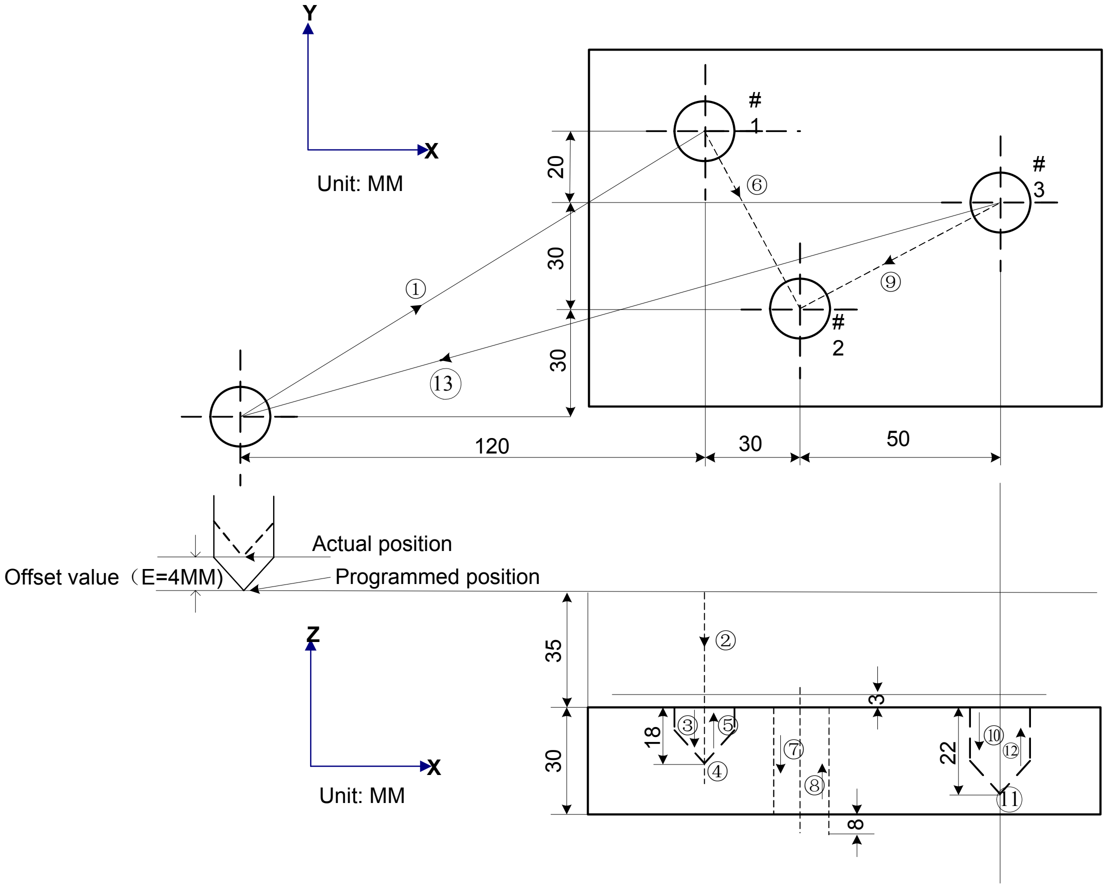


Fig. 4-5-1-1

N1 G91 G00 X120 Y80 ； ....(1)

N2 G43 Z-32 H01 ； .... (2)

N3 G01 Z-21 F200 ； .... …(3)

N4 G04 P2000 ； .... (4)

N5 G00 Z21 ； ....(5)

N6 X30 Y-50 ； ....(6)

N7 G01 Z-41 F200 ； ....(7)

N8 G00 Z41 ； ....(8)

N9 X50 Y30 ； .... ….... (9)

N10 G01 Z-25 F100 ； ....(10)

N11 G04 P2000 ； ....(11)

N12 G00 Z57 H00 ； ....(12)

N13 X-200 Y-60 ； ....(13)

N14 M30 ；

4.5.2 Tool radius compensation G40/G41/G42

Format:

G41 D_ X_Y_；
G42 D_ X_Y_；
G40 X_Y_；

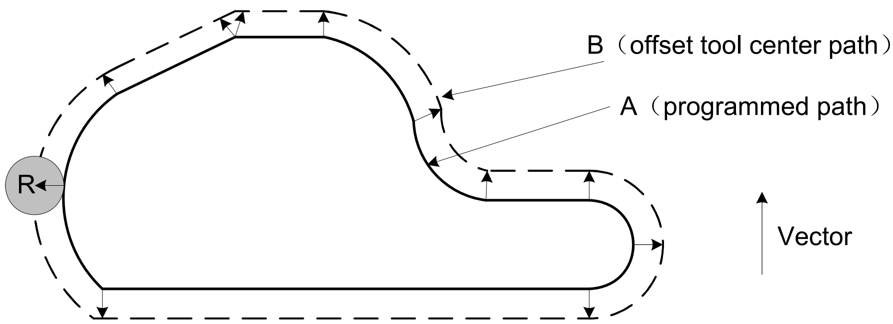
Function:

G41 specifies the left compensation of the tool moving.
G42 specifies the right compensation of the tool moving.
G40 cancels the tool radius compensation.

Explanation:

1. Tool radius compensation

As the following figure, when using a tool with radius R to cut workpiece A, the tool center path is shown as B, and the distance from path B to path A is R. That the tool is moved by tool radius apart from the workpiece A is called compensation.



Compensation and vector

Fig. 4-5-2-1

Programmers write machining programs using the tool radius compensation mode. During the machining, the tool diameter is measured and input into the CNC memory, then the tool path turns into offset path B.

2. Offset value (D value)

The radius offset number is specified by D code. The offset value corresponding to the offset number is added to or subtracted from the moving instruction value in the program, thus obtains the new moving instruction value. The offset number can be specified by D00～D256 as required. Whether the radius offset value is set by parameter value or radius value is selected by bit parameter **N0：40#7.**

The offset value assigned to the offset number can be saved into the offset memory in advance using LCD/MDI panel.

The range of the offset value is as follows:

Table 4-5-2-1

	Range
Offset value D (input in mm)	-999.999mm～999.999mm
Offset value D (input in inch)	-99.9998 inch～99.9998 inch

Note: The default offset value of D00 is 0 that cannot be set or modified by the user.
The change of the offset plane can only be performed after the offset mode is cancelled. If the offset plane is changed without cancelling the offset mode, an alarm will be issued.

3. Plane selection and vector

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

In simultaneous 3-axis control, only the tool path projected on the offset plane is compensated.

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

Table 4-5-2-2

G code	Offset 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 in combination with G00, G01, G02, G03 to define a mode to determine the value and the direction of the offset vector.

Table 4-5-2-3

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

5. G53, G28 or G30 instruction in tool radius compensation mode

If G53, G28, or G30 instruction is specified in tool radius compensation, the offset vector of tool radius offset axis is cancelled after the specified position is reached. (cancelled at the specified position in G53, cancelled at the reference point in G28,G30), and the other axes except tool radius offset axes are not cancelled. When G53 is in the same block with G41/G42, all the axes cancel their radius compensation when the specified position is reached; when G28 or G30 is in the same block with G41/G42, all the axes cancel their radius compensation after the reference point is reached. The cancelled tool radius compensation vector will be restored in the next buffered block containing a compensation plane.

Note: in offset mode, whether the compensation is temporarily cancelled when G28 or G30 moves to the intermittent point is decided by bit parameter No: 40#2.

Tool radius compensation cancel (G40)

In G00, G01 mode, using the following instruction to perform the linear motion from the old vector of the start point to the end point: G40 X__ Y__ . In G00 mode, rapid traverse is performed to the end point along each axis. By using this instruction, the system switches from tool compensation mode to tool compensation cancel 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 forms a new vector perpendicular 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 vector at the start point.

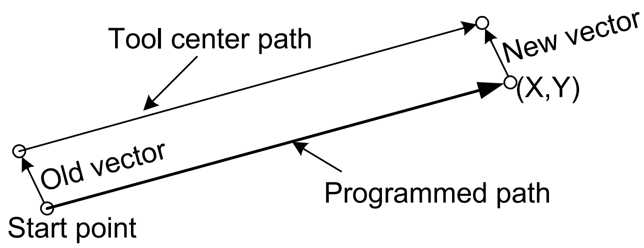


Fig. 4-5-2-2

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

2) In G02, G03 mode

G41……;

……

……

G02 /G03 X__ Y__ R__ ；

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

The offset vector points towards or is apart from the arc center from the start point or the end point.

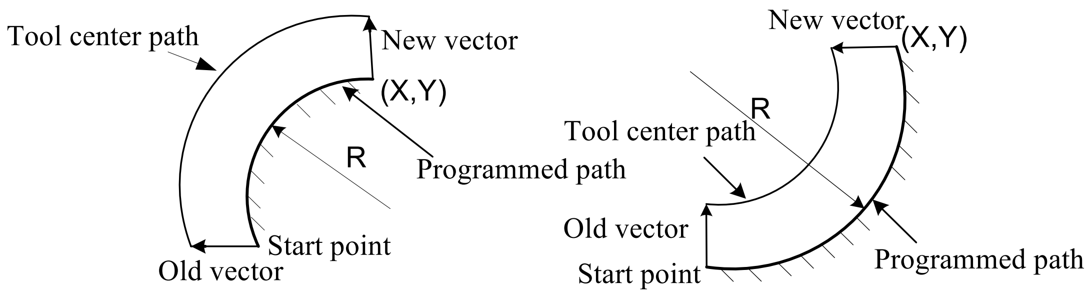


Fig. 4-5-2-3

Tool radius compensation right G42)

In contrast with G41, G42 specifies the tool to deviate at the right side of the workpiece along the tool advancing direction, i.e. the vector direction obtained in G42 is reverse to the vector direction obtained in G41. Except for the direction, the deviation of G42 is identical with that of G41.

1) In G00, G01 mode

G42 X__ Y__ D__ ；

G42 X__ Y__ ；

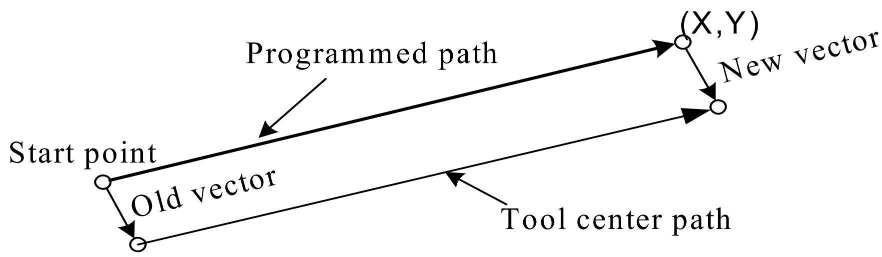


Fig. 4-5-2-4

2) In G02, G03 mode

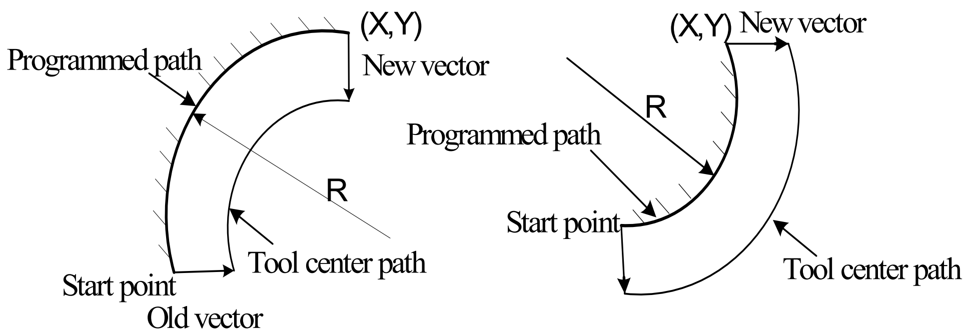


Fig. 4-5-2-5

6. Precautions on offset

（A）Offset number specification

G41，G42 and G40 are modal instructions. The offset number can be specified by D code anywhere before the offset cancel mode is switched to the tool radius compensation mode.

(B) Switching 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 mode to tool radius compensation mode. The circular interpolation (G02, G03) is not permitted.

(C) Switching between tool radius compensation left and tool radius compensation right

In general, the offset direction is changed from the left to the right or vice versus via offset cancel mode, but the direction in positioning (G00) or linear interpolation (G01) can be changed directly regardless of the offset cancel mode, and the tool path is as follows:

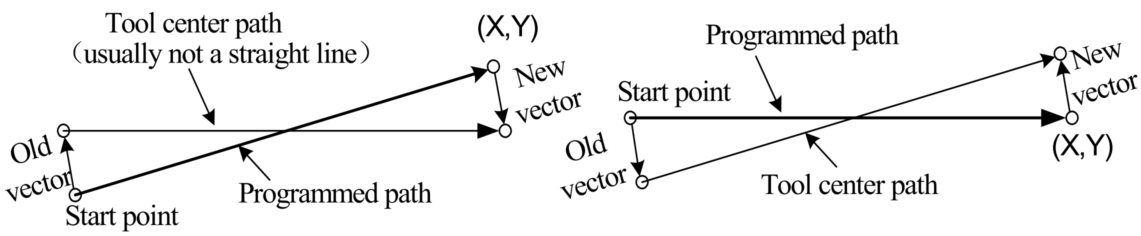


Fig. 4-5-2-6

G1G41 D__X__ Y__；	G42 D__X__ Y__；
……	……
G1G42 D__X__ Y__；	G41 D__ X__ Y__；

(D) Change of offset value

In general, the tool offset value is changed in the offset cancel mode when the tool is changed, but for positioning (G00) and linear interpolation, the value can also be changed in the offset mode. It is shown as follows:

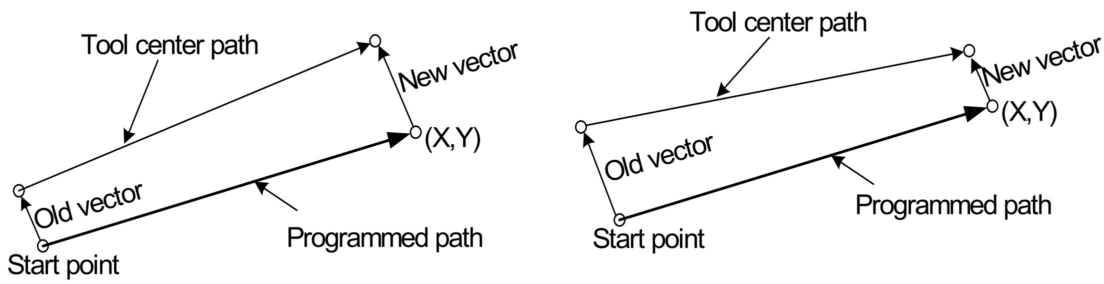


Fig. 4-5-2-7 (Change of offset value)

(E) Positive and negative offset value and the tool center path

If the offset value is negative, the workpiece is machined in the same way as G41 and G42 are replaced with each other in the program. Therefore, the outer cutting for workpiece turns into inner cutting, and the inner cutting turns into outer cutting.

As the usual programming shown in the following figure, the offset value is assumed as positive: When a tool path is programmed as (A), if the offset value is negative, the tool center moves as in (B); when a tool path is programmed as (B), if the offset value is negative, the tool center moves as in (A).

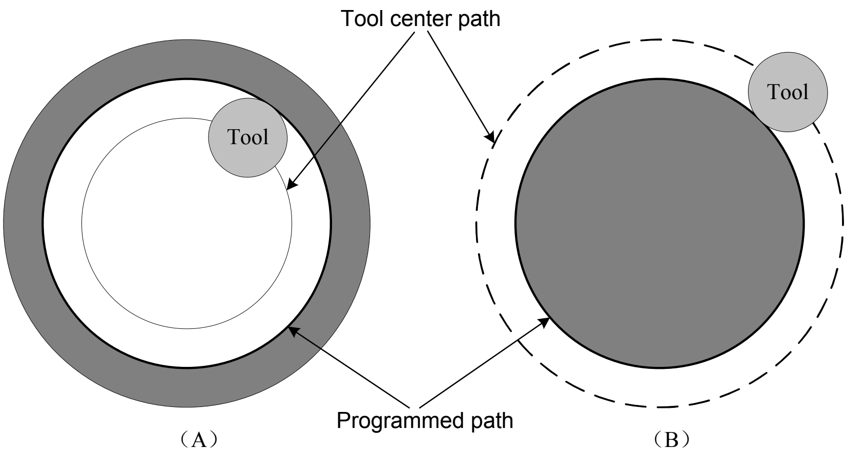


Fig. 4-5-2-8

It is common to see a figure with acute angles (figure with sharp-angle arc interpolation). However, if the offset value is negative, the inner side of the workpiece cannot be machined. When cutting the inner sharp angle at a point, insert an arc with a proper radius there, and then perform cutting after the smooth transition.

The compensation for left or right means the compensation direction is at the left side or right side of the tool moving direction relative to the workpiece (workpiece assumed as unmovable). By G41or G42, the system enters compensation mode, and by G40 the compensation mode is cancelled.

The example for compensation program is as follows:

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

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

Example for tool path compensation program

G92 X0 Y0 Z0；

(1) N1 G90 G17 G0 G41 D7 X250 Y550 ； （Offset value must be preset using 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 ；

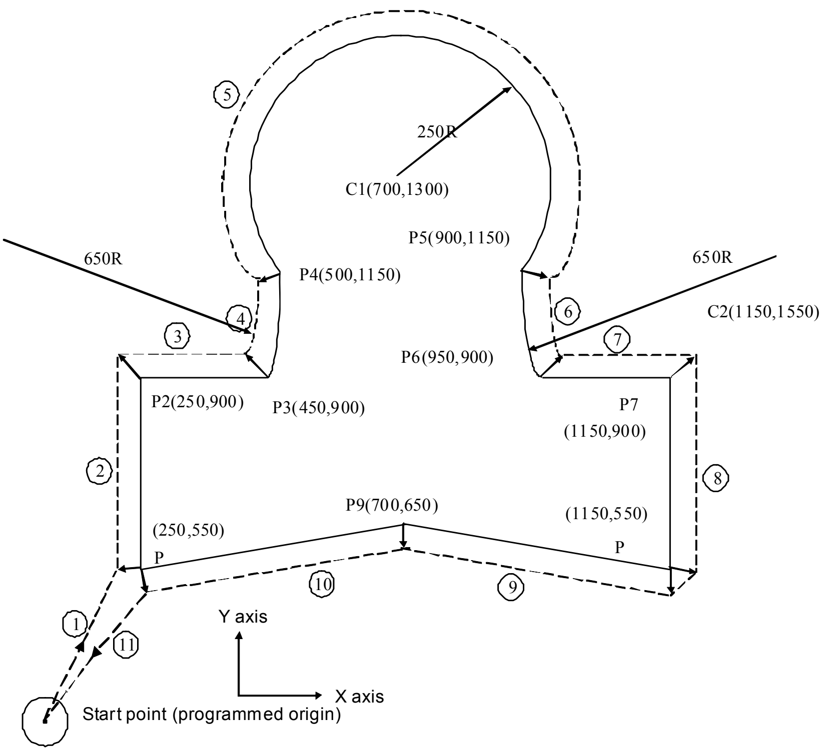


Fig. 4-5-2-9

4.5.3 Explanation for 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.

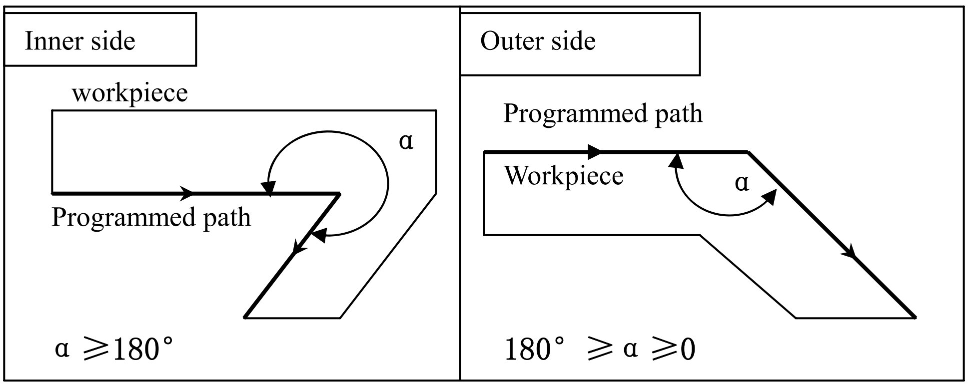


Fig. 4-5-3-1

Symbol meanings:

The following symbols are used in subsequent figures:

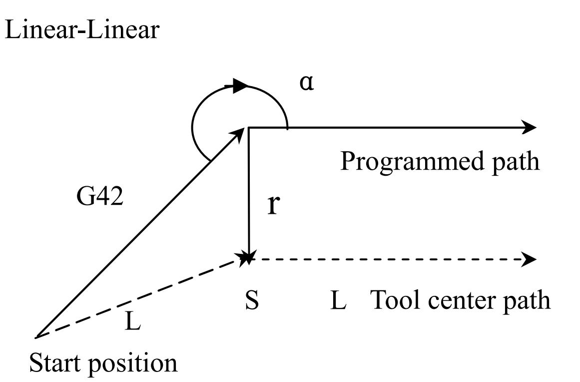
——S indicates a position at which a single block is executed once.
——SS indicates a position at which a single block is executed twice.
——SSS indicates a position at which a single block is executed three times
——L indicates that the tool moves along a straight line.
——C indicates that the tool moves along an arc.
——r indicates the tool radius compensation value.
——An intersection is a position at which the programmed paths of two blocks

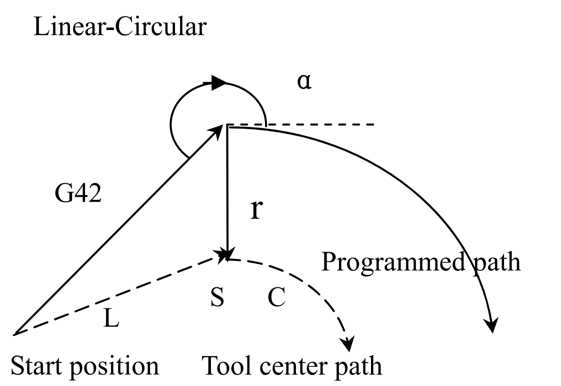
intersect with each other after they are shifted by r.

——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°）

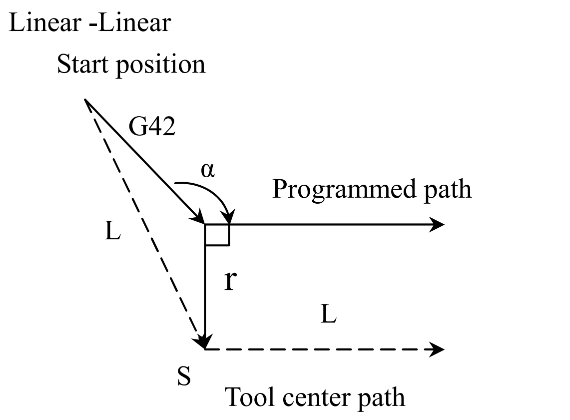


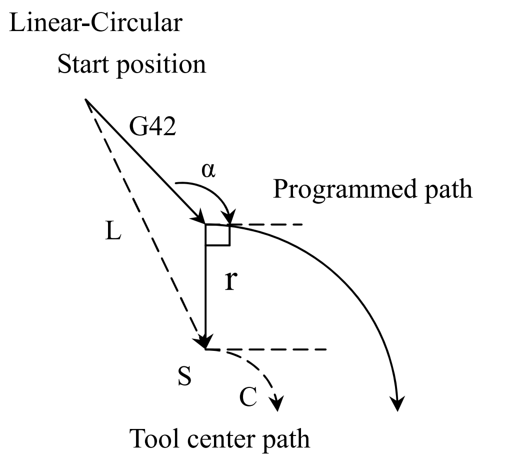


(b) Tool movement around an outer side of a corner at an obtuse angle （180°＞α≥90°）

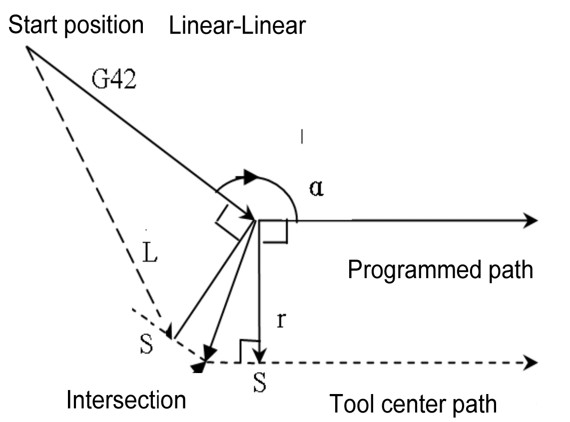
There are 2 tool path types at offset start or cancel: A and B, which are set by bit parameter No：40#0.

A

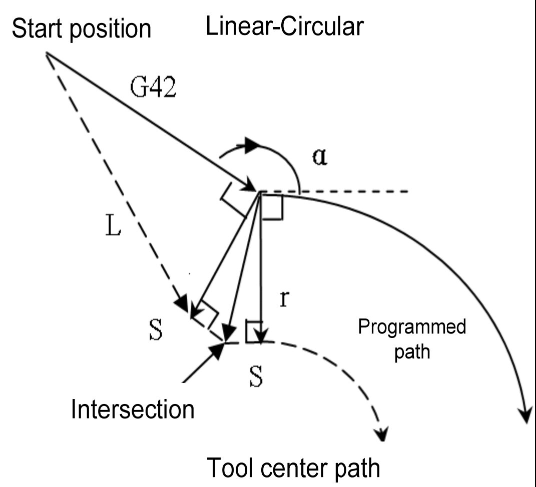




B



Note: Intersection is the position where offset paths of two successive blocks intersect.

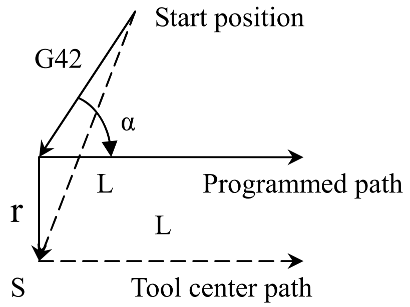


（C）Tool movement around an outer side of a corner at an acute angle （α＜90°）

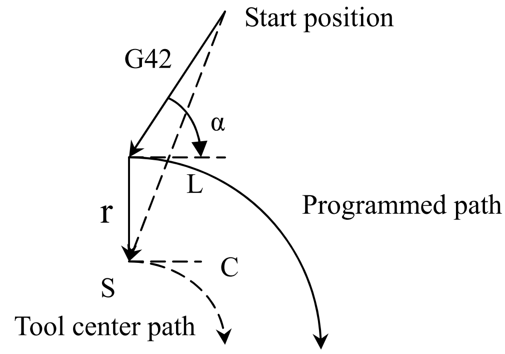
There are 2 tool path types at offset start or cancel: A and B, which are set by bit parameter NO:40#0.

A

Linear-Linear

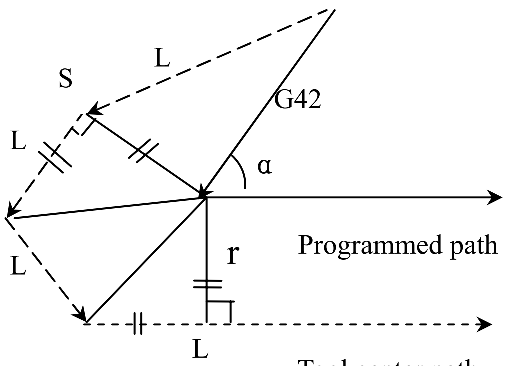


Linear-Circul

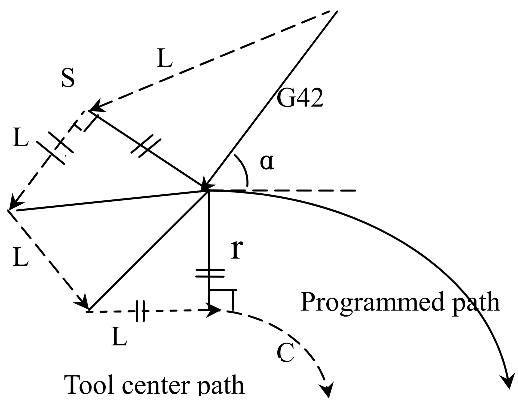


B

Linear-Linear



Linear-Circular



（d）Movement around the outer side of a corner at an acute angle less than 1°（α＜1°）. Linear- linear

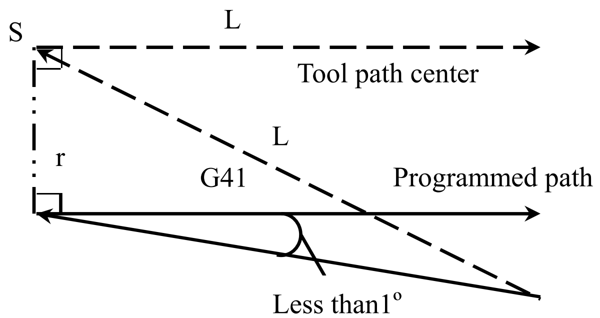


Fig. 4-5-3-2

Book 1 Programming

2. Tool movement in offset mode

An alarm occurs and the tool is stopped if the offset plane is changed when the offset mode is being performed. The tool movement in the offset mode is shown as the figure below.

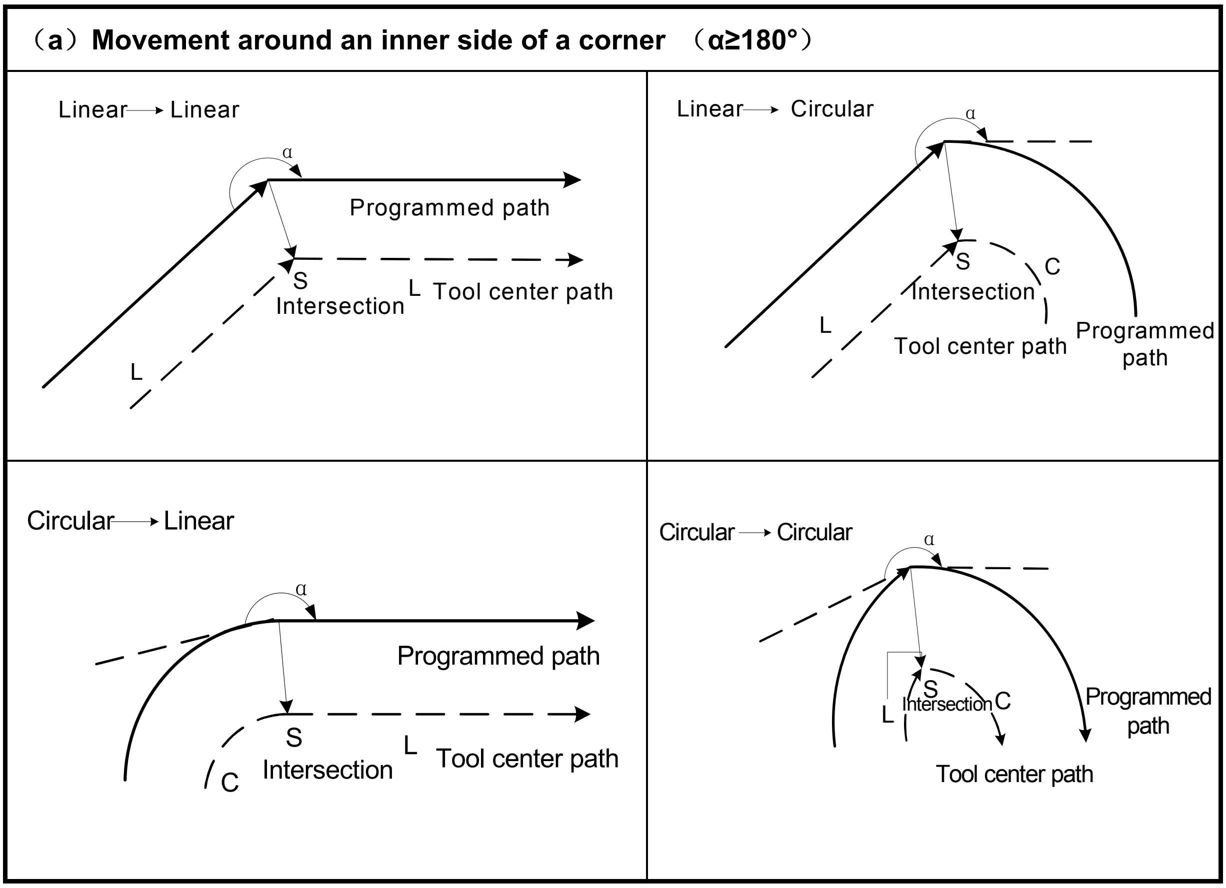
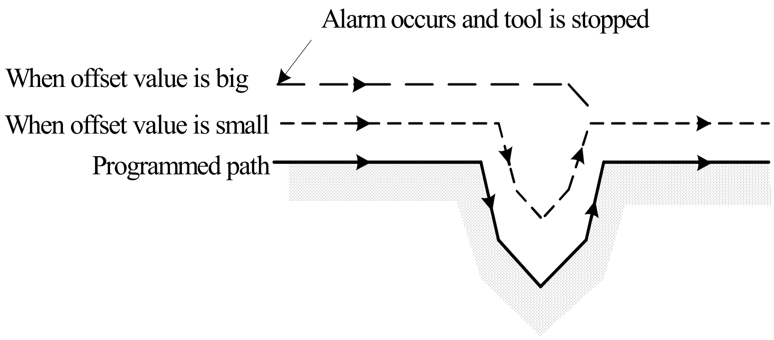


Fig. 4-5-3-3

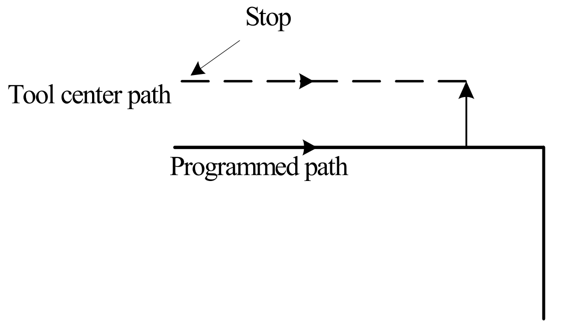
3. Exceptional cases

(I) Without an intersection



In the left figure, if the tool radius is small, the offset value of the arc has an intersection; if the radius gets larger, the intersection may not exist, and an alarm "Interference exist in tool offset C" will be issued in the system.

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



In the left figure, an alarm "Radius tolerance exceeded" will be issued.

（G41）

N5 G01 X100;

N6 G02 X100 I0 J0;

N7 G03 Y-100 J-100;

Fig. 4-5-3-4

4. Tool movement in offset cancel mode

In the offset mode, when a block that satisfies any of the following conditions is performed, the system enters into offset cancel mode. The operation of this block is called the offset cancel.

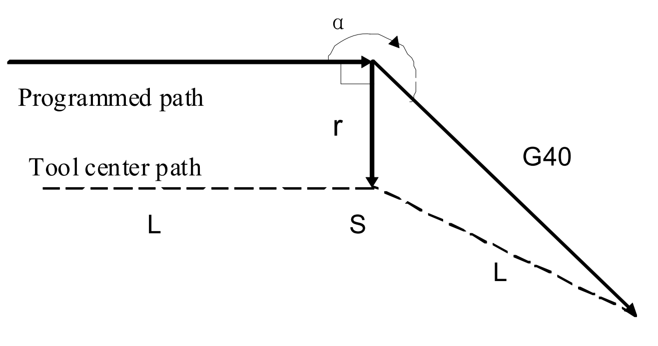
a) Instruction G40

b) When the tool radius compensation number is 0.

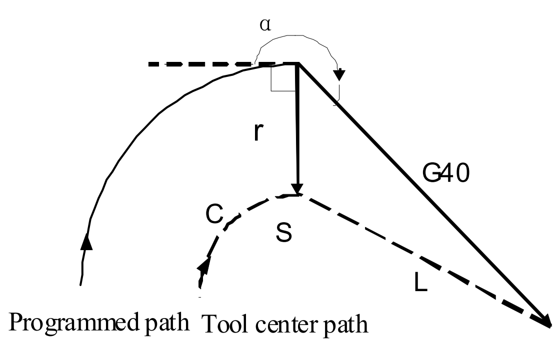
Arc instruction （G03 or G02）cannot be used for cancellation in offset cancel mode. An alarm is issued and tool is stopped if an arc is specified.

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

Linear→Linear



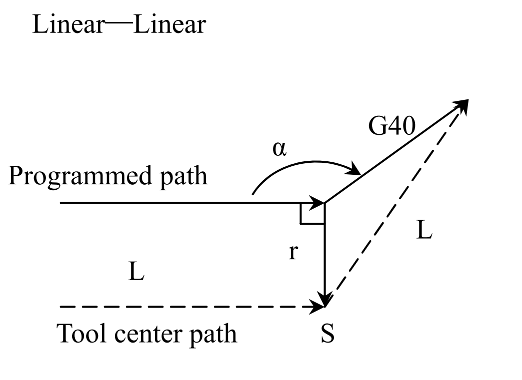
Circular→Linear

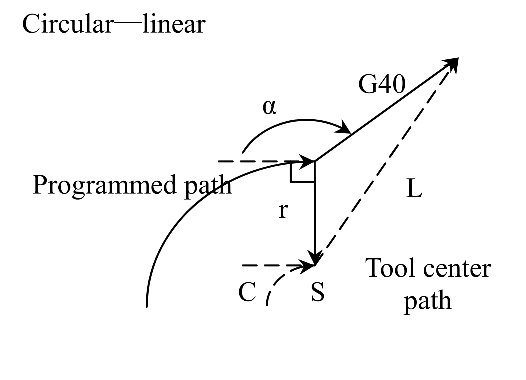


(b) Tool movement around the inner side of a corner (90°≤α<180°)

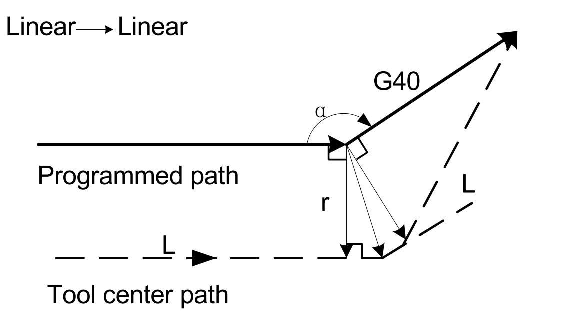
There are 2 tool path types at offset start or cancel: type A and type B, which are set by bit parameter NO：40#0.

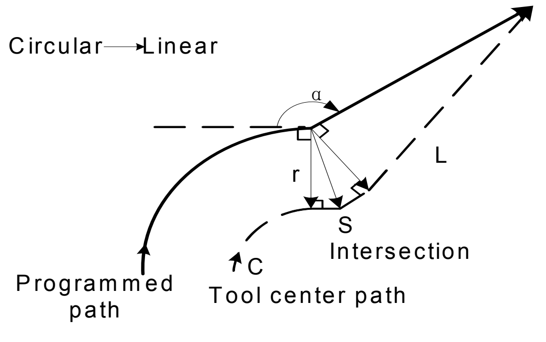
A





B

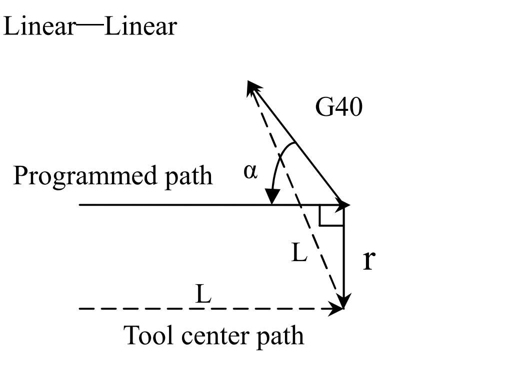


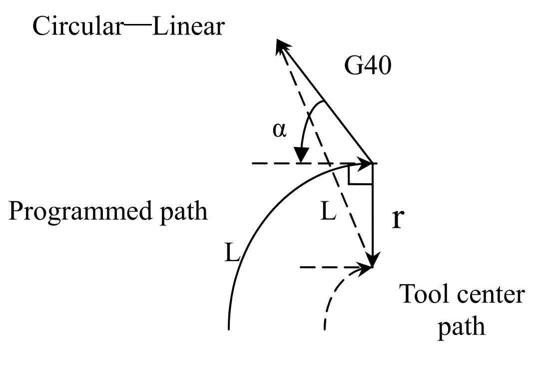


(c) Tool movement around an outer side of a corner at an acute angle (α< 90°)

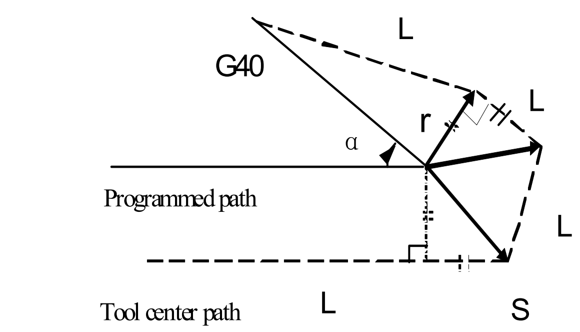
There are 2 types of tool paths at offset start or cancel: type A and type B, which are set by bit parameter NO：40#0.

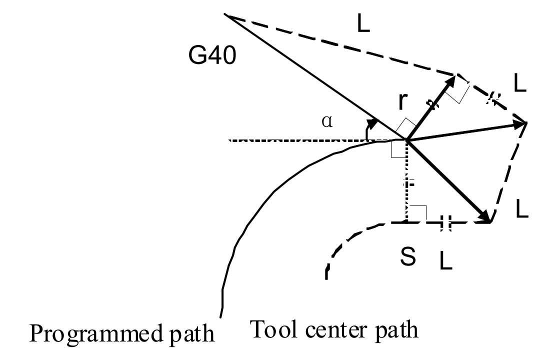
A





B





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

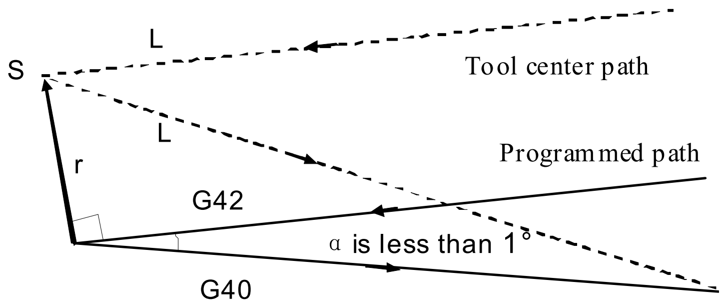


Fig. 4-5-3-5

5. Changing offset direction in offset mode

The offset direction is determined by tool radius compensation G code. The signs of the offset value are as follows:

Table 4-5-3-1

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

In a special case, the offset direction can be changed in offset mode. However, the direction change is unavailable in the start-up block and the block following it. There is no such concepts as inner and outer side when the offset direction is changed. The following offset value is assumed to be positive.

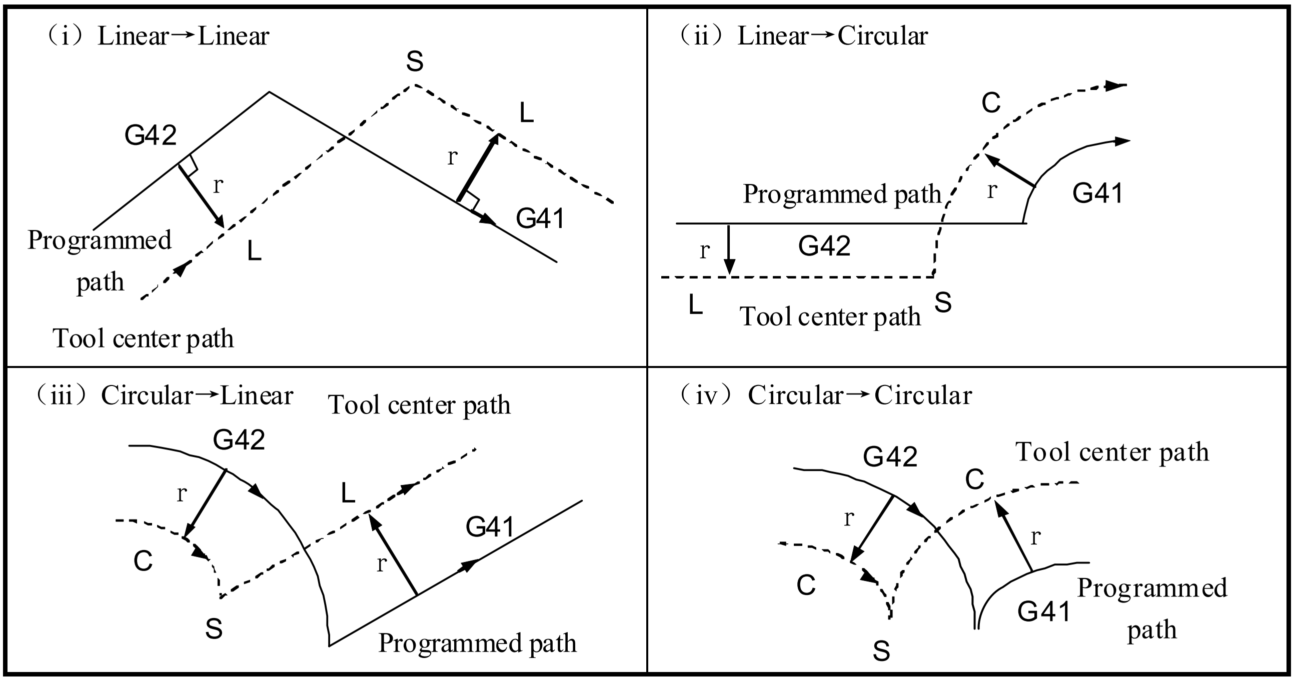


Fig. 4-5-3-6

（v）When the tool compensation is executed normally without an intersection

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

(1) Linear-----Linear

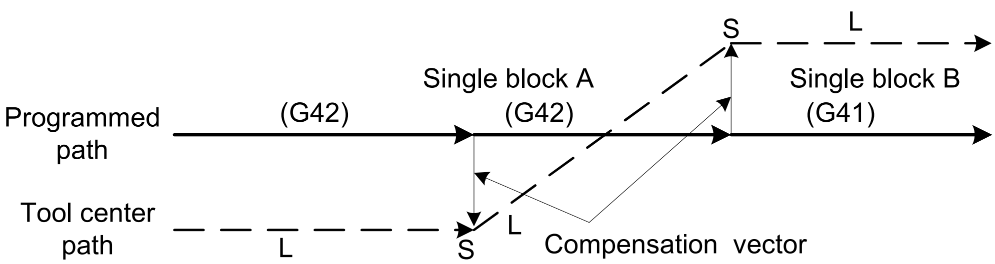


Fig. 4-5-3-7

(2) Linear-----Circular

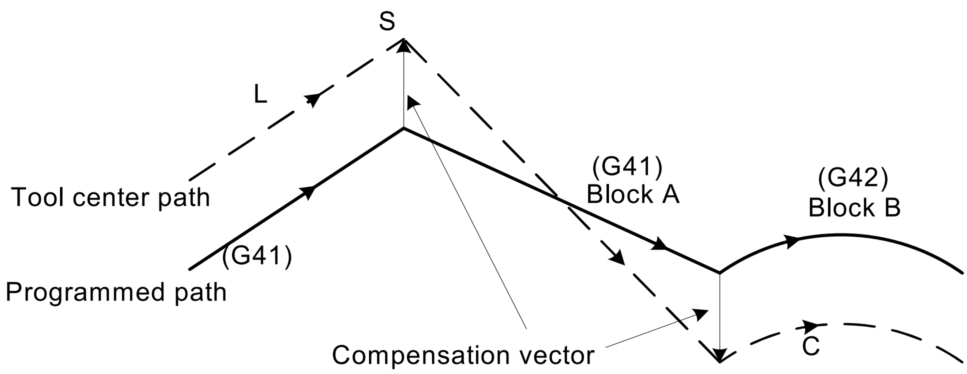


Fig. 4-5-3-8

(3) Circular-----Circular

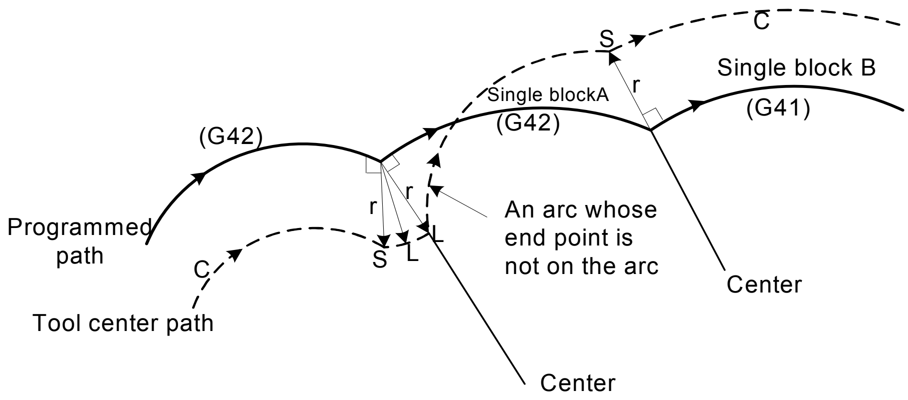


Fig. 4-5-3-9

（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. However, when G41 and G42 are changed, the following situation may occur:

Circular ----- circular (linear-----circular) An alarm occurs when the tool offset direction is changed, and an alarm “Tool offset cannot be cancelled by arc instruction” is issued when the tool number is D0.

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

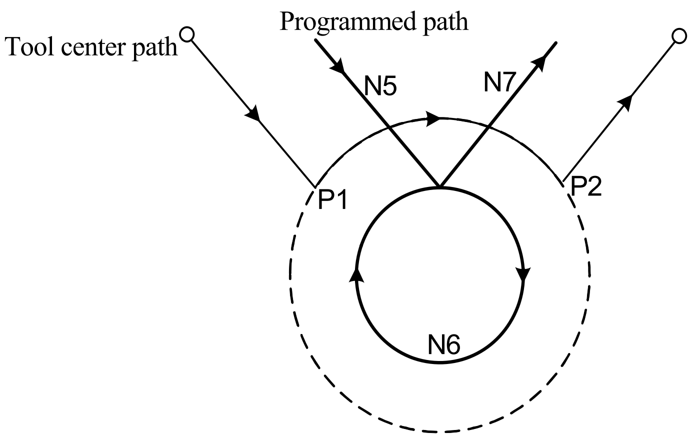


Fig. 4-5-3-10

(G42)
N5 G01 G91 X500 Y-700;
N6 G41 G02 J-500;
N7 G42 G01 X500 Y700;

Here, the tool center path is not an arc of a circle, but an arc from P1 to P2. Under some conditions, an alarm may occur because of the interference check.

To move the tool around a full circle, the circle must be specified in segments.

6. Temporary offset cancel

In offset mode, bit parameter NO：40#2 determines whether the offset is canceled at the intermediate point temporarily when G28, G30 is specified. Please refer to the description of offset cancel and compensation start for detail information about this operation.

a) G28 automatic reference point return

If G28 is specified in offset mode, the offset is cancelled at the intermediate point and automatically restored after reference point return.

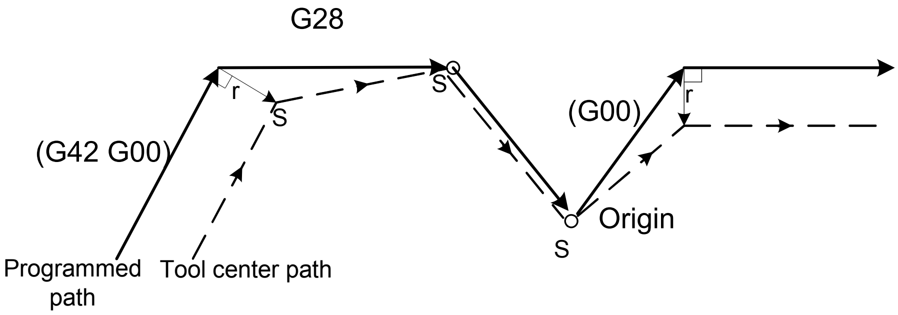


Fig. 4-5-3-11

b) G29 automatic return from reference origin 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:

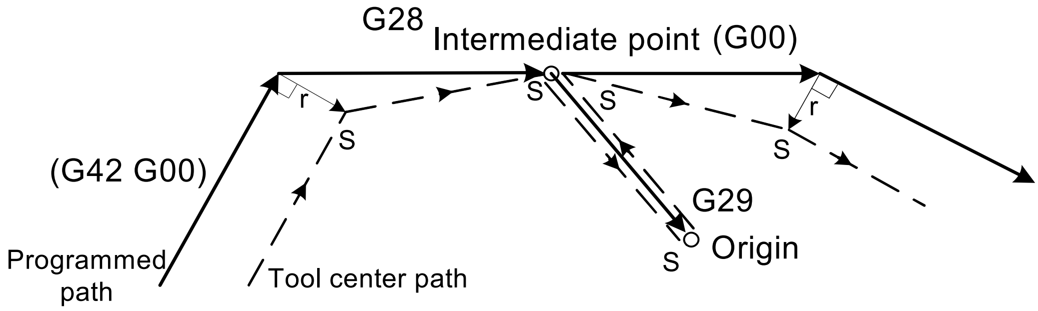


Fig. 4-5-3-12

If it is not specified immediately after G28:

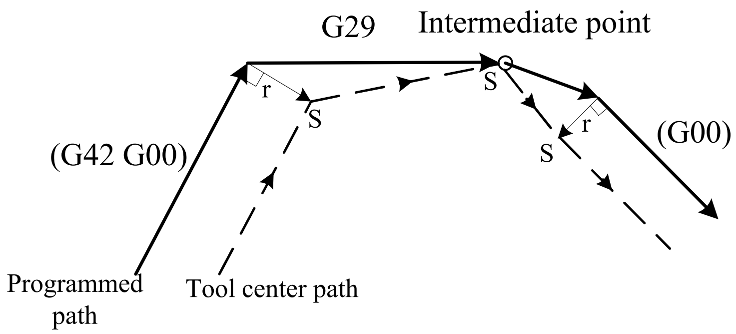


Fig. 4-5-3-13

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 can be set to form a right angle to the moving direction in the previous block, 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.

Refer to (5) when the offset direction is changed using tool radius compensation G (G41，G42).

Linear---Linear

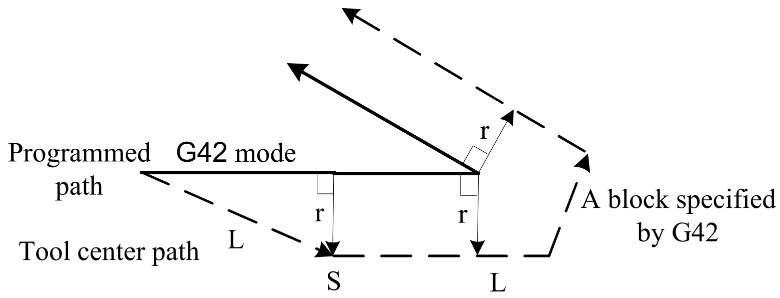


Fig. 4-5-3-14

Circular---Linear

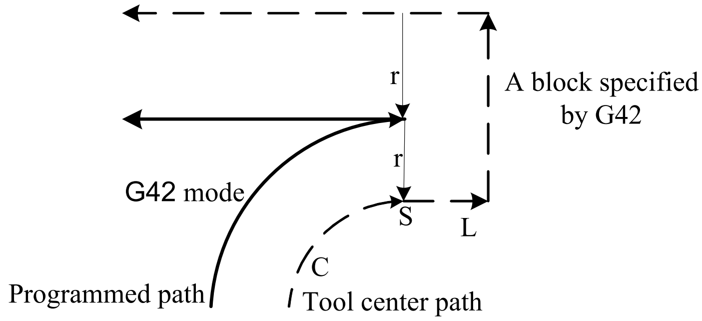


Fig. 4-5-3-15

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 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. When offset mode is restored, the tool moves directly to the intersection again.

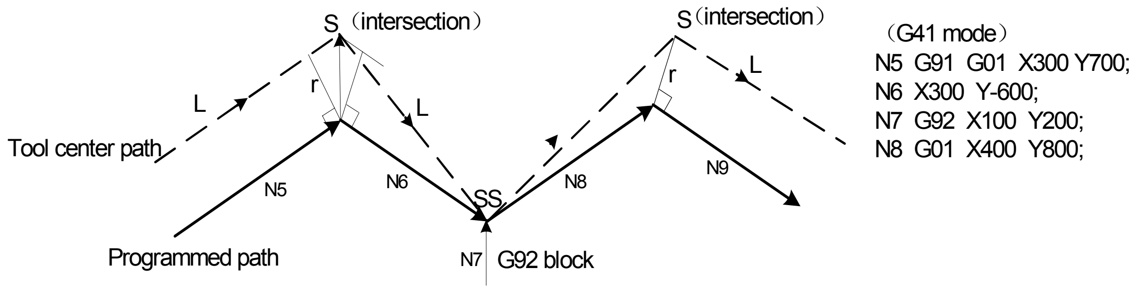


Fig. 4-5-3-16

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 mode 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.

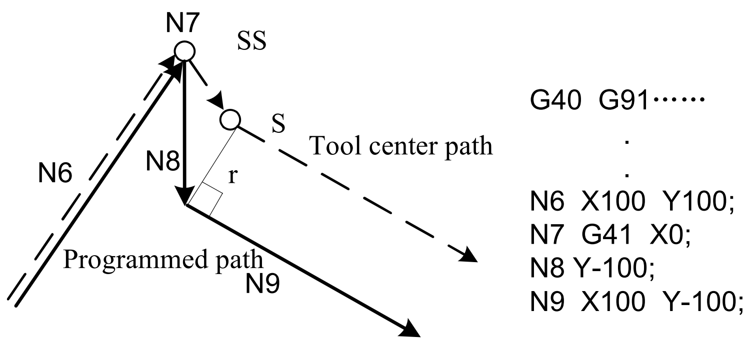


Fig. 4-5-3-17

b) Specified in offset mode

If a single block with no tool movement is specified in offset mode, the vector and the tool center path are the same as when the block is not specified. (Refer to item (3) Offset mode).This block is executed at the single block stop position.

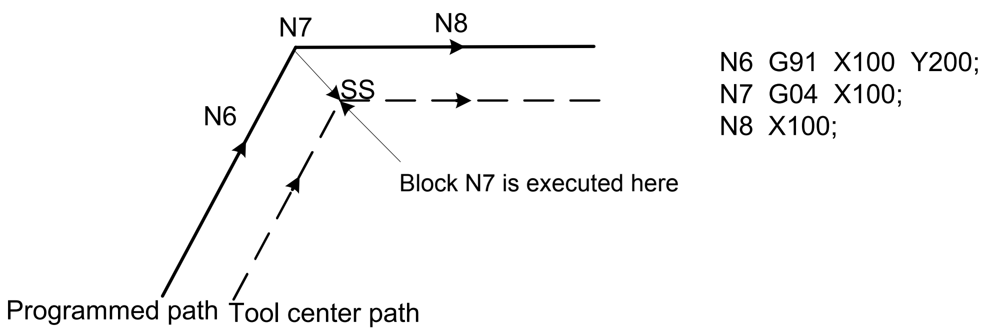
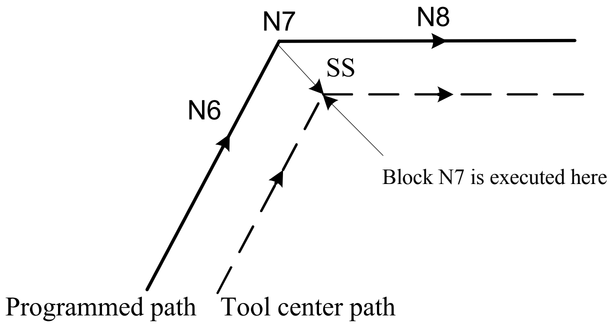


Fig. 4-5-3-18

However, when the block moving amount is 0, the tool movement is the same as that of two or more blocks without moving instructions even if only one block is specified.



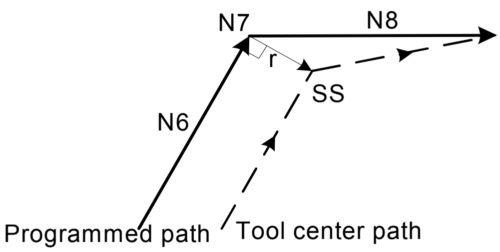
N6 G91 X100 Y200;
N7 X0;
N8 X100;

Fig. 4-5-3-19

Note: The blocks above are executed in G1, G41 mode. The path in G0 does not conform to the figure.

c) Specified together with offset cancel

A vector with a length of offset value and with its direction perpendicular to the movement direction of the previous block is formed when the block specified together with offset cancel contains no tool movement. This vector will be cancelled in next moving instruction.



N6 G91 X100 Y100;
N7 G40;
N8 X100 Y0;

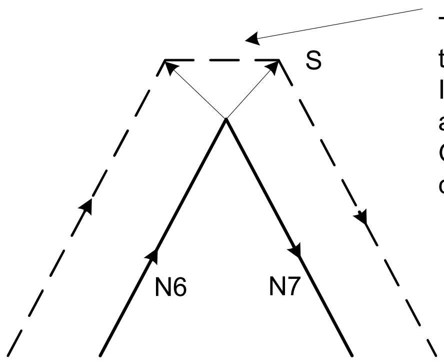
Fig. 4-5-3-20

10. Corner movement

If two or more vectors are formed at the end of the block, the tool traverses linearly from one vector to another. The movement is called corner movement.

If ΔVX≤ΔV limit and ΔVY≤ΔV limit, the latter vector is ignored.

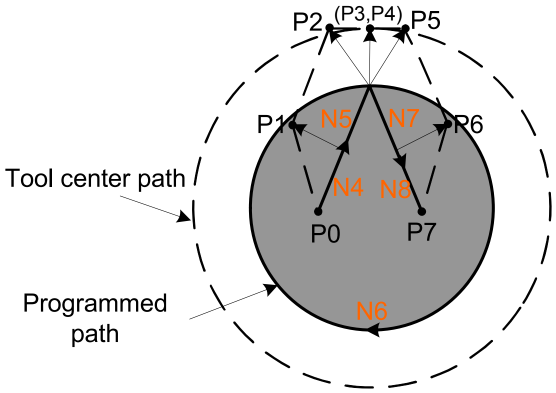
If these vectors do not coincide, then a movement around the corner is created. This movement belongs to the former block.



This move belongs to block N6, thus, the feedrate is equal to that in block N6. If block N6 is G00 mode, the tool moves at rapid feedrate; if block N6 is G01, G02, G03 mode, the tool moves at cutting feedrate.

Fig. 4-5-3-21

However, if the path of the next block overpasses the semicircle, the function above is not performed. The reason is that:




```
N4 G41 G91 X150 Y200;
N5 X150 Y200;
N6 G02 J-600;
N7 G01 X150 Y-200;
N8 G40 X150 Y-200;
```

Fig. 4-5-3-22

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 →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 detected by grammar check function after the program is loaded, an alarm is issued. Whether the interference check is performed during radius compensation is set by bit parameter **NO：41#6.**

Basic conditions for interference

（1）The moving distance of the block which establishes tool radius compensation is less than the tool radius.

(2) The direction of the tool path is different from that of the program path. (The included angle between the two paths is from 90°to 270°).

（3）Besides the above conditions, in arc machining, the included angle between the start point and the end point of the tool center path is very different from that between the start point and end point of the program path(above 180°).

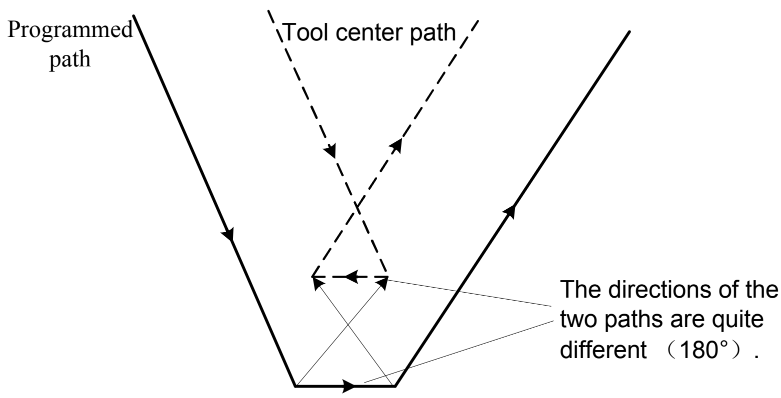


Fig. 4-5-3-23

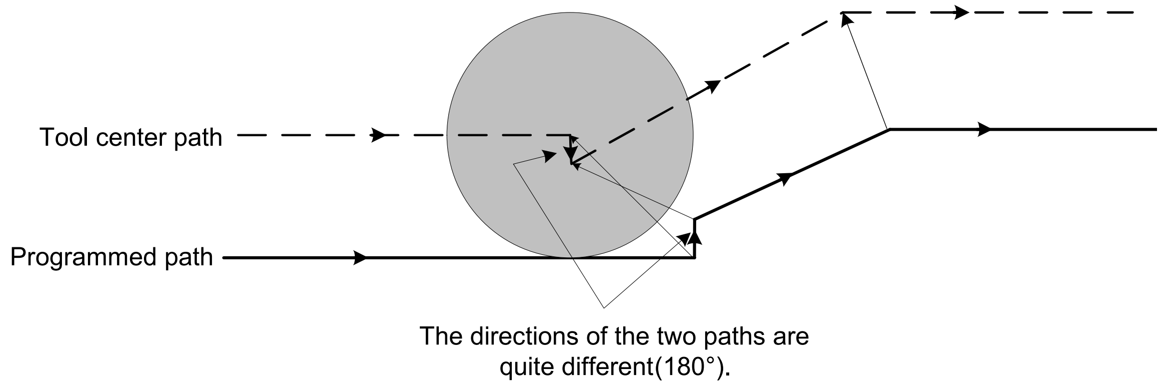


Fig. 4-5-3-24

12. Manual operation

Refer to Manual Operation section in Operation part for the manual operation during the tool radius offset.

13. Precautions for offset

a) Specifying offset value

The offset value number is specified by D code. Once specified, D code keeps effective till another D code is specified or the offset is cancelled. D code is not only used for specifying the offset value for the tool radius compensation, but also for specifying offset value for tool offset.

b) Changing offset value

In general, 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 calculated at the end of the block.

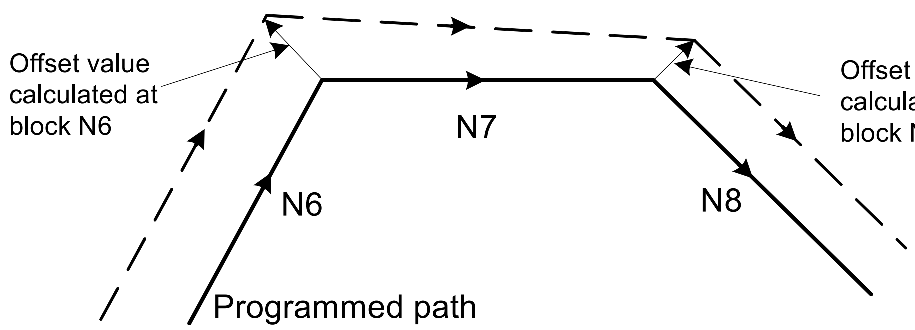


Fig. 4-5-3-25

c) Positive/negative offset value and tool center path

If the offset value is negative(—), G41 and G42 are replaced with each other in the program. If the tool center is passing around the outer side of the workpiece, it will pass around the inner side instead, and vice versa.

As shown in the example below: In general, 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. Therefore, 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.

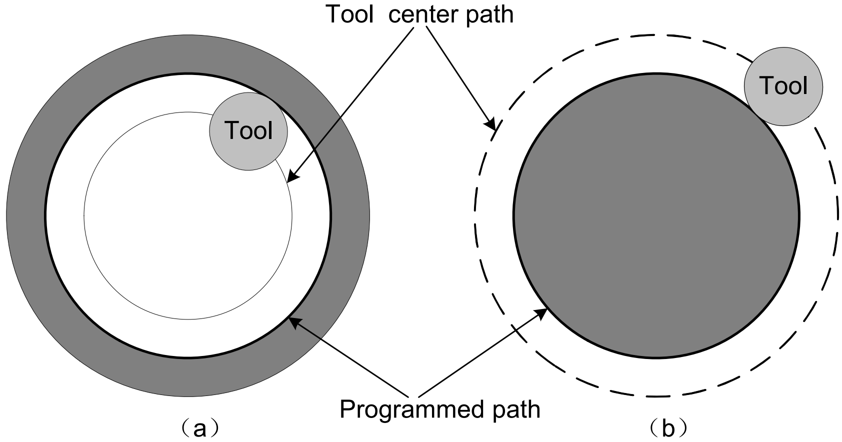


Fig. 4-5-3-26

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 overcutting, an alarm for interference occurs and the CNC stops before the execution of the program.

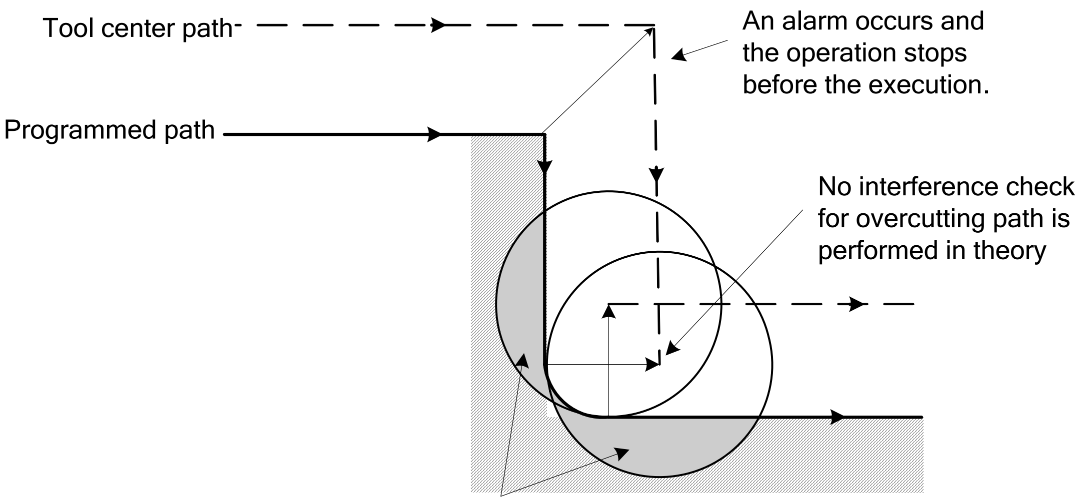


Fig. 4-5-3-27

(2) When machining a groove smaller than the tool radius

When a groove smaller than the tool radius is machined, since the tool radius offset forces the path of the tool center to move in the reverse direction of the programmed path, the overcutting will occur.

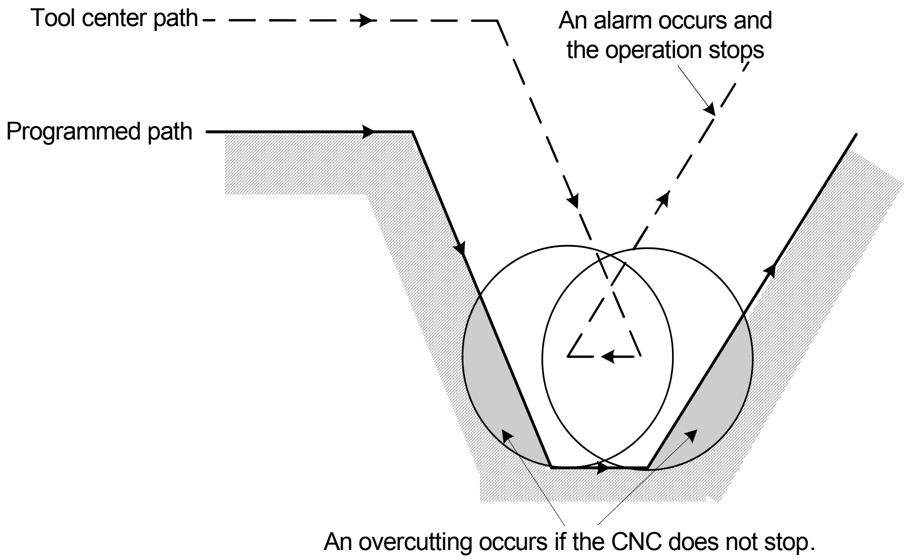


Fig. 4-5-3-28

(3) Machining a step smaller than the tool radius

When the machining of the step is instructed by circular machining in the case of a program containing a step smaller than the tool radius, 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 continues. If the step is linear, no alarm will be issued and the tool cuts correctly. However, the uncut part will exist.

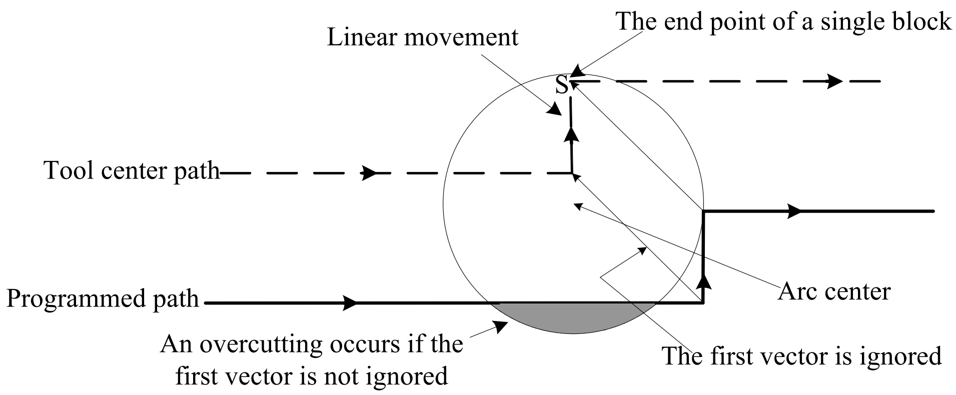


Fig. 4-5-3-29

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 procedure below:

If block N3 is divided as follows:

N1 G91 G00 X500 Y500 H01;

N3 Z-250;

N5 G01 Z-50 F1;

N6 Y100 F2;

N1 G91 G0 X500 Y500 H01;
N3 G01 Z-300 F1;
N6 Y100 F2;

N6 is entered into the buffer storage when N3 is being executed. By the relationship between them the correct offset is performed in the left figure.

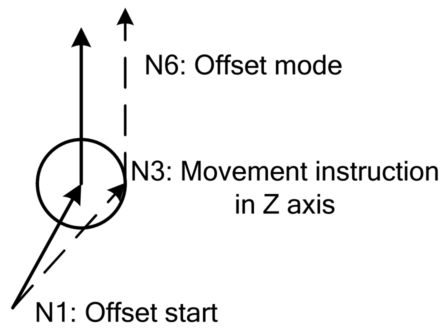


Fig. 4-5-3-30

4.5.4 Corner offset circular interpolation（G39）

Format: G39

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. Whether the corner arc is valid or not is determined by bit parameter **NO：41#5.**

Explanation:

1. When G39 is specified, corner circular interpolation of 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 is programmed, the arc is formed at the corner 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:

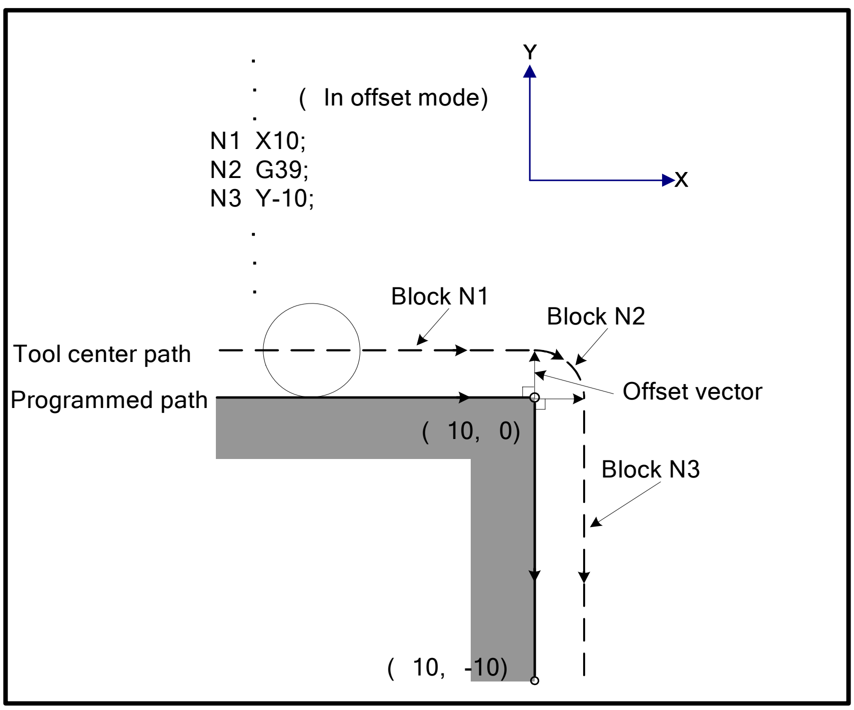


Fig. 4-5-4-1 G39

4.5.5 Tool offset value and offset 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）
Value to be added to the value of the specified offset number in incremental mode（G91）(the sum is the tool offset value).

Explanation: The range of tool offset value:
Geometric offset: metric input -999.999mm~+999.999mm；
inch input -99.9998inch~+99.9998inch
Wear offset: metric input -400.000mm~+400.000mm；
inch input -40.0000inch~+40.0000inch

Note 1: For inch and metric conversion, whether the tool offset value is converted automatically is set by bit parameter No.41#0.
Note 2: The max. value of the wear offset is restrained by data parameter P267.

4.6 Feed G code

4.6.1 Feed mode G64/G61/G63

Format: Exact stop mode **G61**
Taping mode **G63**
Cutting mode **G64**

Function:

Exact stop mode G61: Once specified, this function keeps 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 keeps effective till G61, G62 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, both feedrate override and feed hold are invalid.

Cutting mode G64: Once specified, this function keeps effective till G61, G62 or G63 is specified. The tool is not decelerated at the end point of a block, and the next block is executed.

Explanation:

1. No parameter format.
2. G64 is the system default feed mode, no deceleration is performed at the end point of a block, and next block is executed directly.
3. The purpose of in-position check in exact stop mode is to check whether the servo motor has reached within a specified position range.
4. In exact stop mode, the tool movement paths in cutting mode and tapping mode are different.

See figure 4-6-1-1

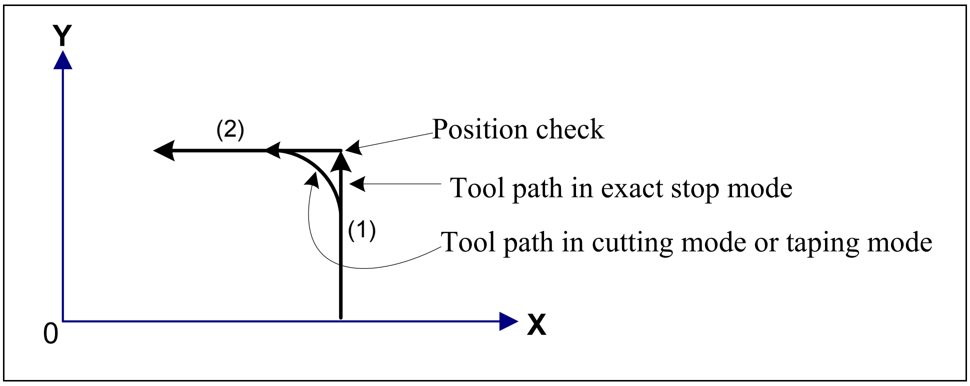


Fig. 4-6-1-1 Tool path from block 1 to block 2

4.6.2 Automatic override for inner corners（G62）

Format: G62

Function: Once specified, this function keeps 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 time. In this way, a smooth machined surface is produced.

Explanation:

1. When the tool moves along an inner corner and inner arc area during tool radius compensation, it is decelerated automatically to reduce the load on the tool and produce a smooth machined surface.
2. Whether automatic corner override function is valid or not is set by bit parameter **NO：16#7;** Automatic corner deceleration function is controlled by bit parameter NO：15#2(0: angle control, 1: speed difference control).
3. When G62 is specified, and the tool path with tool radius compensation applied forms an inner corner, the feedrate is automatically overridden at both ends of the corner. There are four types of inner corners as shown in Fig. 4-6-2-1. In the figure: 2°≤θ≤θp≤178°; θp is set by data parameter P144.

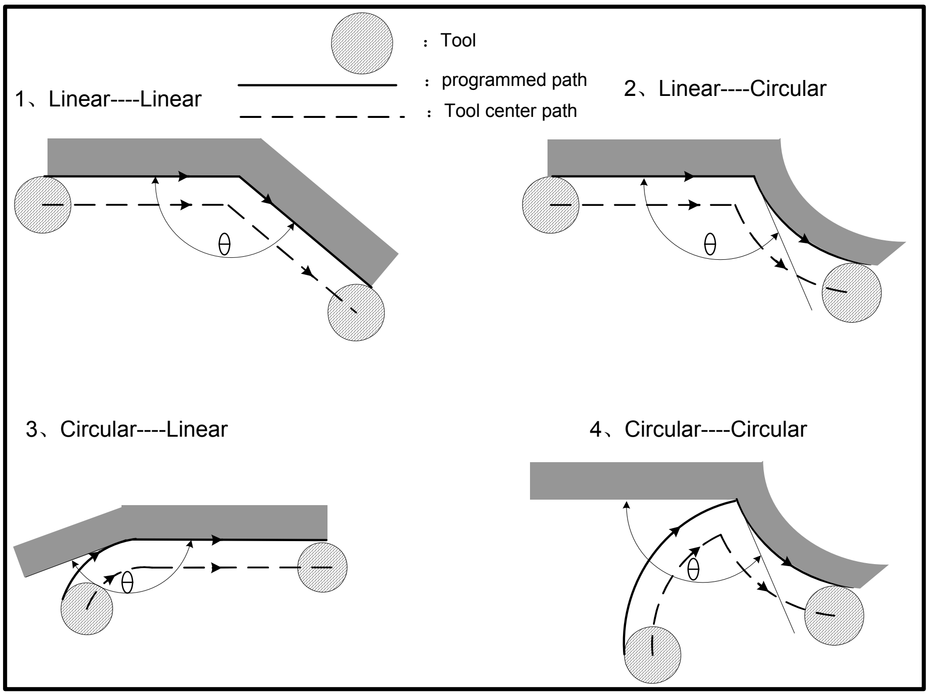


Fig. 4-6-2-1

4. When a corner is determined to be an inner corner, the feedrate is overridden before and after the inner corner. The Ls and Le, where the feedrate is overridden, are distances from points on the tool center path to the corner. As shown in Fig. 4-6-2-2, Ls+Le≤2mm.

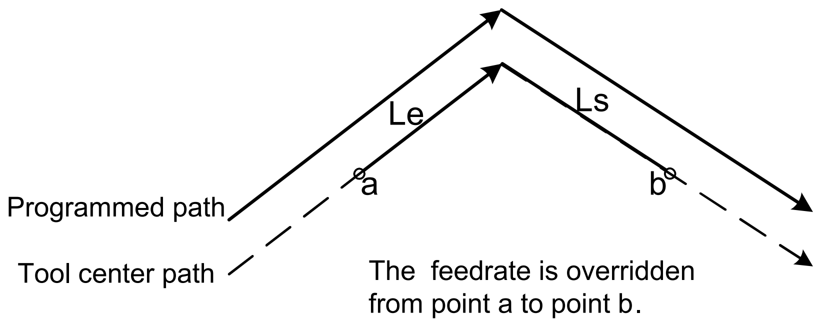


Fig. 4-6-2-2 Straight line to straight line

5. When a programmed path consists of two arcs, the feedrate is overridden if the start and end points are in the same quadrant or in adjacent quadrants. (Fig. 4-6-2-3)

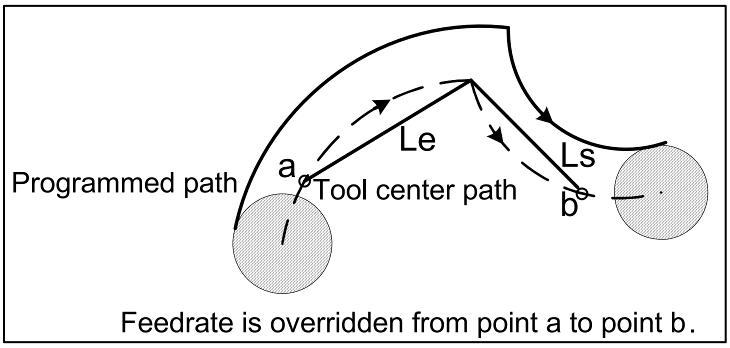


Fig. 4-6-2-3 Arc to arc

6. Regarding a program from straight line to arc or from arc to straight line, the feedrate is overridden from point a to point b and from point c to point d. (Fig. 4-6-2-4)

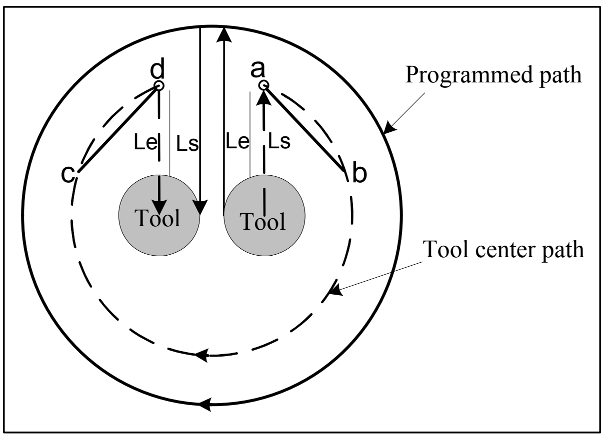


Fig. 4-6-2-4 Straight line to straight line, arc to straight line

Restrictions:

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 corners is not performed if the offset is zero.

4.7 Macro G code

4.7.1 Custom macro

The functions realized by a group of instructions can be prestored into memory like a subprogram using an representing instruction. If the instruction is written into the program, all these functions can be realized. This group of instructions is called custom macro body, and the representing instruction is called "custom macro instruction". Moreover, the custom macro body is also called "macro program" for short, and the custom macro instruction is also called macro calling instruction.

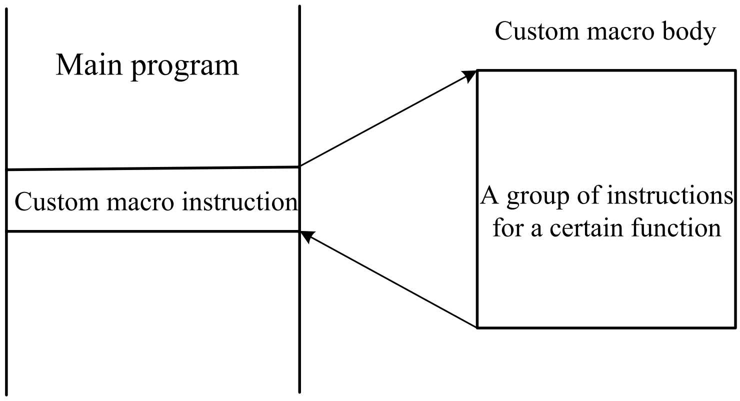


Fig. 4-7-1-1

Variables can be used in custom macro body. Operation can be performed between them and they can be assigned values by macro instructions.

4.7.2 Macro variables

The common CNC instructions and the variables, operation as well as the transfer instructions can be used in the custom macro body.

The custom macro body begins with a program number and ends with M99.

```
O0066;                          Program number
G65  H01  ……;                   Operation instruction
 G90  G00  X#101  ……;           CNC instruction using variables
……
……
……
G65  H82  ……;                   Transfer instruction
……
……
M99;                            Custom macro body ends
```

Fig. 4-7-1-2 (structure of custom macro body)

1. Variable 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 the execution of custom macro body.

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, the format of which is as follows:

#i (i = 1， 2， 3， 4 ……)

(example) #5， #109， #1005

(2) Variable reference

The variable can be used to replace the value of a parameter.

(Example) F#103 When #103 = 15, it is the same as F15.

G#130 When #103 =3, it is the same as G3.

Note 1: Variables cannot be referenced by parameter word O and N (program number and sequence number), e.g., O#100 and N#120 are not permitted in programming.
Note 2: Variables exceeding the max. limit of the parameter cannot t be used. When #30 = 120, M#30 exceeds the max. limit of the instruction.
Note 3: Display and setting of variable values: The values can be displayed on LCD, or be set by MDI mode.

2. Types of variables

Variables are divided into null variables, local variables, common variables and system variables depending on their different applications and characteristics.

（1）Null variable: #0 (This variable is always null, so 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 or the program ends (M30 or M02 is executed), they are cleared automatically; whether the local variables are cleared or not after reset is set by bit parameter NO：52#7. When a macro is called, arguments are assigned to local variables.

（3）Common variables: #100～#199，#500～#999：

Whether common variables #100～#199 are cleared or not after reset is set by bit parameter NO：52#6.

The common variables 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 the same as those in other macro programs. Therefore, the common variable #I of operation result of a macro program can be used in other macro programs.

The usage of common variables is not specified in this system, users thus can define it freely.

Table 4-7-1-1

Variable number	Variable type	Function
# 100～ # 199	Common variable	They are cleared at power-off, and all are initialized to “null” at power-on

# 500～ # 999		Data is saved in files and it will not be lost even if the power is turned 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 --- #1015 (read signal input to system from PLC by bit, i.e. G signal)
#1032 (read signal input to system from PLC by byte, i.e., G signal)
2) Interface output signal #1100 --- #1115 (write signal output to PLC from the system by bit, i.e. F signal)
#1132 (write signal output to PLC from the system by byte, i.e. F signal)
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) Alarm #3000
8) User data list #3500～#3755 （read-only，unwritable）
9) Modal message #4000～#4030 （read-only，unwritable）
10) Position message #5001～#5030 （read-only，unwritable）
11) Workpiece zero offset #5201～#5235 （readable and writable）
12) Additional workpiece coordinate system #7001～#7250 （readable and writable）

3. Explanation for system variables

1) Modal message

Table 4-7-1-2

Variable number	Function	Group number
#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	G22,G23,G24,G25,G26 G32,G33,G34,G35,G36,G37,G38 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	Reserved	16
#4017	Reserved	17
#4018	Reserved	18
#4019	Reserved	19
#4020	Reserved	20
#4021	Reserved	21
#4022	D	

#4023	H	
#4024	F	
#4025	M	
#4026	S	
#4027	T	
#4028	N	
#4029	O	
#4030	P (current selected additional workpiece coordinate system)	

Note 1: P code indicates the current selected additional workpiece 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 not written.

2) Current position message

Table 4-7-1-3

<table>
<tr><th>Variable number</th><th>Position message</th><th>Relative coordinate system</th><th>Reading operation during moving</th><th>Tool offset value</th></tr>
<tr><td>#5001</td><td>Block end position of X axis (ABSIO)</td><td rowspan="4">Workpiece coordinate system</td><td rowspan="4">allowed</td><td rowspan="4">Tool nose position not involved （Position instructed by program）</td></tr>
<tr><td>#5002</td><td>Block end position of Y axis (ABSIO)</td></tr>
<tr><td>#5003</td><td>Block end position of Z axis (ABSIO)</td></tr>
<tr><td>#5004</td><td>Block end position of 4th axis (ABSIO)</td></tr>
<tr><td>#5006</td><td>Block end position of X axis (ABSMT)</td><td rowspan="4">Machine coordinate system</td><td rowspan="8">unallowed</td><td rowspan="12">Tool reference Position involved （Machine coordinate）</td></tr>
<tr><td>#5007</td><td>Block end position of Y axis (ABSMT)</td></tr>
<tr><td>#5008</td><td>Block end position of Z axis (ABSMT)</td></tr>
<tr><td>#5009</td><td>Block end position of 4th axis (ABSMT)</td></tr>
<tr><td>#5011</td><td>Block end position of X axis (ABSOT)</td><td rowspan="8">Workpiece coordinate system</td></tr>
<tr><td>#5012</td><td>Block end position of Y axis (ABSOT)</td></tr>
<tr><td>#5013</td><td>Block end position of Z axis (ABSOT)</td></tr>
<tr><td>#5014</td><td>Block end position of 4th axis (ABSOT)</td></tr>
<tr><td>#5016</td><td>Block end position of X axis (ABSKP)</td><td rowspan="4">allowed</td></tr>
<tr><td>#5017</td><td>Block end position of Y axis (ABSKP)</td></tr>
<tr><td>#5018</td><td>Block end position of Z axis (ABSKP)</td></tr>
<tr><td>#5019</td><td>Block end position of 4th axis (ABSKP)</td></tr>
<tr><td>#5021</td><td>Tool length offset value of X axis</td><td rowspan="2"></td><td rowspan="2">unallowed</td><td></td></tr>
<tr><td>#5022</td><td>Tool length offset value of Y</td><td></td></tr>
</table>

	axis			
#5023	Tool length offset value of Z axis			
#5024	Tool length offset value of 4th 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			

Note 1: ABSIO： The end point coordinates of the last block in workpiece coordinate system.
Note 2: ABSMT： The current machine coordinate system position in machine coordinate system
Note 3: ABSOT： The current coordinate position in workpiece coordinate system
Note 4: ABSKP： The effective position of the skip signal of block G31 in workpiece coordinate system.

3) Workpiece zero offset value and additional zero offset value

Table 4-7-1-4

Variable number	Function
#5201 … #5204	External workpiece zero offset value of 1st axis … External workpiece zero offset value of 4th axis
#5206 … #5209	G54 workpiece zero offset value of 1st axis … G54 workpiece zero offset value of 4th axis
#5211 … #5214	G55 workpiece zero offset value of 1st axis … G55 workpiece zero offset value of 4th axis
#5216 … #5219	G56 workpiece zero offset value of 1st axis … G56 workpiece zero offset value of 4th axis
#5221 … #5224	G57 workpiece zero offset value of 1st axis … G57 workpiece zero offset value of 4th axis
#5226 … #5229	G58 workpiece zero offset value of 1st axis … G58 workpiece zero offset value of 4th axis
#5231 … #5234	G59 workpiece zero offset value of 1st axis … G59 workpiece zero offset value of 4th axis
#7001 … #7004	G54 P1 workpiece zero offset value of 1st axis … G54 P1 workpiece zero offset value of 4th axis
#7006 … #7009	G54 P2 workpiece zero offset value of 1st axis … G54 P2 workpiece zero offset value of 4th axis
#7246 … #7249	G54 P50 workpiece zero offset value of 1st axis … G54 P50 workpiece zero offset value of 4th axis

4. Local variables

The correspondence between address and local variable:

Table 4-7-1-5

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. Except letters G, L, O, N, H and P, all the other 20 letters can assign values for arguments. Each letter from A-B-C-D… to X－Y－Z can assign a value once and the assignment needs not to be performed in alphabetical order. The addresses that assign no values can be omitted.
Note 2: G65 must be specified before any argument is used.

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.
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: $1.7\times10^{-308}\sim1.7\times10^{308}$
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 portions 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. The average time is usually 10ms.

4.7.3 Custom macro call

When G65 is specified, the custom macro specified by address P is called, and the data is transferred to the custom macro body by arguments.
Format:

G65 P □□□□□L□□□□ <argument specification>；

Calling times

Program 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 number of repeats behind L code from 1-9999; 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 such subprograms are called, they can be executed, but the cursor will keep staying at block N65 and the program page displays the main program all the time. (The subprogram can be displayed by setting bit parameter No: 27#4)

Note 3: The macro program cannot be called in DNC mode.

4.7.4 Custom macro function A

1. Format

G65 Hm P#i Q#j R#k ；

m： 01～99 indicate functions of operation instruction or transfer instruction。

#i： Variable name for saving the operation result.

#j： Variable name 1 for operation, or a constant which is expressed 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 on the offset selection.

Table 4-7-4-1

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	Exclusive OR	#i ＝ #j XOR #k
G65	H21	Square root	#i = $\sqrt{\#j}$
G65	H22	Absolute value	#i ＝| #j |
G65	H23	Complement	#i＝#j-trunc(#j ÷ #k)×#k

G65	H26	Compound multiplication and division operation	#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	Arc tangent	#i ＝ ATAN(#j/#k)
G65	H80	Unconditional transfer	GOTO 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
G65	H89	Alarm	

2. Operation instruction

1) Variable assignment: # I = # J

G65 H01 P#I Q#J；

（e.g.）G65 H01 P#101 Q1005； （#101 = 1005）

G65 H01 P#101 Q#110； （#101 = #110）

G65 H01 P#101 Q-#102； （#101 = -#102）

2) Addition: # I = # J+# K

G65 H02 P#I Q#J R#K；

（e.g.）G65 H02 P#101 Q#102 R15； （#101 = #102+15）

3) Subtraction: # I = # J- K

G65 H03 P#I Q#J R# K；

（e.g.）G65 H03 P#101 Q#102 R#103； （#101 = #102-#103）

4）Multiplication：# I = # J×# K

G65 H04 P#I Q#J R#K；

（e.g.）G65 H04 P#101 Q#102 R#103； （#101 = #102×#103）

5）Division：# I = # J÷# K

G65 H05 P#I Q#J R#K；

（e.g.）G65 H05 P#101 Q#102 R#103； （#101 = #102÷#103）

6）Logic addition（OR）：# I = # J.OR. # K

G65 H11 P#I Q#J R#K；

（e.g.）G65 H11 P#101 Q#102 R#103； （#101 = #102.OR. #103）

7）Logic multiplication（AND）：# I = # J.AND. # K

G65 H12 P#I Q#J R#K；

（e.g.）G65 H12 P# 101 Q#102 R#103； （#101 = #102.AND.#103）

8）Exclusive OR：# I = # J.XOR. # K

G65 H13 P#I Q#J R#K；

（e.g.）G65 H13 P#101 Q#102 R#103；（#101 = #102.XOR. #103）

9）Square root：# I = $\sqrt{\#j}$

G65 H21 P#I Q#J；

（e.g.）G65 H21 P#101 Q#102 ；（#101= $\sqrt{\#102}$ ）

10）Absolute value：# I = | # J |

G65 H22 P#I Q#J ；

（e.g.）G65 H22 P#101 Q#102 ；（#101 = | #102 |）

11）Complement：# I = # J－TRUNC(#J/#K)×# K，TRUNC：Removing decimal part

G65 H23 P#I Q#J R#K；

（e.g.）G65 H23 P#101 Q#102 R#103；（#101 = #102- TRUNC (#102/#103)×#103）

12）Compound multiplication and division operation：# I =（# I×# J）÷# K

G65 H26 P#I Q#J R# k；

（e.g.）G65 H26 P#101 Q#102 R#103；（#101 =（#101×# 102）÷#103）

13) Compound square root: # I = $\sqrt{\#j^2+\#k^2}$

G65 H27 P#I Q#J R#K;

(e.g.) G65 H27 P#101 Q#102 R#103; (#101 = $\sqrt{\#102^2+\#103^2}$

14）Sine：# I = # J•SIN（# K）（Unit：°）

G65 H31 P#I Q#J R#K；

（e.g.）G65 H31 P#101 Q#102 R#103；（#101 = #102•SIN（#103））

15）Cosine：# I = # J•COS（# K）（Unit：°）

G65 H32 P#I Q#J R# K；

（e.g.）G65 H32 P#101 Q#102 R#103；（#101 =#102•COS（#103））

16）Tangent：# I = # J•TAN（# K）（Unit：°）

G65 H33 P#I Q#J R# K；

（e.g.）G65 H33 P#101 Q#102 R#103；（#101 = #102•TAN（#103））

17）Arc tangent：# I = ATAN（# J /# K）（Unit：°）

G65 H34 P#I Q#J R# K；

（e.g.）G65 H34 P#101 Q#102 R#103；（#101 =ATAN（#102/#103））

Note 1: The unit of angular variable is degree.
Note 2: If the required Q and R are not specified in operations above, their values are 0 by default.
Note 3: trunc: rounding operation, the decimal portion is abandoned.

3. Transfer instruction

1）Unconditional transfer

G65 H80 Pn； n：Sequence number

（e.g.）G65 H80 P120；（Go to block N120）

2）Conditional transfer 1 #J.EQ.# K (＝)

G65 H81 Pn Q#J R# K； n：Sequence number

(e.g.) G65 H81 P1000 Q#101 R#102；

When # 101 = #102, it goes to block N1000; when #101 ≠ #102, the program is executed in sequence.

3）Conditional transfer 2 #J.NE.# K (≠)

G65 H82 Pn Q#J R# K； n：Sequence number

(e.g.) G65 H82 P1000 Q#101 R#102；

When # 101 ≠ #102, it goes to block N1000; when #101 = #102, the program is executed in sequence.

4）Conditional transfer 3 #J.GT.# K (＞)

G65 H83 Pn Q#J R# K； n：Sequence number

(e.g.) G65 H83 P1000 Q#101 R#102；

When #101 ＞ #102, it goes to block N1000; when #101 ≤ #102, the program is executed in sequence.

5）Conditional transfer 4 #J.LT.# K (＜)

G65 H84 Pn Q#J R# K； n：Sequence number

(e.g.) G65 H84 P1000 Q#101 R#102；

When # 101＜#102, it goes to block N1000; when #101 ≥ #102, the program is executed in sequence.

6）Conditional transfer 5 #J.GE.# K (⩾)

G65 H85 Pn Q#J R# K； n：Sequence number

(e.g.) G65 H85 P1000 Q#101 R#102；

When # 101 ≥ #102, it goes to block N1000; when #101＜#102, the program is executed in sequence.

7）Conditional transfer 6 #J.LE. # K (⩽)

G65 H86 Pn Q#J R# K； n：Sequence number

(e.g.) G65 H86 P1000 Q#101 R#102；

When # 101≤ #102, it goes to N1000; when #101＞#102, the program is executed in sequence.

Note: The sequence number can be specified by variables. Such as G65 H81 P#100 Q#101 R#102； if the conditions are satisfied, it goes to the block of which the number is specified by #100.

4. Logic AND, logic OR and logic NOT instructions

Example:

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, for 3 is 011, and the operation result is #100=7;
G65 H12 P#100 Q#101 Q#102；
The binary expression for 5 is 101, for 3 is 011, and the operation result is #100=1.

5. Macro variable alarm

Example:
G65 H99 P1； Macro variable 3001 alarm
G65 H99 P124； Macro variable 3124 alarm

Example for custom macro

1. Bolt hole cycle

To drill N equal-spaced holes on the circumference of the circle whose center is the reference point (X0，Y0) and radius is R, with an intial angle (A).

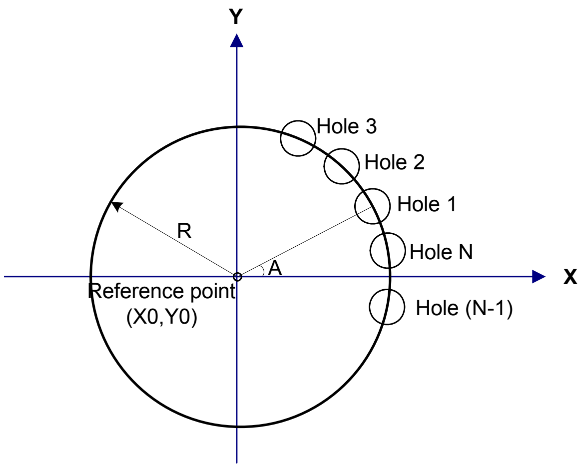


Fig. 4-7-5-1

X0，Y0 is the coordinates of the reference point in bolt hole cycle.
R：Radius, A：Initial angle, N：Number. Parameters above use the following variables:
#500： X coordinate value of the reference point (X0)
#501：Y coordinate value of the reference point (Y0)
#502：Radius (R)
#503：Initial angle (A)
#504： N numbers
If N＞0, the rotation is CCW, and the number is N
If N＜0, the rotation is CW, and the number is N

The variables below are used for the operation in macro.
#100： For the counting of the hole I machining (I)
#101： The final value of the counting(＝┃N┃)(IE)
#102： The angle of hole I (θI)
#103： X coordinate of hole I (Xi)
#104： Y coordinate of hole I (Yi)

The custom macro body can be programmed as follows:

O9010;

N100 G65 H01 P#100 Q0;	I＝0
G65 H22 P#101 Q#504;	IE＝|N|
N200 G65 H04 P#102 Q#100 R360;	
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;	Positioning of hole I
G**;	Hole machining G code
G65 H02 P#100 Q#100 R1;	I＝I＋1
G65 H84 P200 Q#100 R#101;	When I＜IE, go to block N 200, drill IE holes.
M99;	

Example for a program calling the above custom macro body is as follows:

O0010;

G65 H01 P#500 Q100;	X_0=100MM
G65 H01 P#501 Q-200;	Y_0=-200MM
G65 H01 P#502 Q100;	R=100MM
G65 H01 P#503 Q20;	A=20°
G65 H01 P#504 Q12;	N=12 in CCW rotation
G92 X0 Y0 Z0;	
M98 P9010;	Calling the custom macro
G80;	
X0 Y0;	
M30;	

4.7.5　Custom macro function B

1. Arithmetic and logic operation

The operations listed in the following table can be executed on variables. The expressions on the right of the operation characters can contain constants and/or variables constituted by functions or operation characters. The variables #j and #k in the expression can be replaced by constants. The values of the variables on the left can also be assigned by an expression.

Table 4-7-4-2-1 Arithmetic and logic operation

Function	Format	Remarks
Definition	#i = #j	
Addition Subtraction Multiplication Division	#i = #j + #k; #i = #j - #k; #i = #j * #k; #i = #j / #k;	
Sine Arcsine Cosine Arc cosine Tangent Arc tangent	#i = SIN[#j]; #i = ASIN[#j]; #i = COS[#j]; #i = ACOS[#j]; #i = TAN[#j]; #i = ATAN[#j] / [#k];	The angle is specified by degree. 90°30' indicates an angle of 90.5°.
Square root Absolute value Rounding-off Rounding up to an integer Rounding down to an integer Natural logarithm Exponential function	#i = SQRT[#j]; #i = ABS[#j]; #i = ROUND[#j]; #i = FUP[#j]; #i = FIX [#j j]; #i = LN[#j]; #i = EXP[#j];	
OR Exclusive OR AND	#i = #j OR #k; #i = #j XOR #k; #i = #j AND #k;	Logic operation is executed by the binary system.
BCD to BIN Bin to BCD	#i = BIN[#j]; #i = BCD[#j];	Used for switching with PMC signal

Explanation:

(1) **Angle unit**

The angle unit of functions SIN，COS，ASIN，ACOS，TAN and ATAN is degree, e.g., 90°30 ´ indicates an angle of 90.5°.

(2) **ARCSIN #i = ASIN [#j]**

Ranging from -90° to 90°.

When #j is beyond the range from -1 to 1, an alarm occurs.

（3）**ARCCOS #i = ACOS [#j]**

Ranging from 180° to 0°。

When #j is beyond the range from -1 to 1, an alarm occurs.

Variable #j can be replaced by constants.

（4）**ARCTAN #i = ATAN [#j] / [#k]**

Specify the lengths of two sides, separated by a slash（/）.

Ranging from 0° to 360°.

[Example] When #1 = ATAN [-1] / [-1]; is executed, #1=225°.

Variable #j can be replaced by constants.

（5）**Natural logarithm #i = LN [#j]**

When antilog (# j) is 0 or smaller, an alarm occurs.

Variable #j can be replaced by constants.

（6）**Exponential function #i = EXP [#j]**

When the operation result exceeds 99997.453535 (j is about 11.5129), an overflow occurs and an alarm is issued.

(7) **ROUND（rounding-off）function**

The round function rounds off at the first decimal place.

Example:

When #1=ROUND[#2]; is executed where #2 holds 1.2345, the value of variable #1 is 1.0.

（8）**Rounding up and down to a integer**

When the value operation is processed by CNC, if the absolute value of the integer produced by an operation on a number is greater than the absolute value of the original number, such an operation is referred to as rounding up to an integer. If the absolute value of the integer produced by an operation on a number is smaller than the absolute value of the original number, such an operation is referred to as rounding down to an integer. Please be careful when handling negative numbers.

Example：

Suppose that #1=1.2，#2=－1.2。

When #3=FUP[#1] is executed, 2.0 is assigned to #3.

When #3=FIX[#1] is executed, 1.0 is assigned to #3.

When #3=FUP[#2] is executed, -2.0 is assigned to #3.

When #3=FIX[#2] is executed, -1.0 is assigned to #3.

（9）**The abbreviations of the arithmetic and logic instructions.**

When a function is specified in a program, the first two characters of the function name can be used to specify the function. (See table 4-7-4-2-1)

Example:

ROUND→RO

FIX→FI

（10）**Operation sequence**

① Function

② Multiplication and division operation（* / AND）

③ Addition and subtraction operation（+ - OR XOR）

Example)
$$\#1 = \underbrace{\#2 + \underbrace{\#3 * \underbrace{SIN[\#4]}_{\boxed{1}}}_{\boxed{2}}}_{\boxed{3}} ;$$

$\boxed{1}$, $\boxed{2}$ and $\boxed{3}$ indicate the operation sequence.

（11）**Restrictions**

Brackets [,] are used to enclose an expression.

When a divisor of 0 is specified in a division or TAN[90], an alarm is given.

2. Transfer and loop

1) Transfer and loop

In the program, GOTO statement and IF statement are used to change the control flow. There are three types of transfer and loop operations:

Transfer and cycle
- GOTO statement（unconditional transfer）
- IF statement（conditional transfer：IF…THEN…）
- WHILE statement（When… loop is performed）

2) Unconditional transfer

➢ GOTO statement

Transfer to the block with sequence number n. The sequence number can be specified by an expression.

GOTOn; n: Sequence number (1~99999)

Example：

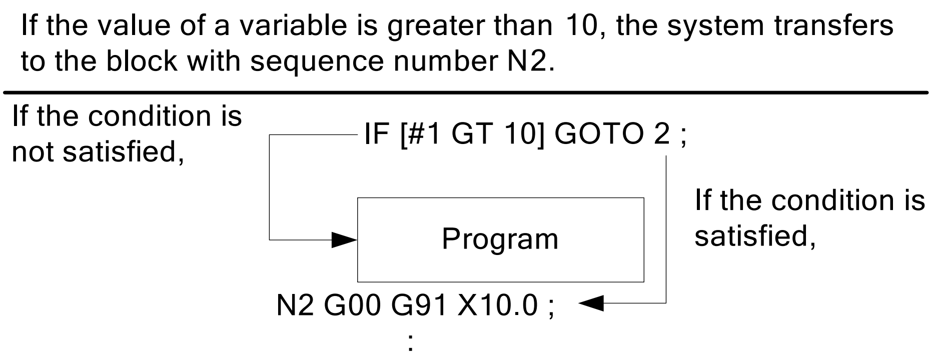
GOTO 1；

GOTO #10；

3) Conditional transfer (IF statement) [<conditional expression>]

IF[<conditional expression**>]GOTO n**

If the specified conditional expression is satisfied, the system transfers to the block with sequence number n; if the specified conditional expression is not satisfied, the next block is executed.



IF[<conditional expression>]THEN

If the conditional expression is satisfied, a predetermined macro statement is executed. Only a single macro statement is executed.

If the values of #1 and #2 are the same，0 is assigned to #3.
IF[#1 EQ #2] THEN #3=0;

Explanation:

➢ Conditional expression

A conditional expression must include an operator, which is inserted between two variables or between a variable and a constant, and must be enclosed with brackets （[，]）. An expression can replace a variable.

➢ Operator

Operators each consists of two letters are used to compare two values to determine whether they are equal or one is greater or smaller than the other one.

Table 4-7-4-2-2 Operators

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

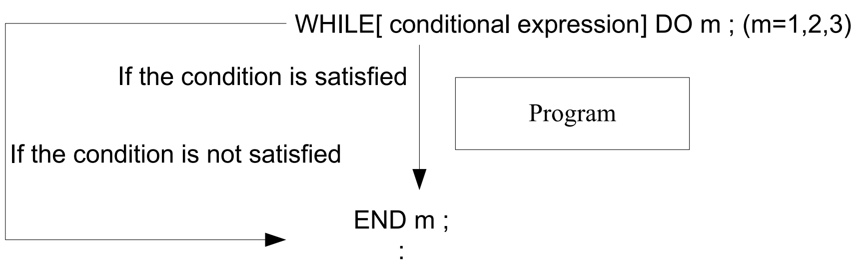
- Typical program

  The program below calculates the sum of numerical value 1 to 10.

```
O9500；
#1=0；                      Initial value of the variable to hold the sum
#2=1；                      Initial value of the variable as an addend
N1 IF[#1 GE 10]GOTO 2；     Transfers to N2 when the addend is greater
                            than or equal to 10
#1=#1+#2；                  Calculation to find the sum
#1=#2+1；                   The next addend
GOTO 1；                    Traverse to N1
N2 M30；                    Program end
```

4) Loop (WHILE statement)

Specify a conditional expression behind WHILE, when the specified condition is satisfied, the program from DO to END is executed, otherwise, program execution proceeds to the block after END.



When the specified condition is satisfied, the program from DO to END is executed. Otherwise, program execution proceeds to the block after END. This kind of instruction format is applicable to IF statement. A number after DO and a number after END are the identification numbers for specifying the range of execution. The identification numbers are 1, 2 and 3. If numbers other than 1, 2 and 3 are used, an alarm occurs.

Explanation:

- Nestling

  The identification numbers (1 to 3) in the loop from D0 to END can be used repeatedly as required. However, when a program includes crossing repetition loop (overlapped DO ranges), an alarm occurs.

1.The identification numbers (1 to 3) can be used as many times as required.

```
WHILE[ … ] DO 1;
    Program
END 1 ;
    :
WHILE[ … ] DO 1;
    Processing
END 1 ;
    :
```

2. The ranges of DO cannot overlap

```
WHILE[ … ] DO 1;
    Program
WHILE[ … ] DO 2;
    :
END 1 ;
    Program
END 2 ;
```

3. DO loops can be nested to 3 levels

```
WHILE[ … ] DO 1;
    :
  WHILE[ … ] DO 2;
      :
    WHILE[ … ] DO 3;
        Program
    END 3 ;
  END 2 ;
END 1 ;
```

4. The control can be transferred to the outside of a loop.

```
WHILE[ … ] DO 1;
IF[…] GOTO n ;
END 1 ;
Nn
```

5. Transfer cannot enter the loop area.

```
IF [⋯] GOTO n ;
    :
WHILE [⋯] DO 1 ;
Nn⋯ ;
END 1 ;
```

Explanation:

➢ Infinite loop

When DO is specified without specifying WHILE statement, an infinite loop from DO to END is produced.

- Processing time

When a transfer to a sequence number in GOTO statement occurs, the sequence number is searched for. Processing in the reverse direction is longer than the one in the forward direction. The processing time can be reduced by using WHILE statement for repetition.

- Undefined variables

In the conditional expression using EQ or NE, <vacant> and zero have different affects. In the other conditional expressions, <vacant> is taken as 0.

- Typical program

The program below calculates the sum of numbers 1 to 10.

```
O0001 ；
#1=0；
#2=1；
WHILE [#1 LE 9] DO 1;
#1=#1+#2;
#1=#2+#1;
END 1;
M30;
```

Precautions:

- When a macro program is called by G65, and M, S, T, D and F are used for transferring variables, only positive integers can be transferred. This limitation does not apply to other letters.
- The line number N code cannot be in the same line with WHILE/DO/END, or the loop is ineffective.
- Loop and skip instructions cannot be used in DNC mode.
- A GOTO statement starts searching at the beginning of the program and skips when the first corresponding line number is retrieved. Try not to use the same N code in one program.
- When the variable number is expressed by a decimal fraction, the system will remove the decimal part with carry ignored.
- The values of local variables are retained before the main program ends. They are common to each subprogram.

CHAPTER 5 MISCELLANEOUS FUNCTION M CODE

The M codes of this machine available for users are listed as follows:

Table 5-1

	M code	Function
M codes used for control program	M30	The program ends and returns to the program beginning, the machining number increases by 1.
	M02	The program ends and returns to the program beginning, the machining number increases by 1.
	M98	Subprogram calling
	M99	Subprogram ends and returns/execution is repeated
	M00	Program dwell
	M01	Program optional dwell
M codes controlled by PLC	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 clamp
	M18	Spindle orientation cancel
	M19	Spindle orientation
	M20	Spindle neutral gear instruction
	M21	Tool search instruction in retraction
	M22	Tool search instruction during a new tool catching
	M23	Tool magazine to spindle instruction
	M24	Tool magazine retraction instruction
	M26	Chip flushing water valve ON
	M27	Chip flushing water valve OFF
	M28	Rigid taping cancel
	M29	Rigid taping
	M35	Helical chip remover ON
	M36	Helical chip remover OFF
	M44	Spindle blowing ON
	M45	Spindle blowing OFF
	M50	Auto tool change start
	M51	Auto tool change finish
	M53	Tool judging after tool change
	M55	Tool judging on the spindle

When a move instruction and miscellaneous function are specified in the same block, the instructions are executed in either 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 execution sequence depends on the machine tool builder's specification. Refer to the manual provided by the machine builder for details.

When a numerical value is specified behind address M, code signal and strobe signal are sent to the machine. The machine uses these signals to turn on/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 setting bit parameter No.33#7. Some M codes cannot be specified simultaneously because of the restrictions of the mechanical operation. See the machine manual provided by the tool builder for the mechanical operation restrictions on simultaneous specification for M codes in one block.

5.1 M codes controlled by PLC

If an M code controlled by PLC is in the same block with a move instruction, they are executed simultaneously.

5.1.1 CCW/CW rotation instructions（M03, M04）

Instruction: M03（M04） Sx x x；

Explanation: Viewed from the negative direction to the positive direction along Z axis, that the spindle is rotated counterclockwise (CCW) is defined as CCW rotation, vice versa, that the spindle is rotated clockwise (CW) is defined as CW rotation. The direction of moving forward to the workpiece by the right-hand thread is defined as the positive direction, and the direction of departing from the workpiece by the right-hand thread is defined as the negative direction.

Sx x x specifies the spindle speed, or the current gear in gear control mode.

Unit: revolution per minute (r/min)

When it is controlled by a frequency converter, Sx x x specifies the actual speed. e.g. S1000 specifies the spindle to rotate at a speed of 1000r/min.

5.1.2 M05 Spindle stop (M05)

Instruction: M05. When M05 is executed in auto mode, the spindle is stopped, but the speed specified by S instruction is retained. The deceleration at spindle stop is set by the machine builder. It is usually done by energy consumption brake.

5.1.3 Cooling ON/OFF (M08, M09)

Instruction: M8（M9） It is used to control the ON/OFF operation of the cooling pump. If the miscellaneous functions are locked in auto mode, this instruction is not executed.

5.1.4 A axis release/clamping（M10, M11）

Instruction: M10（M11） It is used for A axis release and clamping.

5.1.5 Tool control release/clamping（M16, M17）

Instruction: M16（M17） It is used for tool release and clamping.

5.1.6 Spindle orientation（M18, M19）

Instruction: M18 for cancelling the spindle orientation; M19 orients the spindle, which is used for the positioning of tool change.

5.1.7 Tool search instruction (M21, M22)

Instruction: M21, the instruction used to search a tool in retraction; M22，the instruction used to search a tool when catching a new one.

5.1.8 Tool retraction instruction （M23, M24）

Instruction: M23, the instruction to move the magazine to the spindle; M24, the instruction to move the magazine back to its normal position.

5.1.9 Rigid taping （M28，M29）

Instruction: M28, for cancelling the rigid taping; M29, for specifying the rigid taping.

5.1.10 Helical chip remover ON/OFF（M35, M36）

Instruction: M35（M36） It is used to control the ON/OFF operation of the helical chip remover.

5.1.11 Chip flushing water valve ON/OFF（M26, M27）

Instruction: M26, for turning on the valve; M27, for turning off the valve.

5.1.12 Spindle blowing ON/OFF (M44, M45)

Instruction: M44（M45）controls the ON/OFF of the spindle blowing.

5.1.13 Auto tool change START/END (M50, M51)

Instruction: M50（M51）controls the START/END of auto tool change.

5.1.14 Tool judging after tool change（M53）

Instruction: M53, used for checking whether the changed tool is correct.

5.1.15 Tool judging on the spindle (M55)

Instruction: M55, used for judging whether there is a tool on the spindle.

5.2 M codes used by control program

M codes used by a program are divided into main program type and macro type. If an M code used by a program and a move instruction are in a same block, the move instruction is executed prior to the M code.

Note 1: Codes M00, M01, M02, M06, M30, M98 and M99 cannot be specified together with other M codes, or an alarm is issued. When these codes are in the same block with other non-M instructions, the non-M instructions are executed prior to the M codes.
Note 2: This kind of M codes include the codes that direct the CNC to perform the internal operation in addition to sending the M codes themselves to the machine, e.g. the M code to disable the block prereading function. Moreover, the codes to send the M codes themselves to the machine (without performing the internal operation) can be specified in the same block.

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 are stopped. Meanwhile, the workpiece machined number increases by 1. Whether the control returns to the beginning of the program after M30 is executed is set by bit parameter N0：33#4; whether the control returns to the beginning of the program after M02 is executed is set by bit parameter N0：33#2. If M02 and M03 are in a subprogram, then the control returns to the program calling the subprogram after they are executed and proceeds to the following blocks.

5.2.2 Program dwell (M00)

In Auto running, the automatic operation pauses after a block containing M00 is executed. Meanwhile, the previous modal information will be saved. The automatic operation is continued by pressing Cycle Start key, which is equivalent to pressing down key Feed Hold.

5.2.3 Program optional stop (M01)

Automatic operation is stopped optionally after a block containing M01 is executed. If the “Optional Stop” switch is set to ON, M01 is equivalent to M00; if the “Optional Stop” switch is set to OFF, M01 is ineffective. See *OPERATION MANUAL* for its operation.

5.2.4 Subprogram calling（M98）

M98 is used to call a subprogram in a main program. Its format is as follows:

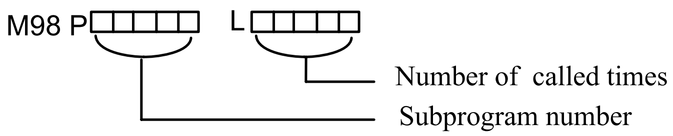


Fig. 5-2-4-1

5.2.5 Program end and return (M99)

1. In auto mode, if M99 is executed at the end of the main program, the control returns to the program beginning to continue automatic operation. Meanwhile, the following blocks are not to be executed, and the number of the machined workpieces is not accumulated.
2. If M99 is executed at the end of a subprogram, the control returns to the main program and proceeds to the next block following the subprogram block.
3. In DNC mode, M99 is processes as M30, thus the cursor keeps staying at the end of the program.

CHAPTER 6 SPINDLE FUNCTION S CODES

By using an S code and the numerical values behind it, the code signal can be converted to the analog signal and then sent to the machine, for controlling 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 the analog voltage which is specified by address S and the numerical values behind. See *OPERATION* in the manual for details.

Format: S_

Explanation:

1. Only one S code can be specified in a block.
2. The spindle speed is specified directly by address S and a numerical value behind it. Unit: r/min. e.g. For M3 S300, it means the spindle is rotated at a speed of 300 r/min.
3. If a move instruction and an S code are specified in the same block, they are executed simultaneously.
4. The spindle speed is controlled by an S code followed by a numerical value.

6.2 Spindle switch value control

When the bit parameter NO.1#2 SPT=1, the spindle speed is controlled by the switch value, which consists of an address S and a two-digit number behind it.

Three mechanical gears for the spindle are provided when the spindle speed is controlled by the switch value. For the correspondence between S codes and spindle speed as well as the number of spindle gears, please see the manual provided by the machine tool builder.

Format: S01（S1）；

S02（S2）；

S03（S3）；

Explanation:

1. There are 8 gears in the software at present, and 3 gears in the ladder diagram. When S codes beyond the codes above are specified, the system displays “Miscellaneous function being executed”.
2. If a four-digit number is specified behind S, the latter two digits are effective.

6.3 Constant surface speed control G96/G97

Format:

Constant surface speed control instruction: G96 S_ Surface speed（m/min or inch/min）

Constant surface speed control cancel instruction: G97 S_ Spindle speed (r/min)

Constant surface speed controlled axis instruction: G96 Pn_ P1 X: axis；P2: Y axis；P3: Z axis；P4: 4th axis

Clamp of max. spindle speed: G92 S_ S specifies the max. spindle speed（r/min）

Function: The number following S is used to specify the surface speed (relative speed between 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 specified S value 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 a workpiece coordinate system, then the coordinate value at the center of the rotary axis becomes zero.

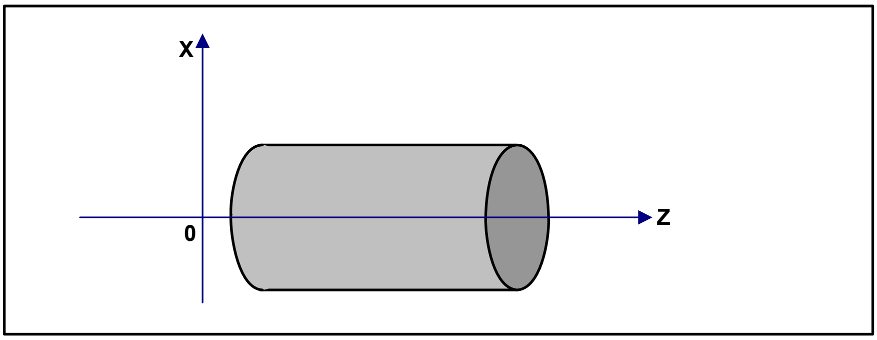


Fig. 6-3-1 Workpiece coordinate system for constant surface speed control

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, and the maximum spindle speed is not yet set, the S in G96 is regarded as zero till M3 or M4 appears in the program.

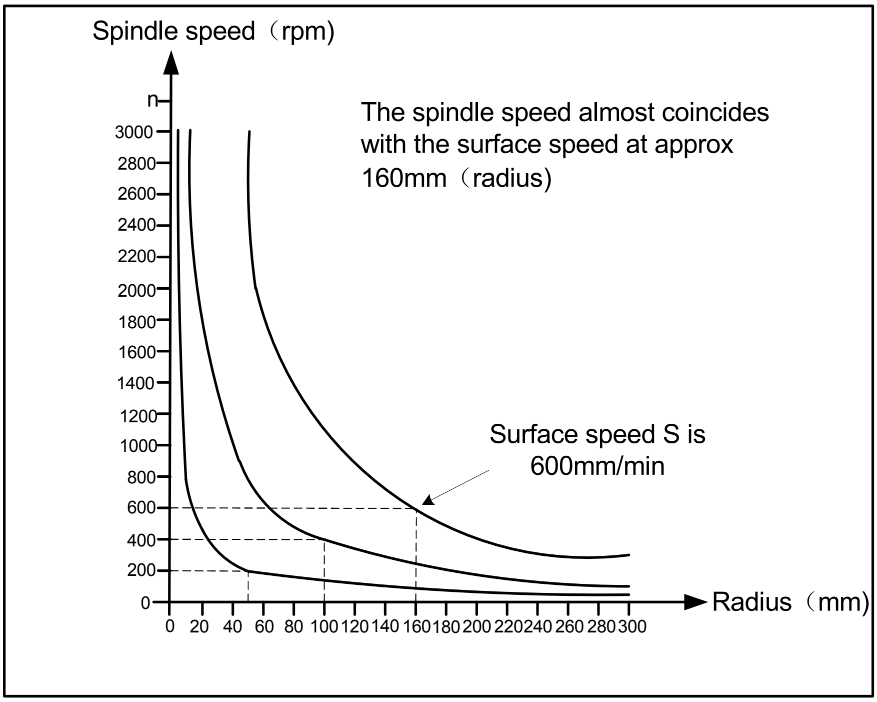


Fig. 6-3-2 Relation between workpiece radius, spindle speed and surface speed

5. Surface speed specified in G96 mode:

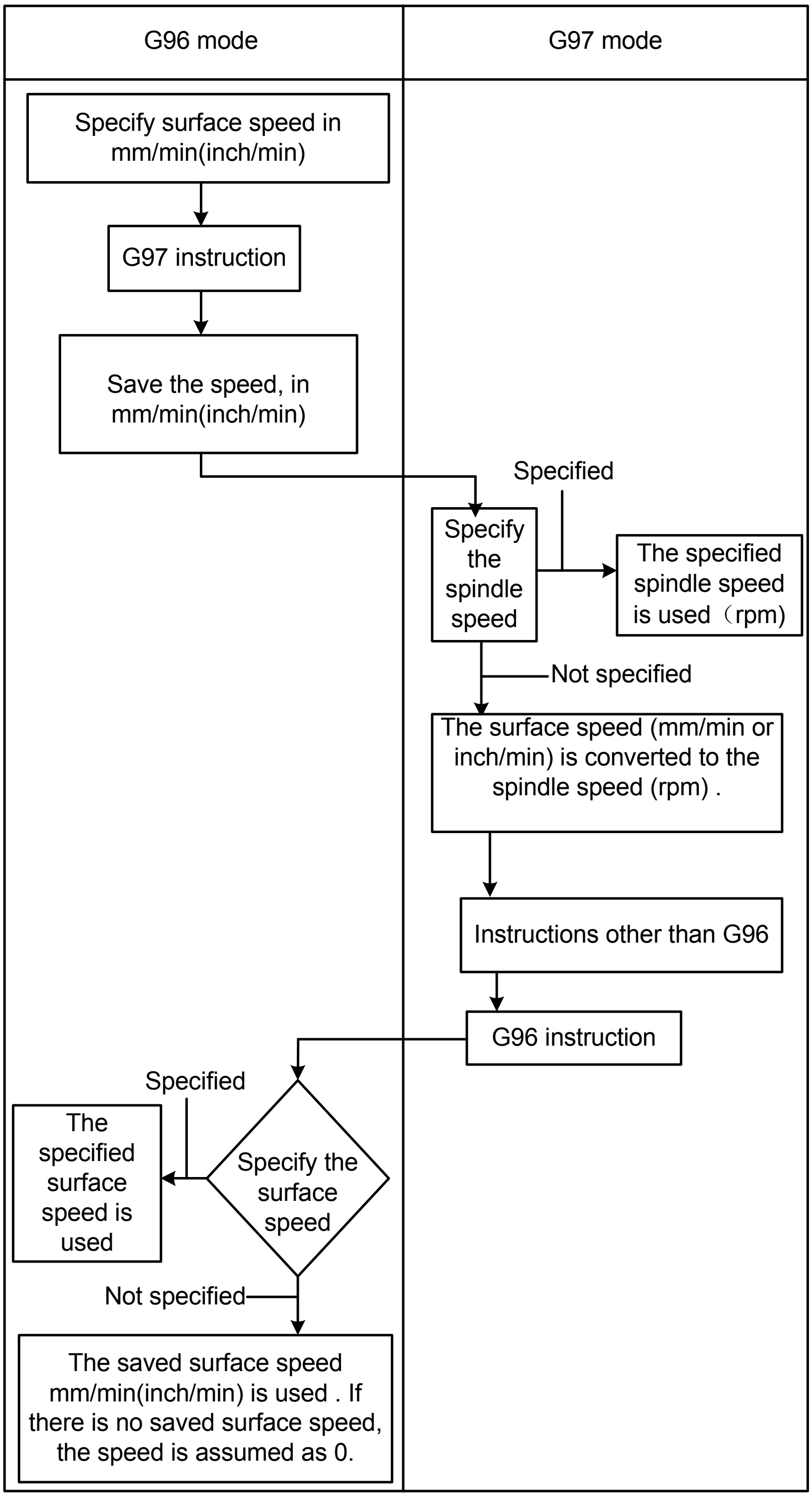


Fig. 6-3-3

Restrictions：

1. Because the response problem in the servo system may not be considered when the spindle speed changes, and the constant surface speed is also effective during threading, it is recommended to cancel the constant surface speed by G97 before threading.
2. In a rapid 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 rapid traverse block, on the condition that cutting is not performed during rapid traverse. Therefore, the constant surface cutting speed is not used.

CHAPTER 7 FEED FUNCTION F CODE

The feed functions are used to control the feedrate of the tool. The functions and control modes are as follows:

7.1 Rapid traverse

G00 instruction is used for rapid positioning. The traverse speed is set by data parameters P88~P92. An override can be applied to the traverse speed by the OVERRIDE adjusting keys on the operator panel, which are shown as follows:

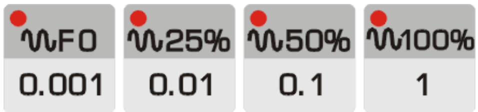


Fig. 7-1-1 Keys for rapid traverse override

F0 is set by data parameter P93.

The acceleration of rapid positioning (G0) can be set by data parameters P105~123. It can be properly set depending on the machine and the motor response characteristics.

Note: In a block containing G00, the feedrate instruction F is invalid even if it is specified. The system performs positioning at the speed specified by G0 instead.

7.2 Cutting feedrate

The tool feedrates in linear interpolation (G01) and circular interpolation(G02，G03) are specified with the numbers after F code in mm/min. The tool is moved by the programmed feedrate. An override can be applied to the cutting feedrate using the override keys on the operator panel (Override range： 0％～ 200%).

In order to prevent mechanical vibration, acceleration/deceleration is automatically applied at the beginning and the end of the tool movement respectively. The acceleration can be set by data parameters P125~P128.

The minimum cutting feedrate is set by data parameter P96, and the maximum cutting feedrate in the forecast mode is set by P97. If it is smaller than the lower limit, the cutting feedrate is clamped to the lower limit.

The cutting feedrate in auto mode at power-on is set by data parameter P87.

The cutting feedrate can be specified by the following two types:

A) Feed per minute (G94): it is used to specify the feed amount per minute after F code.

B) Feed per revolution (G95): it is used to specify the feed amount per revolution after F code.

Note: When the cutting feedrate is specified with F, the system displays the value as an integer. If the input value is not an integer, the system displays it as an integer obtained after rounding it off, but still performs processing using the actual input value. If the pitch is specified by F, the system displays one decimal point for the input value, but still performs processing using the actual input value.

7.2.1 Feed per minute (G94)

Format: G94 F_

Function: It specifies the tool feed amount per minute. Unit：mm/min or inch/min.

Explanation:

1. After G94 is specified (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 remains effective till G95 is specified. The default at power-on is feed per minute mode.
3. An override from 0% to 200% can be applied to feed per minute with the override keys or band switch on the operator panel.

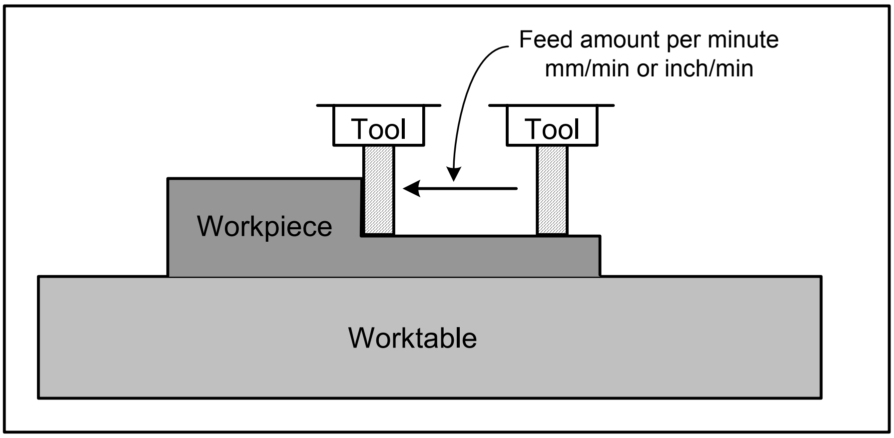


Fig. 7-2-1-1 Feed per minute

Restriction: Feed per minute mode cannot be applied to some instructions such as threading.

7.2.2 Feed per revolution (G95)

Format: G95 F_

Function: Feed amount per revolution. Unit: mm/r or inch/r

Explanation:

1. This function is unavailable until a spindle encoder is installed on the machine.
2. After specifying G95 (feed per revolution mode), the feed amount of the tool per revolution is directly specified by a number after F.
3. G95 is a modal code. Once specified, it keeps effective till G94 is specified. The default feedrate per revolution during initialization is 0.
4. An override from 0% to 200% can be applied to feed per revolution with the override keys or band switch on the operator panel.

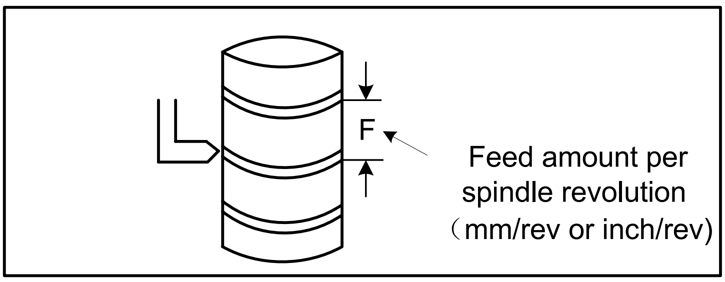


Fig. 7-2-2-1 Feed per revolution

Note: When the spindle speed is low, feedrate fluctuation may occur. The lower the spindle speed is, the more frequently the feedrate fluctuation occurs.

7.3 Tangential speed control

The cutting feed usually controls the speed in the tangential direction of the contour path to make it reach the specified speed value.

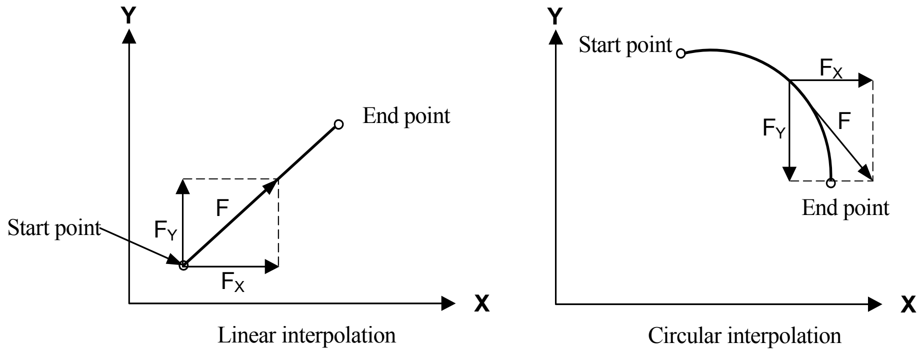


Fig. 7-3-1

F：The speed along the tangent $F=\sqrt{F_X^2+F_Y^2+F_Z^2}$

Fx： The speed along X axis

Fy： The speed along Y axis

Fz： The speed along Z axis

7.4 Keys for feedrate override

The feedrate in MANUAL mode and AUTO mode can be overridden by the override keys on the operator panel. The override ranges from 0～200%(21 gears with 10% per gear). In AUTO mode, if the feedrate override is adjusted to 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 system enables the motor to perform acceleration/deceleration control at the beginning and the end of the movement, which thus obtains a stable start and stop. In addition, the automatic acceleration/deceleration can also be applied when the moving speed is changed, the speed thus can be changed steadily. Therefore, the acceleration/deceleration needs not to be considered during programming.

Rapid traverse: Pre-acceleration/deceleration （0：linear type；1：S type）
Post acceleration/deceleration（0：linear type；1：exponential type）
Cutting feed: Pre-acceleration/deceleration （0：linear type；1：S type）
Post acceleration/deceleration（0：linear type；1：exponential type）
MANUAL feed: Post acceleration/deceleration（0：linear type；1：exponential type）

(Set the common time constant for each axis by parameters)

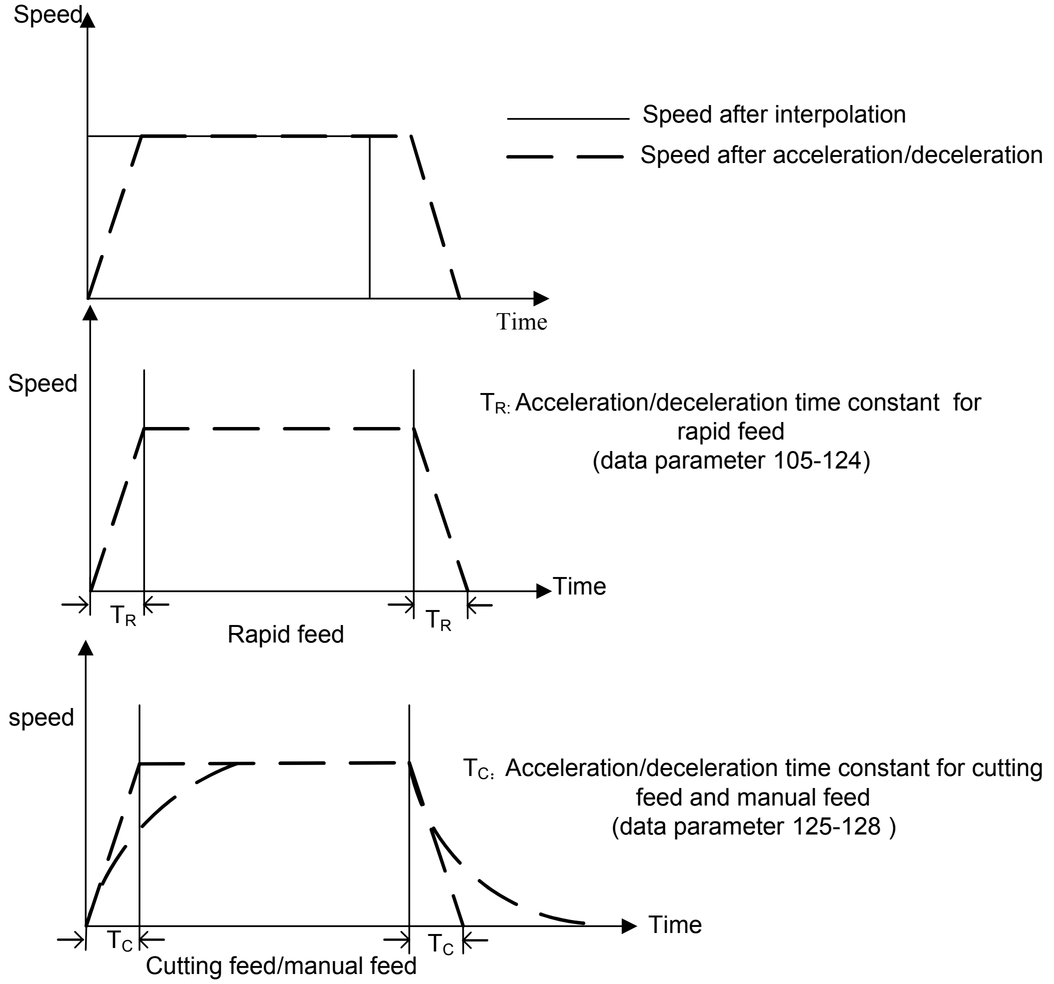


Fig. 7-5-1

7.6 Acceleration/deceleration at the corner in a block

Example: If a block containing only Y movement is followed by a block containing only X movement, the latter X block accelerates as the former Y block decelerates. The tool path is as follows:

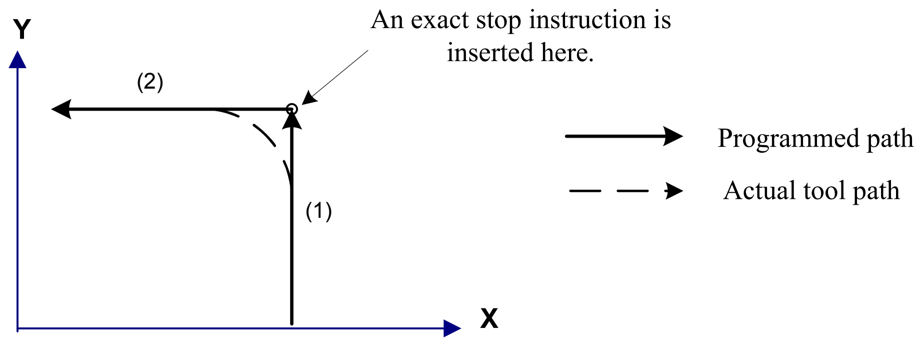


Fig. 7-6-1

If an exact stop instruction is inserted, the tool is moved along the real line as in the above figure by the program, otherwise the bigger the cutting feedrate is, or the longer the time constant of the acceleration/deceleration is, the bigger the arc at the corner is. For circular instruction, the actual arc radius of the tool path is smaller than the arc radius specified by the program. The mechanical system permitting, reduce the acceleration/deceleration time constant as far as possible to minimize the error at the corner.

CHAPTER 8 TOOL FUNCTION

8.1 Tool function

By specifying a numerical value (up to 8 digits) following address T, the tools on the machine can be selected.

Only one T code can be specified in a block by principle. However, if no alarm occurs when a block contains two or more instructions of the same group via setting, the last T code takes effect. Refer to the manual provided by the tool machine builder for the digits after address T and the corresponding machine operation of T code.

When a movement instruction and a T code are specified in the same block, the instructions are executed simultaneously.

When the T code and tool change instruction are in the same block, the T code is executed before tool change instruction. If they are not in the same block, M06 executes the T code specified by the last program.

Such as the program below:

```
O00010;
N10 T2M6;        Spindle tool number is T2
N20 M6T3;        Spindle tool number is T3
N30 T4;          Spindle tool number is T3
N40 M6;          Spindle tool number is T4
N50 T5;          Spindle tool number is T4
N60 M30
%
```

After the tool change, the spindle tool number is T4.

BOOK Ⅱ OPERATION

Book 2 Operation

CHAPTER 1 OPERATION PANEL

1.1 Panel layout

An integrated operator panel is applied to **GSK218MC** CNC system, while separate-type structure is adopted for **GSK 218MC-H** and **GSK218MC-V**. The layout of the panel consists of LCD area, editing keyboard area, soft key function area and machine control area. See the figures below:

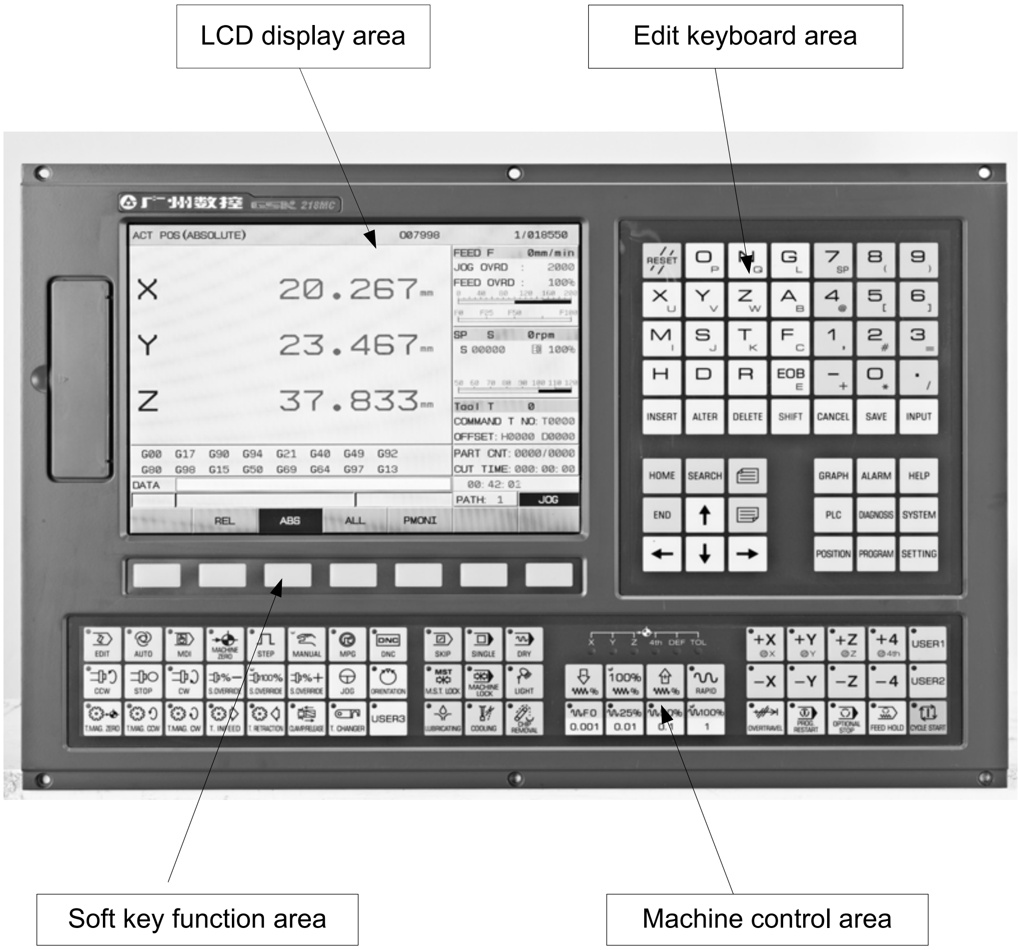


Fig. 1-1-1 Panel of GSK218MC

Fig. 1-1-2 Panel of GSK218MC-H

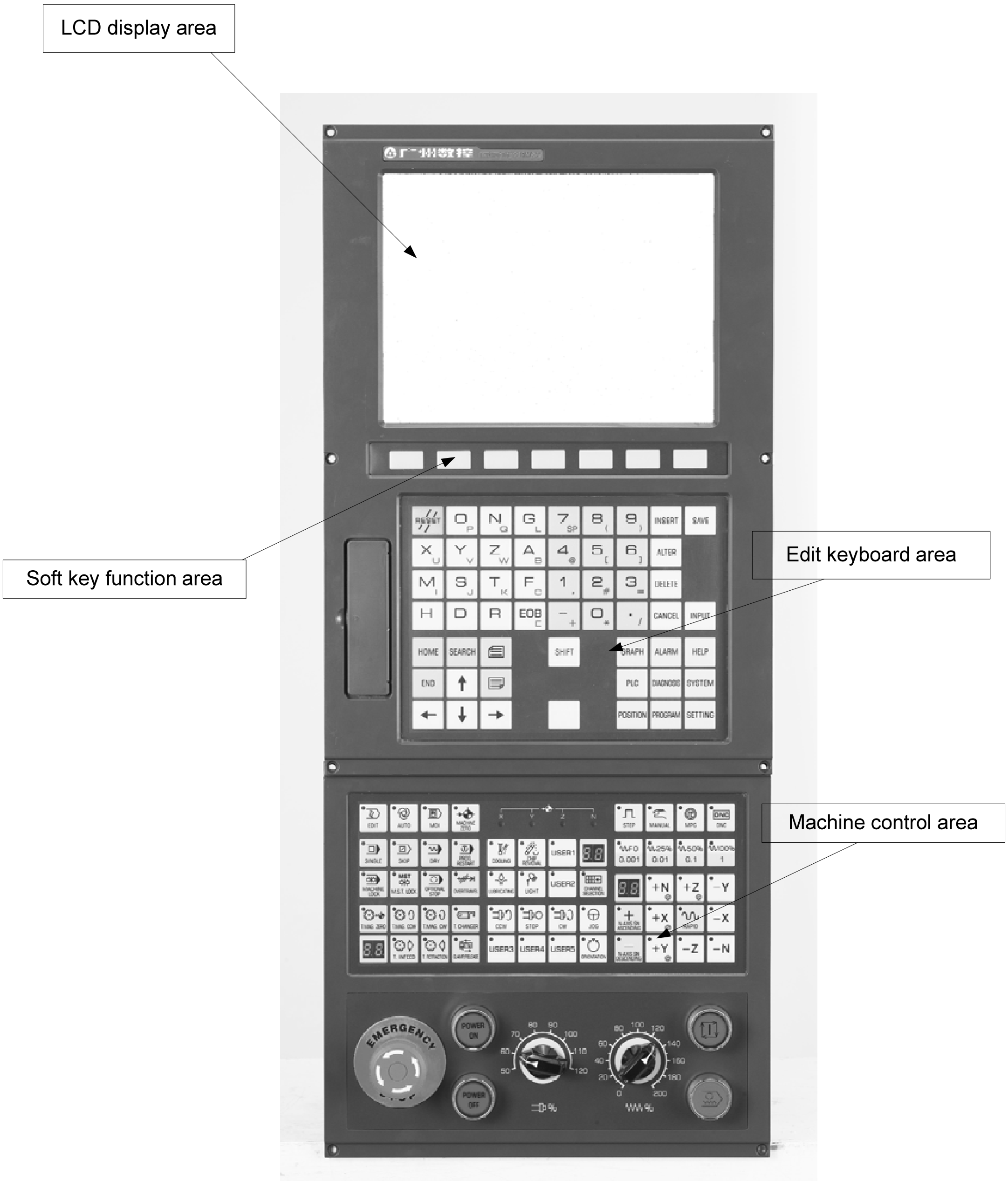


Fig. 1-1-3 Panel of GSK218MC-V

1.2 Explanation for panel functions

1.2.1 LCD display area

GSK 218MC and **GSK 218MC-V** systems are employed with 10.4 inch color displays with resolution of 800×600, and **GSK 218MC-H** system is employed with an 8.4 inch color display with resolution of 800×600.

1.2.2 Editing keyboard area

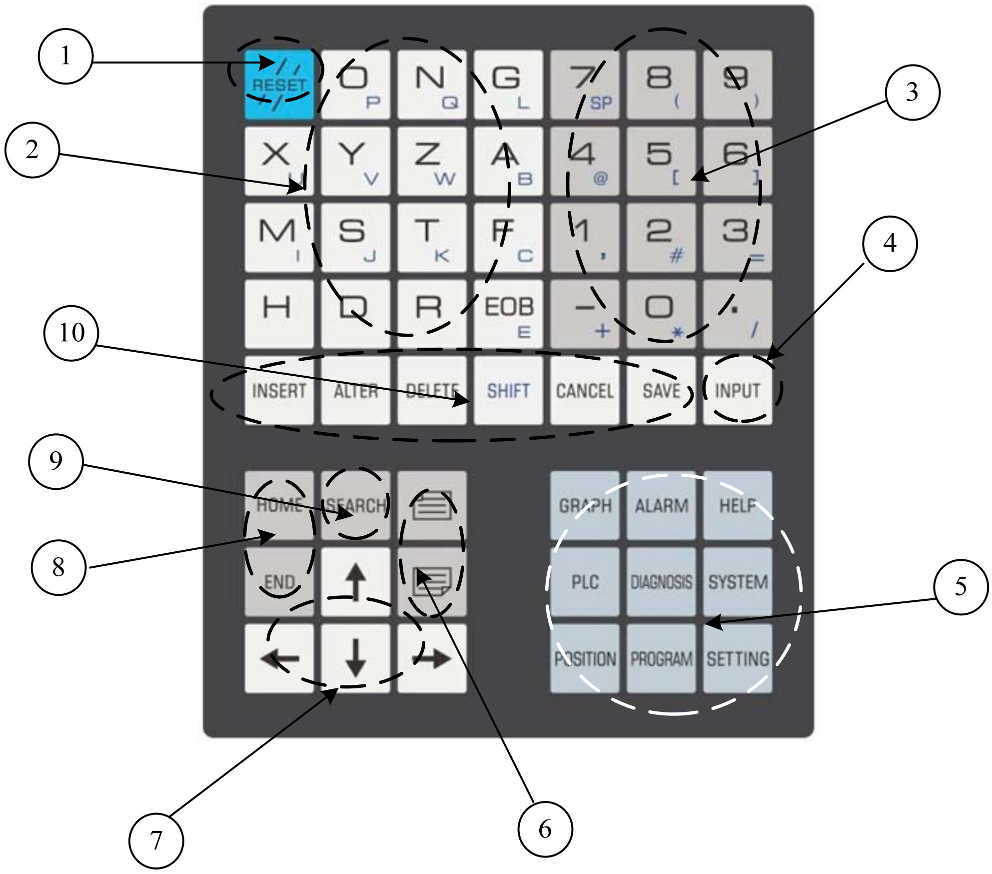


Fig. 1-2-2-1 Editing keyboard area of 218MC and 218MC-H

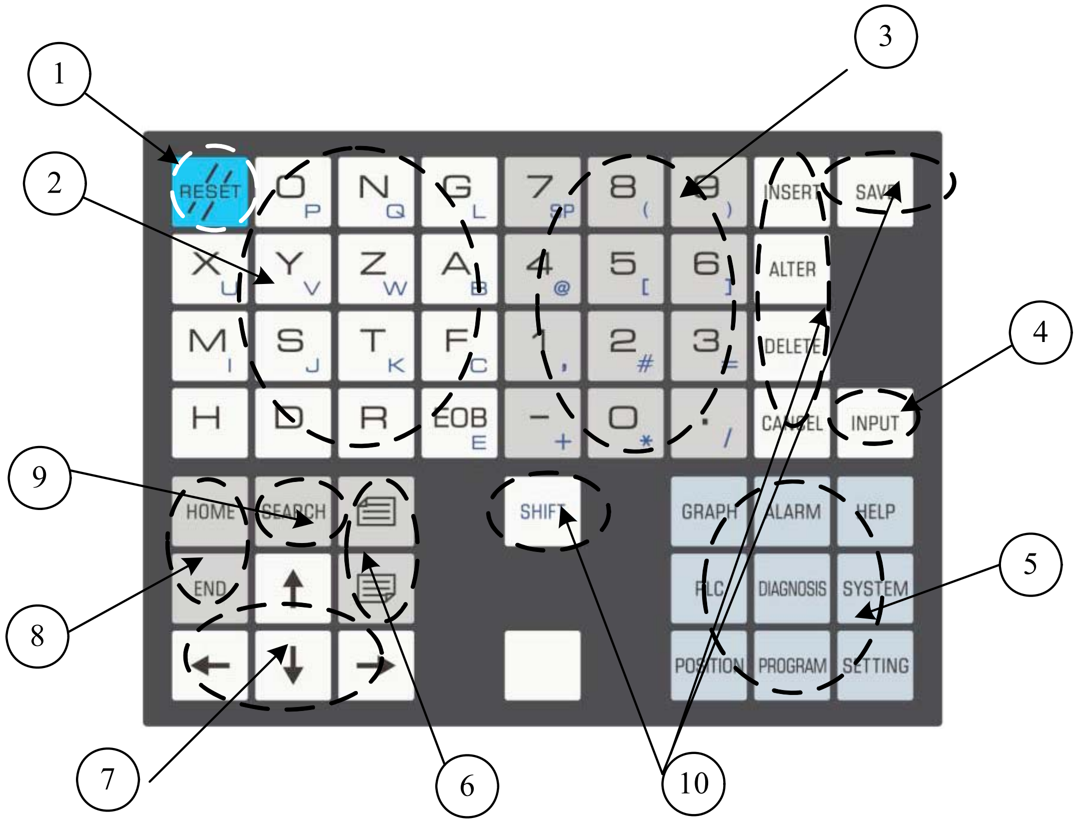


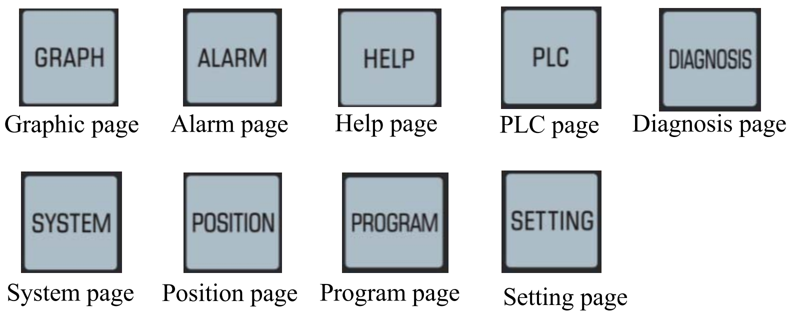
Fig. 1-2-2-1 Editing keyboard area of GSK218MC-V

The functions of the keys on the editing keyboard area are divided into 10 small areas, which are explained as follows:

No.	Designation	Explanation
1	Reset key	For system reset, feed and output stop
2	Address key	For inputting addresses in MDI mode
3	Number key	For inputting numerical values in MDI mode
4	Input key	For Inputting numerical values, addresses or data into the buffer area; confirming the operation result
5	Screen operation key	By pressing any of the keys, the corresponding page is entered. See chapter 3 for details.
6	Page key	For page switching in the same display mode, and page down/up in the program
7	Cursor key	For moving the cursor in different directions
8	Editing key	For moving the cursor to the beginning or the end of a block or a program.
9	Search key	For searching data and addresses to view and modify
10	Editing key	For inserting, modifying or deleting a program or a block during programming, by using compound keys.

1.2.3 Screen operation keys

There are 8 display keys for operation pages and 1 display key for the help page on the panel in this system. See the figure below:



Designation	Explanation	Remarks
Graphic page	Press this key to enter graphic page	Subpages for graphic parameters and graphic display can be viewed by switching corresponding soft keys. The center, size and ratio for the graph are set using graphic parameters
Alarm page	Press this key to enter alarm page	Subpages for a variety of alarm message can be viewed by switching corresponding soft keys.
Help page	Press this key to enter help page	Help message about the system can be viewed in this page by switching corresponding soft keys.

PLC page	Press this key to enter PLC page	The version of the PLC ladder and the configuration of system I/O can be viewed on this page, and the modification for PLC ladder is available in MDI mode.
Diagnosis page	Press this key to enter diagnosis page	The states of I/O signals on the system side can be viewed in this page by switching corresponding soft keys
System page	Press this key to enter system page	Subpages for tool offsets, parameters, macro variables and screw pitch can be displayed by switching corresponding soft keys
Position page	Press this key to enter position page	Subpages for relative coordinates, absolute coordinates and all coordinates of the current point and PLC can be displayed by switching corresponding soft keys
Pgrogram page	Press this key to enter program page	Subpages for programs, MDI, current/mode, current/time, and program directory can be displayed by switching corresponding soft keys. Program names in different pages can be viewed by pressing page keys in directory subpage.
Setting page	Press this key to enter setting page	Four subpages in total. The subpages for setting, workpiece coordinate, data and password setting can be displayed by switching corresponding soft keys.

Note: The page switch above can also be done by pressing corresponding function keys repeatedly after bit parameters NO:25#0～25#7, NO:26#6～26#7 are set. Refer to CHAPTER 3 in this manual for the explanation for each page.

1.2.4 Machine control area of GSK218MC

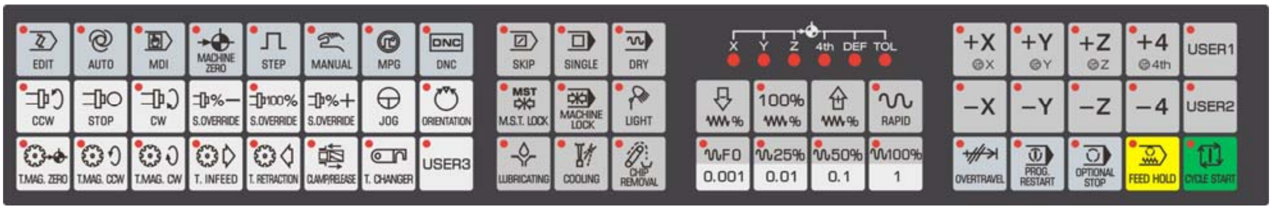
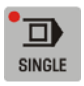
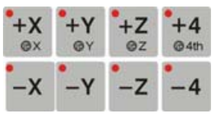
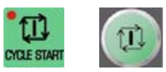


Fig. 1-2-4-1 Machine control area of GSK218MC

Keys	Designation	Explanation	Remarks and operation explanation
	Edit mode key	To enter edit mode	Switching to Edit mode in Auto mode, MDI mode and DNC mode. System decelerates to stop after current block is executed
	Auto mode key	To enter auto mode	In this mode, program in internal memory is selected
	MDI mode key	To enter MDI mode	Switching to MDI mode in Auto mode, system decelerates to stop after current block is executed
	Machine zero mode key	To enter machine zero mode	Switching to Machine zero mode in Auto mode. System immediately decelerates to stop
	Step mode key	To enter step mode	Switching to Step mode in Auto mode, system immediately decelerates to stop
	Manual mode key	To enter manual mode	Switching to Manual Mode in Auto mode, system immediately decelerates to stop
	MPG mode key	To enter MPG mode	Switching to MPG mode in Auto mode, system immediately decelerates to stop
	DNC mode key	To enter DNC mode	Switching to DNC mode in Auto mode, system decelerates to stop after current block is executed
	Block skip key	For a block preceding with "/" sign. If it is on, the indicator lights up And the block is skipped.	Auto mode, MDI mode, DNC mode
	Single block key	For switching the execution between single block and blocks. If it is on, the indicator Lights up.	Auto mode, MDI mode, DNC mode

Keys	Designation	Explanation	Remarks and operation explanation
DRY	Dry run switch	The indicator lights up if dry run is valid.	Auto mode, MDI mode, DNC mode
MST M.S.T. LOCK	M.S.T. lock switch	M.S.T. function output is invalid if the indicator for M.S.T. function lock lights up.	Auto mode, MDI mode, DNC mode
MACHINE LOCK	Machine lock switch	The indicator lights up if it is on, and the axis movement output is invalid.	Auto mode, MDI mode, Machine zero, MPG mode, Step mode, MANUAL mode, DNC mode
LIGHT	Machine working light switch	Machine working light ON/OFF	Any mode
LUBRICATING	Lubricant oil switch	Machine lubricant ON/OFF	Any mode
COOLING	Coolant switch	Coolant ON/OFF	Any mode
CHIP REMOVAL	Chip removal switch	Chip removal ON/OFF	Any mode
CCW STOP CW	Spindle control keys	Spindle CCW Spindle stop Spindle CW	MPG mode, step mode, manual mode
%− S.OVERRIDE 100% S.OVERRIDE %+ S.OVERRIDE	Spindle override keys	Spindle speed adjustment (spindle speed analog control valid)	Any mode
JOG	Spindle JOG switch	Spindle JOG ON/OFF	Manual mode, Step mode, MPG mode
ORIENTATION	Spindle exact stop key	Spindle exact stop ON/OFF	Manual mode, Step mode, MPG mode
T.MAG. ZERO T.MAG. CCW T.MAG. CW T. CHANGER T. INFEED T. RETRACTION	Tool magazine operation keys	Tool magazine operation ON/OFF	Manual mode
CLAMP/RELEASE	Manual tool release/clamp key	Manual tool release/ clamp ON/OFF	Manual mode

Keys	Designation	Explanation	Remarks and operation explanation
	Manual tool change key	Manual tool change	MANUAL mode
	Overtravel release key	An alarm occurs if the hard limit is reached. Press this key with its indicator lighting up to move the machine reversely till the indicator goes off.	MANUAL mode, MGP mode
	Program restart key	For exiting the running program or restoring to the last machining state before a sudden power loss	Auto mode (the distance to go is the straight-line distance from the current point to the break point)
	Optional stop ON/OFF key	Whether the operation is stopped after a block containing M01 is executed.	Auto mode, MDI mode, DNC mode
	Feedrate override key	Rapid traverse ON/OFF	Any mode
	Rapid traverse key	Rapid traverse ON/OFF	Manual mode
	Rapid, Step, and MPG override keys	For selecting rapid override, manual step override and MPG override	Auto mode, MDI mode, Machine zero return, MPG mode, Step mode, Manual mode, DNC mode
	Manual feeding key	For positive/negative movement of X, Y, Z and 4th axes in MANUAL mode and Step mode, and the axis moved in positive direction is selected by MPG	Machine zero return mode, Step mode, Manual mode, MPG mode
	Channel selection key	For machining channel switch (The function is unavailable temporarily)	Any mode
	Feed hold key	Press this key to stop Auto operation	Auto mode, MDI mode, DNC mode
	Cycle start key	Press this key to start program Auto operation	Auto mode, MDI mode, DNC mode

Note: A block with more than 1 “/” sign at its beginning is skipped by the system even if the skip function is OFF.

1.2.5 Machine control area of GSK218MC-H and GSK218MC-V

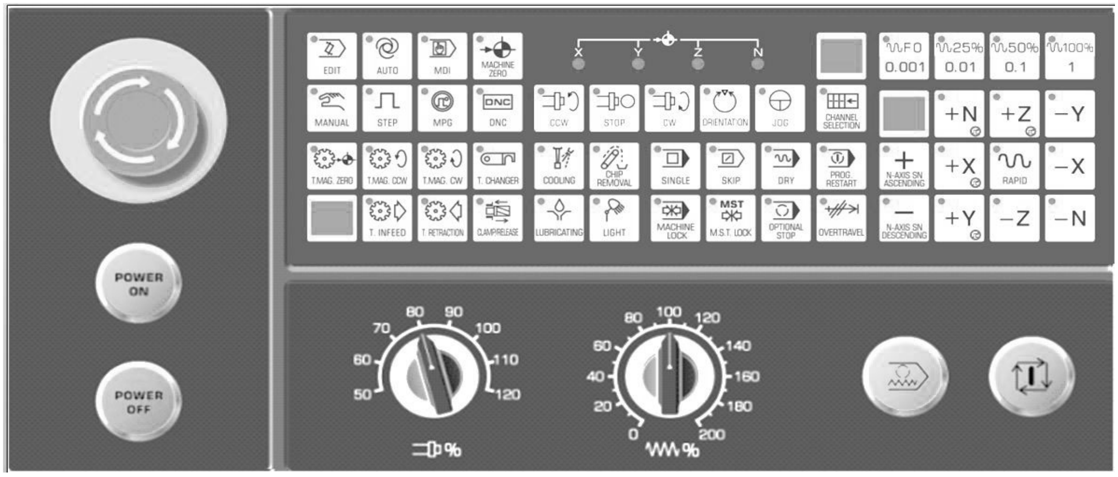


Fig. 1-2-5-1 Machine control area of GSK218MC-H

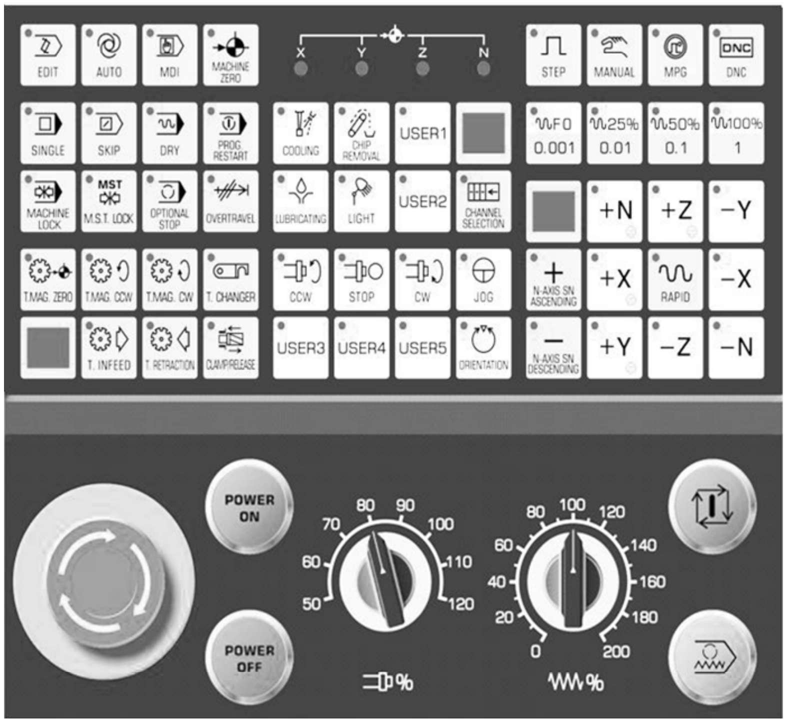


Fig. 1-2-5-2 Machine control area of GSK218MC-V

The use and function definition of the basic keys for the machine control area of GSK218MC-H and GSK218MC-V are the same as those for 218MC. Therefore, only the newly added keys are explained here.

Key	Designation	Explanation	Remarks and operation explanation
	Emergency stop key	The system enters emergency stop state by pressing this key	Any mode
 	N axis selecting key	For axis switch among multiple axes	Manual mode, step mode, MPG mode
	Spindle override switch	For spindle speed adjustment (spindle speed analog control valid)	Any mode
	Feedrate override switch	For feedrate adjustment	Auto mode, MDI mode, manual mode, DNC mode

Note 1: The feed hold key  and cycle start key  of 218MC are equivalent to key  and key  of 218MC-H and 218MC-V. The introduction below is based on the keys of 218MC.

Note 2: When the rapid traverse key is not pressed in manual mode, the manual speed override is adjusted with the feedrate override switch.

Note 3: In the explanation below, the keys in < > are the panel keys, in 【 】 are the soft keys at the bottom of the screen; 【 】 indicates the corresponding page of the current soft key; ⊞ indicates there are submenus.

CHAPTER 2 SYSTEM POWER ON/OFF AND SAFETY OPERATIONS

2.1 System power-on

Before GSK218M CNC system is powered on, ensure that:

1. The machine state 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 coordinates) is displayed after system self-check and initialization.

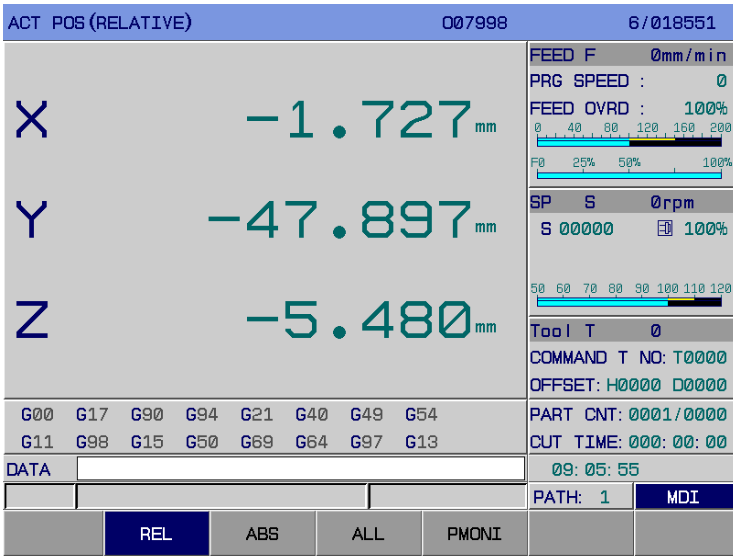


Fig. 2-1-1

2.2 System power-off

Before turning off the system, make sure that:

1. The axes X, Y, Z of the CNC are at halt;
2. Miscellaneous functions (spindle, pump, etc.) are off.
3. The CNC power is cut off prior to cutting off the machine power.

When cutting off the power, check that:

1. The LED, which indicates the cycle start on the operator panel, is off.
2. All the movable parts of the CNC machine tool are at halt.
3. Press POWER OFF button to turn off the power.

Cutting off the power in an emergency

The power should be cut off immediately to prevent accidents in an emergency situation during the machine running. However, the zero return, tool setting, etc. must be performed again because an error between system coordinates and actual coordinates may occur after power-off.

Note: See the manual provided by the machine tool builder for the machine power cut-off.

2.3 Safety operations

2.3.1 Reset operation

With key 

pressed, the system enters the reset state:

1. All axes movement stops;
2. The M functions are ineffective;
3. Whether the G codes are saved after resetting is determined by bit parameters NO:35#1～NO:35#7 and NO:36#0～NO:36#7;
4. Whether F, H, D codes are cleared after resetting is determined by bit parameters NO:34#7;
5. In MDI mode, whether the edited program is deleted after resetting is determined by bit parameters NO:28#7;
6. Whether the relative coordinates are cancelled after resetting is determined by bit parameter NO:10#3;
7. In non-Edit mode, whether the cursor returns to the beginning of the program after resetting is determined by bit parameter NO:10#7;
8. Whether macro local variables #1～#50 are cleared after resetting is determined by bit parameter NO:52#7;
9. Whether macro common variables #100～#199 are cleared after resetting is determined by bit parameter NO:52#6;
10. Resetting can be used during abnormal system output and coordinate axis action.

2.3.2 Emergency stop

If the Emergency Stop button is pressed during machine running, the system enters into emergency state and the machine movement is stopped immediately. Release the button (usually rotate the button towards left) to exit the state.

Note 1: Confirm the faults have been removed before releasing the Emergency Stop button;

Note 2: Perform Reference Point Return again after releasing the Emergency Stop button to ensure the coordinate position is correct.

In general, the emergency stop signal is a normal closed signal. When the contact point is open, the system immediately enters into the emergency stop state and emergently stops the machine. The connection for the emergency stop signal is as follows:

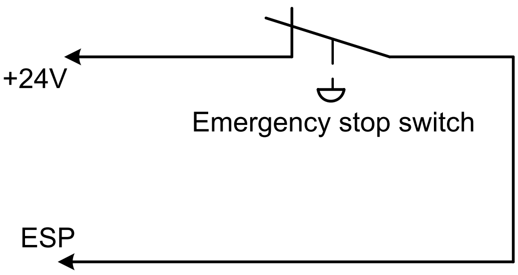


Fig. 2-3-2-1

2.3.3 Feed hold

Users can suspend the execution pressing key 

during the machine running. Please note that the execution is not suspended in rigid tapping instructions and cycle instructions until the current instruction is executed.

2.4 Cycle start and feed hold

The keys 

and 

are used for the program start and dwell operations in Auto mode, MDI mode and DNC modes. Whether the external start and dwell is used is set by PLC address **K5.1**.

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 and Z axes respectively. If the overtravel occurs, the moving axis decelerates and stops after it touches the limit switch. Meanwhile, the overtravel alarm is issued.

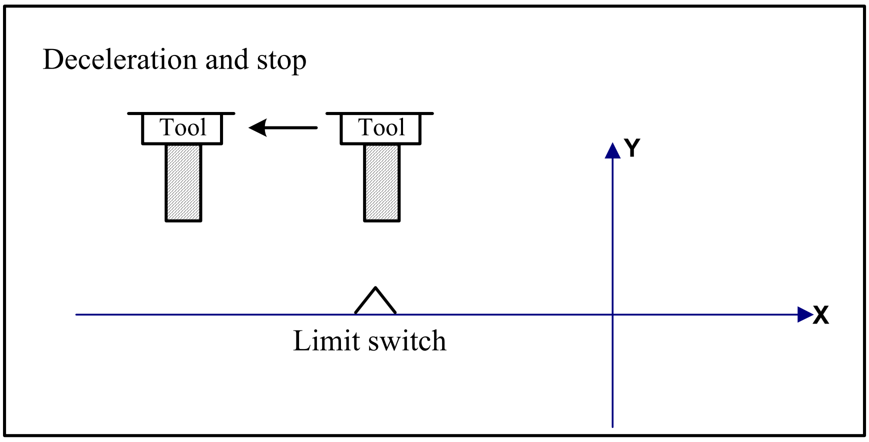


Fig. 2-5-1-1

Explanation:

Overtravel during auto mode

In Auto mode, if the tool hits the stroke limit switch during the movement along an axis, all the axis movements are decelerated to stop with the overtravel alarm being issued. The program execution is stopped at the block where the overtravel occurs.

Overtravel during Manual mode

In MANUAL mode, if any axis contacts the stroke limit switch, all axes will slow down immediately and stop.

2.5.2 Software overtravel protection

The software stroke ranges are set by the data parameters P66~P73, with the machine coordinates taken as the reference values. Overtravel alarm occurs if the moving axis exceeds the setting software stroke. Whether the stroke check is performed after power-on and before manual reference point return is determined by bit parameter N0:11#6 (0: No, 1: Yes). Whether the overtravel alarm is issued before or after the overtravel when the software limit overtravel occurs is set by bit parameter N0:11#7 (0: before, 1: after). After the overtravel occurs, move the axis out of the overtravel range in the reverse direction in Manual mode to release the alarm.

2.5.3 Overtravel alarm release

Method to release the hardware overtravel alarm: In manual or MPG mode, press key 

on the panel, then move the axis in the reverse direction (for positive overtravel, move negatively; for negative overtravel, move positively).

2.6 Stroke check

By stored stroke check 1 and 2, the system can specify 2 areas where the tool is forbidden to enter.

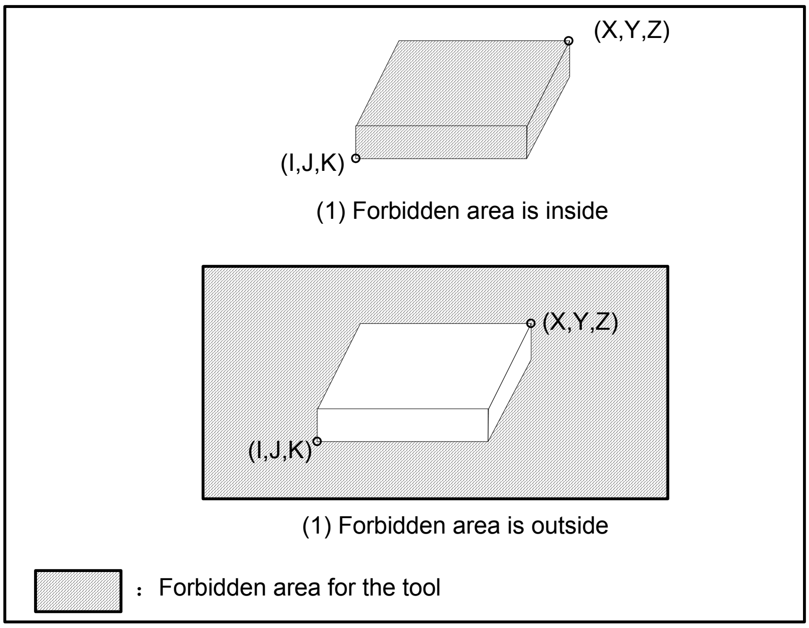


Fig. 2-6-1 Stroke check

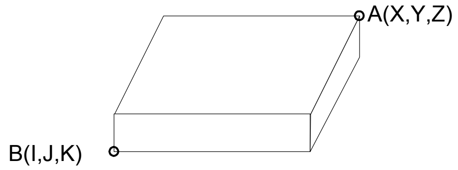
When the tool is moved beyond the stroke, an alarm is issued and the machine is decelerated and stopped.

When the tool enters the forbidden area with an alarm issued, move the tool in the reverse direction relative to the one in which the tool enters.

Explanation:

1. Stored stroke check 1: Its boundary is set by data parameters P66~P73. The outside of this area is the forbidden area, which is usually set as the machine maximum stroke by the machine builder.
2. Stored stroke check 2: Its boundary is set by data parameters P76~P83 or program instructions. The inside or outside of this area can be set as a forbidden area by bit parameter **NO:11#0** (0: inside for forbidden area; 1: outside for forbidden area)

1) Point A and point B in the following figure must be set when the forbidden area is set by parameters.



X>I,Y>J,Z>K

X-I>ζ(Least instruction increment unit)

Y-J> ζ(Least instruction increment unit)

Z-K>ζ (Least instruction increment unit)

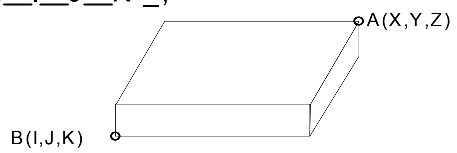
Fig. 2-6-2 Creating or changing forbidden area by parameters

When the forbidden area is set by data parameters P76~P83, the data should be specified by the distance (output increment) from the machine coordinate system in the least instruction increment unit.

2) When the forbidden area is set using program instructions: G12 forbids the tool to enter the forbidden area; G13 allows the tool to enter the forbidden area.

G12 must be specified in a separate block in a program. The instructions below are used for creating or changing the forbidden area.

G12 X__Y__Z__I__J__K _;



X>I,Y>J,Z>K

X-I>ζ(Least instruction increment unit)

Y-J> ζ(Least instruction increment unit)

Z-K>ζ(Least instruction increment unit)

Fig. 2-6-3 Creating or changing forbidden area using programs

If it is set by a G12 instruction, specify the data by the distance from the machine coordinate system in the least input increment (Input increment). The programmed data is then converted into the numerical values in the least command increment, and the values are set as the parameters.

Example 1: The inside is the forbidden area (bit parameter **NO:11#0**=0):

N1 G12 X50 Y40 Z30 I20 J10 K15;	Setting point A (50，40，30) and point B （20，10，15） for the tool forbidden area
N2 G01 X30 Y30 Z20;	Linear interpolation to (30，30，20)
N3 G13;	Cancelling stored stroke check
N4 G01 X50;	

Example 2: The outside is the forbidden area (bit parameter NO: 11#0=1):

N1 G12 X50 Y40 Z30 I20 J10 K15;	Setting point A （50，40，30）and point B （20，10，15）for the tool forbidden area
N2 G01 X10 Y-10 Z-10;	Linear interpolation to (10，-10，-10）
N3 G13;	Cancelling the stored stroke check
N4 G01 X50;	

3) Check point for the forbidden area: Before programming for the forbidden area, please confirm the check point (the top of the tool nose or tool holder). As is shown in Fig.2-6-4, if the check point is A (tool nose), the distance “a” should be set as the data for stored function check; if the check point is B (tool holder), the distance “b” should be set as the data for stored function check. When the check point is A (tool nose), and the tool lengths vary with the tools, the forbidden area should be created according to the longest tool, thus ensuring the safe operation.

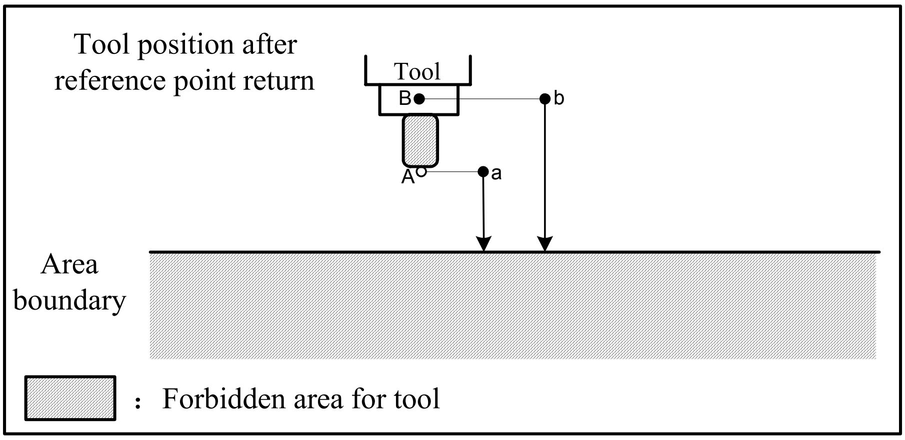


Fig. 2-6-4 Setting forbidden area

4) Tool forbidden area overlap: The forbidden area can be created by overlap, as is shown in the following figure:

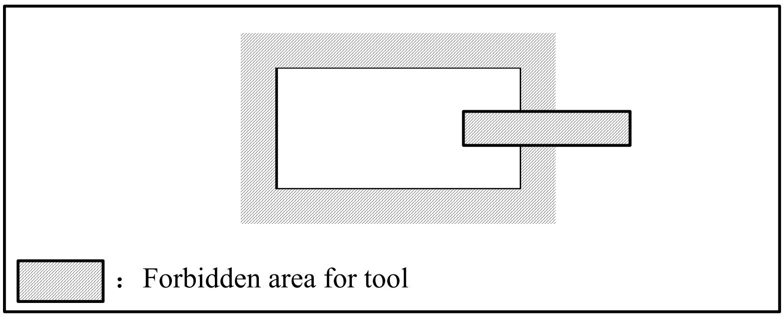


Fig. 2-6-5 Setting overlapping forbidden area

Unnecessary limits should be set beyond the machine stroke.

5) When bit parameter NO:11#6=0, effective time for a forbidden area: after power is switched on, and manual reference point return or automatic reference point return by G28 is executed, the forbidden area becomes effective.
When bit parameter NO:11#6=1, after the power is turned on, if the reference position is in the forbidden area, an alarm occurs (only effective in G12 of stored stroke limit 2)
6) Alarm release: If the tool enters the forbidden area with an alarm being issued, it can only be moved reversely. To release the alarm, move the tool reversely till it is beyond the forbidden area and resets the system. After the alarm is released, the tool can be moved forward or backward freely. See section 2.5.2 in this manual for details.
7) An alarm is issued when G13 is converted to G12 in the forbidden area.
8) Whether the stroke check is performed is set by bit parameter NO:10#1. When bit parameter NO:10#1=0, the stroke check is not performed before movement; when bit parameter NO:10#1=1, the stroke check is performed before movement.

CHAPTER 3 PAGE DISPLAY AND DATA MODIFICATION AND SETTING

3.1 Position display

3.1.1 Four types of position display

Press key 

to enter position page, which consists of 【REL】, 【ABS】, 【All】and 【PMONI】. The four subpages can be viewed using corresponding soft keys, as is shown below:

1) Relative coordinate: It displays the position of the current tool in the relative coordinate system by pressing soft key【REL】. See fig. **3-1-1-1**:

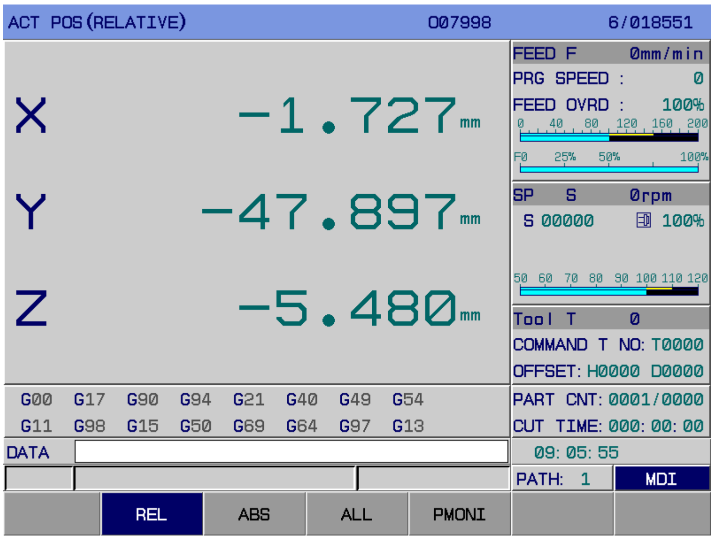


Fig. 3-1-1-1

2) Absolute coordinate: It displays the current position of the tool in absolute coordinate system by pressing soft key【ABS】(see Fig.3-1-1-2).

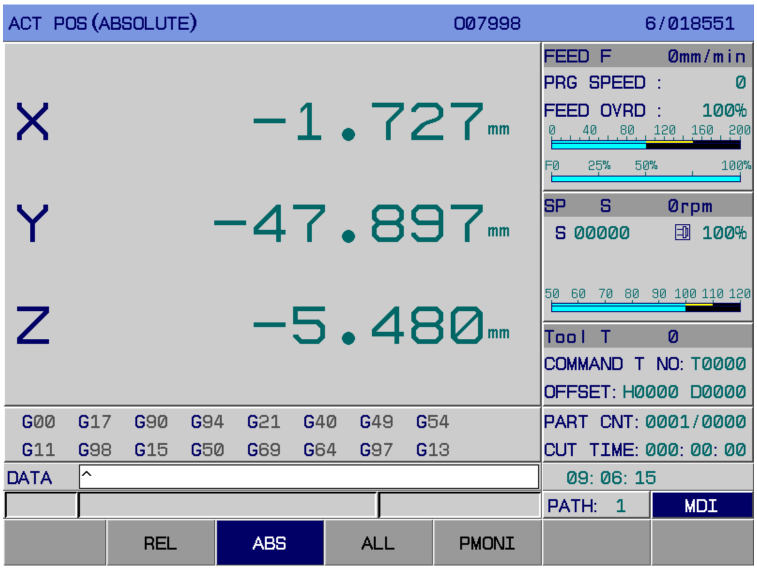


Fig. 3-1-1-2

3) ALL: It enters 【ALL】page by pressing soft key【ALL】, displaying the following items:

(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 (displacement) in MPG interruption;
(E) Speed component;
(F) Remaining distance (only displayed in Auto, MDI and DNC mode)

The display is as follows（Fig.3-1-1-3）:

Fig. 3-1-1-3

4) Monitor mode

It enters 【PMONI】 page by pressing soft key 【PMONI】. In this mode, the absolute coordinates, relative coordinates of the current position as well as the modal message and blocks of the program being executed can be displayed (See Fig. 3-1-1-4):

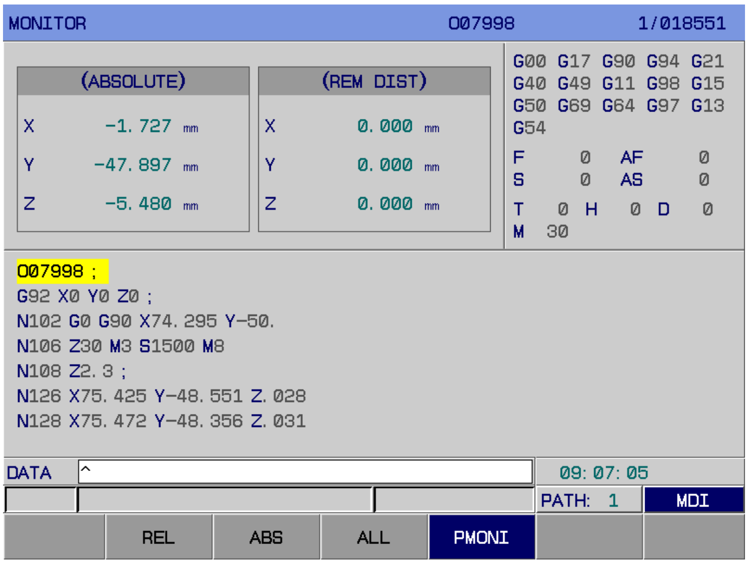


Fig. 3-1-1-4

Note 1: Whether the modes are displayed in 【PMONI】page can be set by parameter NO：23#6. When BIT6=0, the machine coordinates are displayed in the position where the modal instructions are displayed.

Note 2: In <MACHINE ZERO>, <STEP>, <MANUAL> and <MPG> modes, the intermediate coordinate system is a relative one; while in <AUTO>, <MDI> and <DNC> modes, it is the distance to go.

3.1.2 Display of cut time, part count, programming speed, override and actual speed

The programming speed, actual speed, feedrate and rapid override, G codes, tool offset, part number, cut time, spindle override, spindle speed, tools etc. can be displayed on the subpages【REL】and 【ABS】of page <POSITION> (see Fig.3-1-2-1).

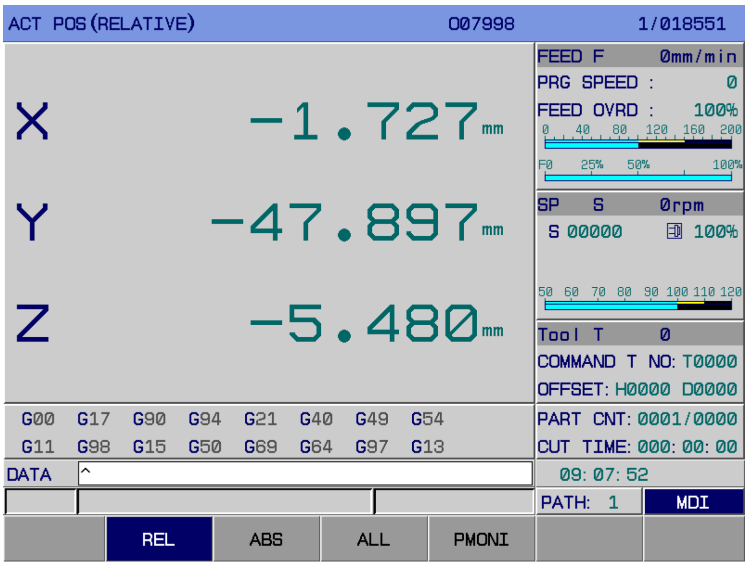


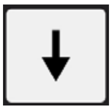
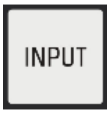
Fig. 3-1-2-1

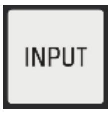
Book 2 Operation

The meanings of them are as follows:
Speed: The actual cutting speed overridden;
Programming speed: Speed specified by F code;
Feedrate override: Feed override selected by feedrate override keys;
Rapid override: Rapid override selected by rapid override keys;
G codes: The values of the G codes in the block being executed;
Tool offset: H0000, the tool length compensation for the current program; D0000, the tool radius compensation for the current program;
Part count: When M30 or M02 is executed in Auto or DNC mode, the count increases by 1. In other modes, the count does not increase when M30 or M02 is executed;
Cut time: Time counting starts after Auto run starts, with a unit of "hour: minute: second";
S_X: Spindle override for adjusting spindle speed
S00000: Actual feedback speed of spindle encoder
T0000: Tool number specified by T code in a program
Note: The part count is reserved after power-down.

Ways to clear part count and cut time:

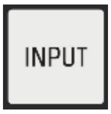
1) Switch to POSITION page, select MDI mode

2) Press key  to locate the cursor to the PRT CNT item, input data and press key 

for confirmation; if key 

is pressed directly, the part count will be cleared.

3) Shift to CUT TIME by keys Up and Down.

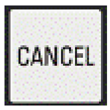
4) Press key 

to clear the CUT TIME.

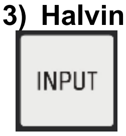
Note 1: To display the actual spindle speed, an encoder must be applied to the spindle.
Note 2: The actual speed= the programming speed F × override; The speed of each axis is set by data parameters P88～P92 in G00 mode and it can be overridden by rapid override; the dry run speed is set by data parameter P86.
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 number of machined workpieces can be set by data parameter P356, and the total number of workpieces to be machined is set by number parameter P357.

3.1.3 Relative coordinate clearing and halving

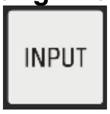
The steps for clearing relative coordinate position are as follows:
1) Enter any page that displays the relative coordinates (Fig. 3-1-2-1);

2) Clearing operation: Press and hold key "X" till X in the page flickers, then press key 

to clear the relative coordinate in X axis; (Fig. 3-1-2-2)

3) Halving operation: Press and hold key "X" till "X" in the page flickers, then press key 

to halve the relative coordinate in X axis. (The relative coordinate of the axis is divided by 2)

4) Coordinate setting: Press and hold key “X” till “X” in the page flickers, input the data to be set and press key  for confirmation, then the data will be input into the coordinate system.

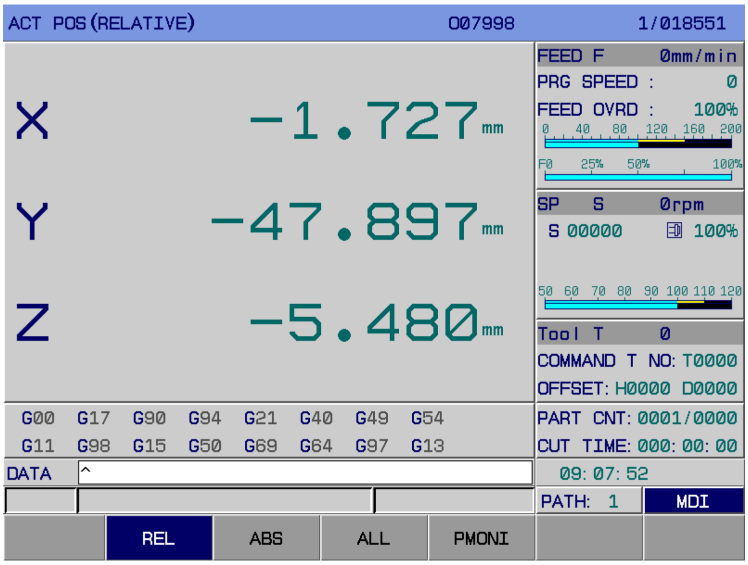


Fig. 3-1-3-1

5) Steps for clearing the relative coordinate positions of axes Y and Z are the same as the above.

3.2 Program display

Press key  to enter program display page which consists of 5 subpages: 【⊞PRG】, 【MDI】, 【CUR/MOD】, 【CUR/NXT】 and 【DIR】. They can be viewed and modified by corresponding soft keys (See Fig.3-2-1).

1） Program display

Press soft key【⊞PRG】 to enter program page. In this page, a page of blocks being executed in the memory can be displayed (See Fig. 3-2-1).

Book 2 Operation

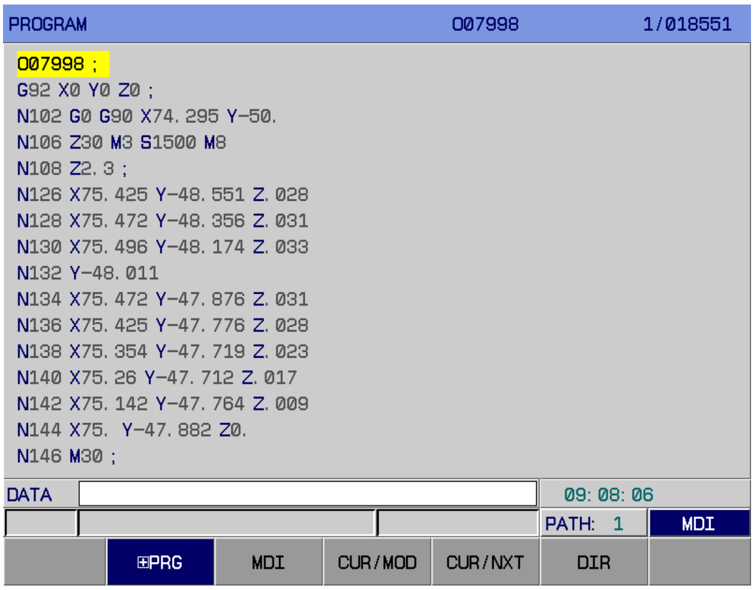


Fig. 3-2-1

By pressing soft key 【⊞PRG】 again, the program EDIT and modification page is entered (see Fig.3-2-2).

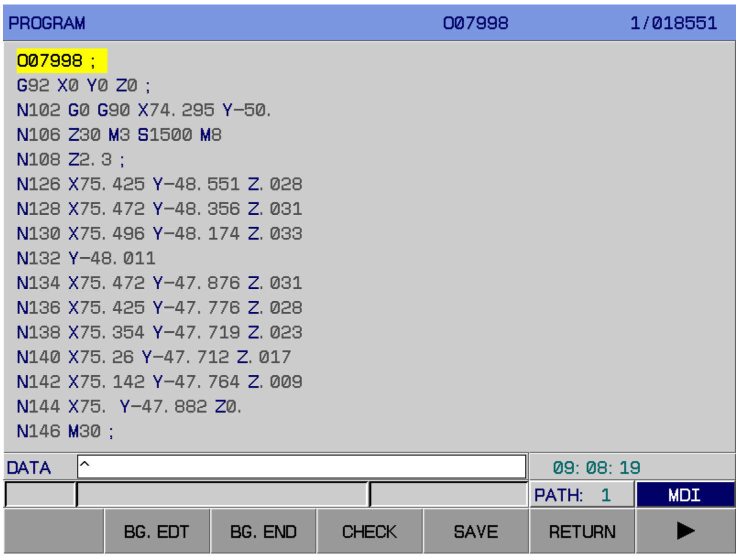


Fig. 3-2-2

Press key【▶】to enter the next page

Press key【▶】to enter the next page

Press key【◀】 to return to the previous page

Note: The 【CHECK】 function can only be performed in Auto mode.

【BG. EDIT】and 【BG. END】are used only in AUTO and DNC mode (background edit function). Functions of 【BG.EDIT】are the same as the program edited in <EDIT> mode (See CHAPTER 10 “Program Edit”). Save the editing by 【BG. END】or exit the background EDIT page by 【RETURN】after editing.

2） MDI display

Press soft key 【MDI】 to enter MDI page. In this mode, multiple blocks can be edited and executed. The program format is the same as that of the editing program. MDI mode is applicable to simple program testing operation (see Fig. 3-2-3) .

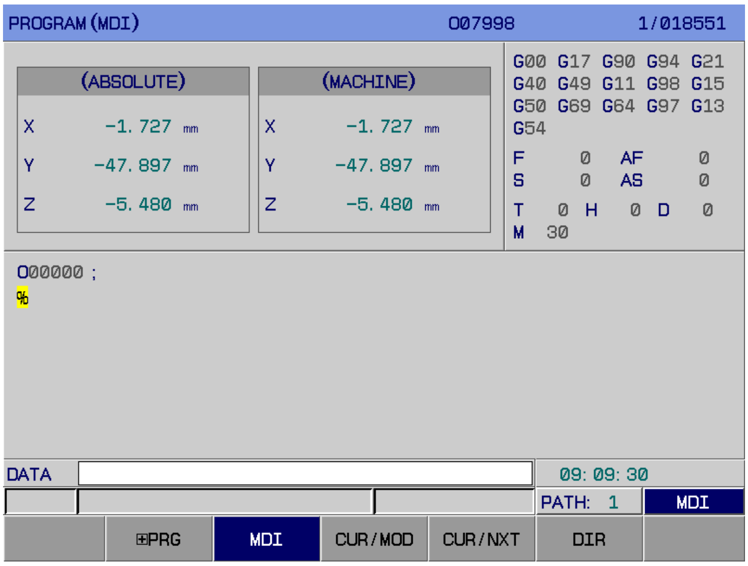


Fig. 3-2-3

3) Program (CUR/MOD) display

Press soft key【CUR/MOD】to enter current/mode page. It displays the instructions of the blocks being executed and the current modal values. MDI data input and execution are available in MDI mode. (See Fig. 3-2-4**).**

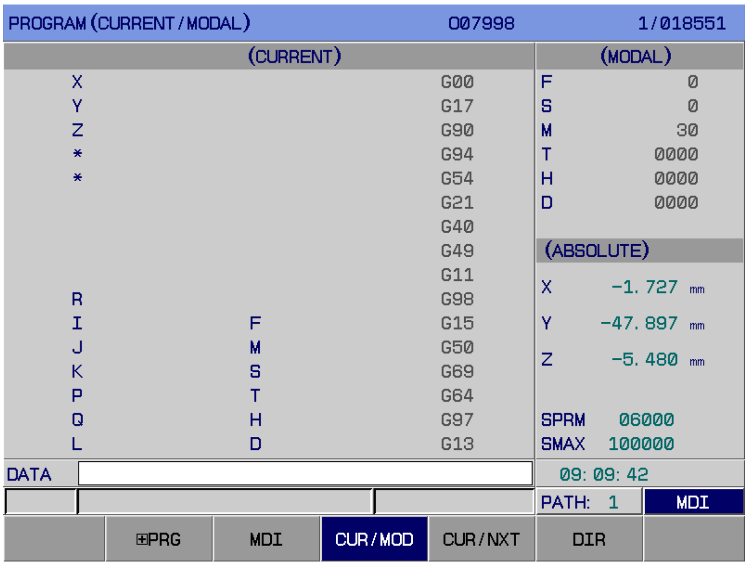


Fig. 3-2-4

4) Program (CUR/NXT) display

Press soft key【CUR/NXT】 to enter current/next page. It displays the instructions of the blocks being executed and the blocks to be executed. (See Fig. 3-2-5**).**

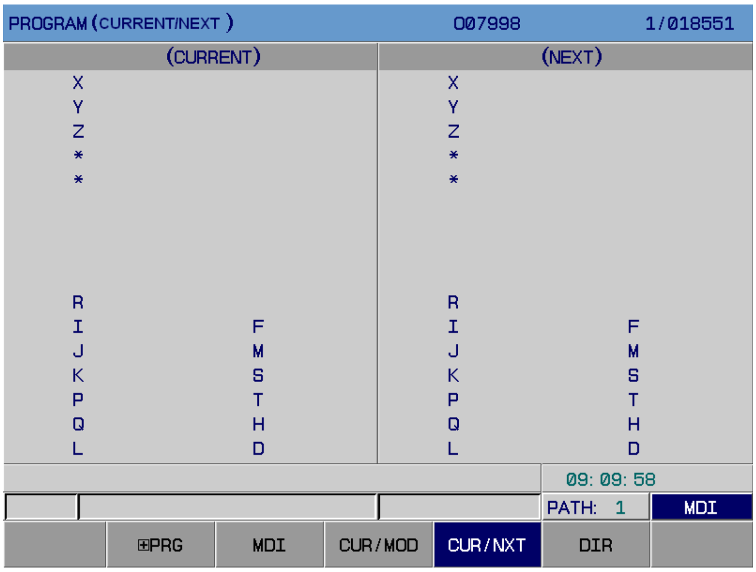


Fig. 3-2-5

5) Program (DIR) display

Ⅰ. Press soft key【DIR】to enter program (DIR) page, the contents of which are displayed as follows (Fig.3-2-6):

(a) PRG USED: The saved programs (including subprograms) /maximum number of the programs that can be saved.

(b) MEM USED: The capacity occupied by the saved programs /the remaining capacity for program storage.

(c) PROGRAM DIR: The sequence numbers of the saved programs are displayed in sequence.

(d) Previewing the program where the cursor is located

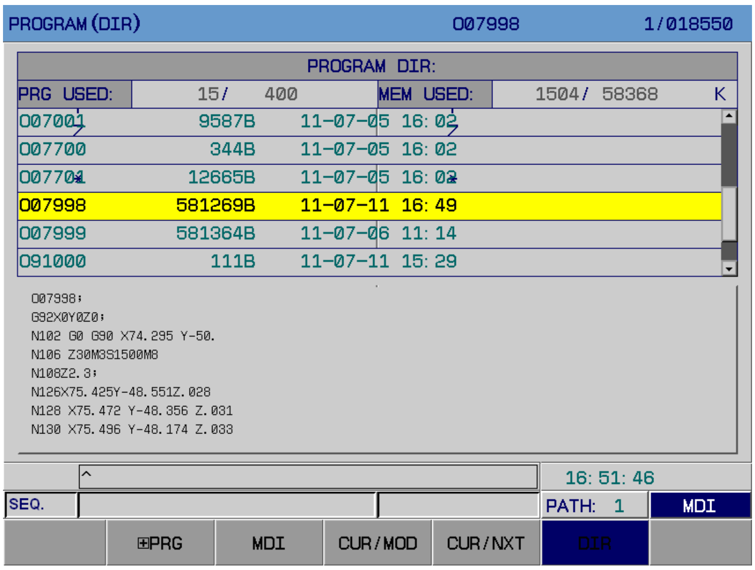


Fig. 3-2-6

Ⅱ Press soft key 【DIR】again to enter PROGRAM (USB DIR) display page, the contents of which are displayed as follows (See Fig. 3-2-7):

:

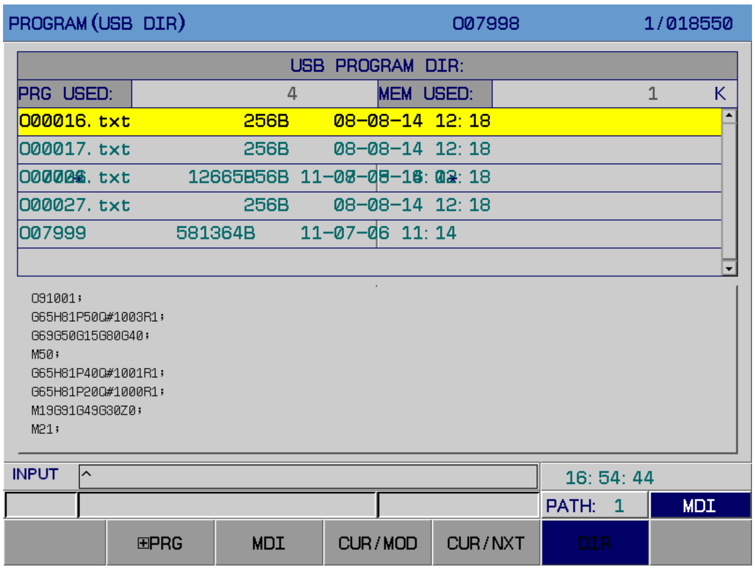


Fig. 3-2-7

Explanation: The program numbers in memory can be displayed by the page keys. The program names with more than 6 digits or irregular formats cannot be previewed.

3.3 System display

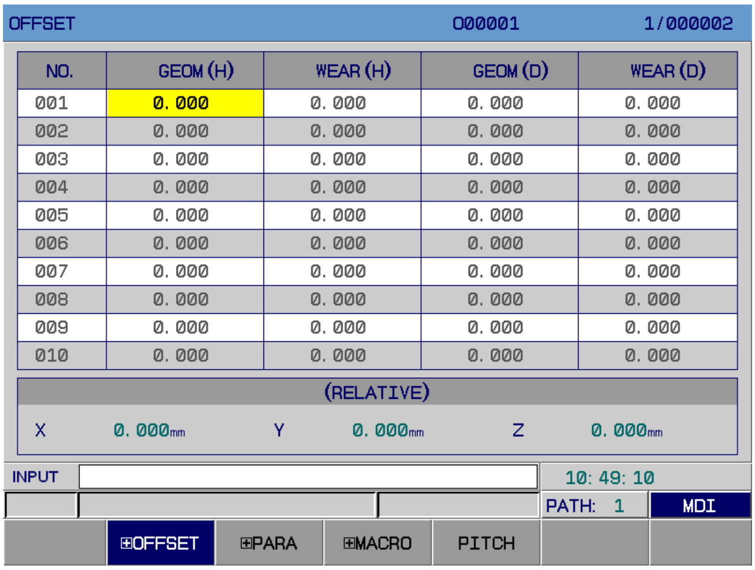
Press key 

to enter system page, which consists of four subpages: 【⊞OFFSET】, 【⊞PARA】, 【⊞MACRO】and【PITCH】. They can be displayed by corresponding soft keys. See fig. 3-6-1 below:

3.3.1 Display, modification and setting for offset

3.3.1.1 Offset display

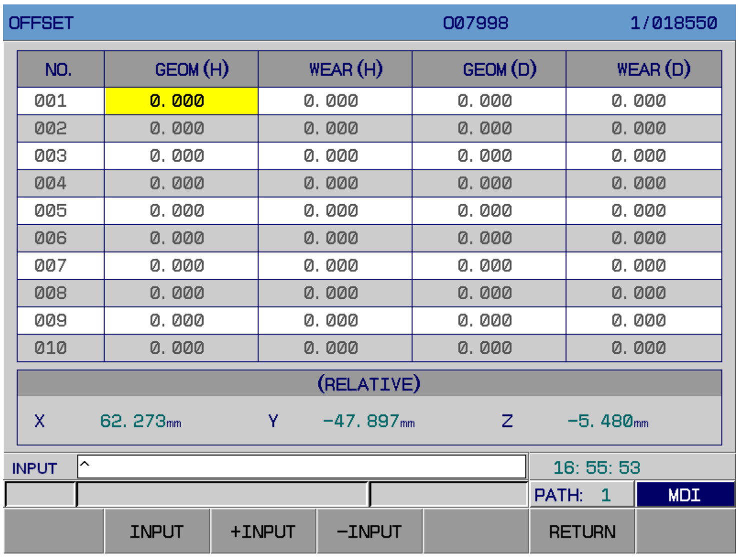
Press soft key【⊞OFFSET】to enter OFFSET page which is shown as follows (fig. 3-3-1-1-1):



NO.	GEOM (H)	WEAR (H)	GEOM (D)	WEAR (D)
001	0.000	0.000	0.000	0.000
002	0.000	0.000	0.000	0.000
003	0.000	0.000	0.000	0.000
004	0.000	0.000	0.000	0.000
005	0.000	0.000	0.000	0.000
006	0.000	0.000	0.000	0.000
007	0.000	0.000	0.000	0.000
008	0.000	0.000	0.000	0.000
009	0.000	0.000	0.000	0.000
010	0.000	0.000	0.000	0.000

Fig. 3-3-1-1-1

Press soft key【⊞OFFSET】in the above figure to enter offset operation subpage. See fig. **3-3-1-1-2:**



NO.	GEOM (H)	WEAR (H)	GEOM (D)	WEAR (D)
001	0.000	0.000	0.000	0.000
002	0.000	0.000	0.000	0.000
003	0.000	0.000	0.000	0.000
004	0.000	0.000	0.000	0.000
005	0.000	0.000	0.000	0.000
006	0.000	0.000	0.000	0.000
007	0.000	0.000	0.000	0.000
008	0.000	0.000	0.000	0.000
009	0.000	0.000	0.000	0.000
010	0.000	0.000	0.000	0.000

Fig. 3-3-1-1-2

The offset value can be input directly or added to or subtracted from the actual position value. GEOM (H) stands for tool length compensation, WEAR (H) for tool length abrasion; GEOM (D) stands for tool radius compensation, and WEAR (D) for tool radius abrasion.

3.3.2.2 Modification and setting for offset value

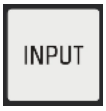
The steps for setting tool offset in Offset page are as follows:

1) Press soft key【⊞OFFSET】to enter offset display page.

2) Move the cursor to the target offset number.

Step 1: Press page keys to display the page where the offset value is to be modified, move the cursor by pressing cursor keys to the offset number to be modified.

Step 2: Press key  to search after inputting the offset number.

3) Input offset value in any mode, and press key 

or soft key【INPUT】for confirmation.

4) In any mode, input offset amount, and then press soft key【+INPUT】 or 【-OUTPUT】. After that, the system computes the offset amount automatically and displays it on the screen.

Note 1: During the tool offset modification, the new offset value is ineffective till the T code which specifies its offset number is specified.

Note 2: The offset value can be modified anytime during the program execution. If the value is required to take effect in time during the program execution, the modification must be completed before the tool offset number is executed.

Note 3: If the length offset value needs to be added to the relative coordinate value of Z axis, the offset value should be specified behind Z code, then they will be automatically added up in the system.
For example, if Z 10 is input, the offset value is the one obtained by adding 10 to the current relative coordinate value of Z axis.

3.3.2 Display, modification and setting for parameters

3.3.2.1 Parameter display

Press soft key【⊞PARA】to enter parameter page. There are two subpages, including【BITPAR】 and 【NUMPAR】. Both of them can be viewed and modified by corresponding soft keys, as is shown below:

1）Bit parameter page Press soft key【BITPAR】to enter this page (see Fig. 3-3-2-1-1):

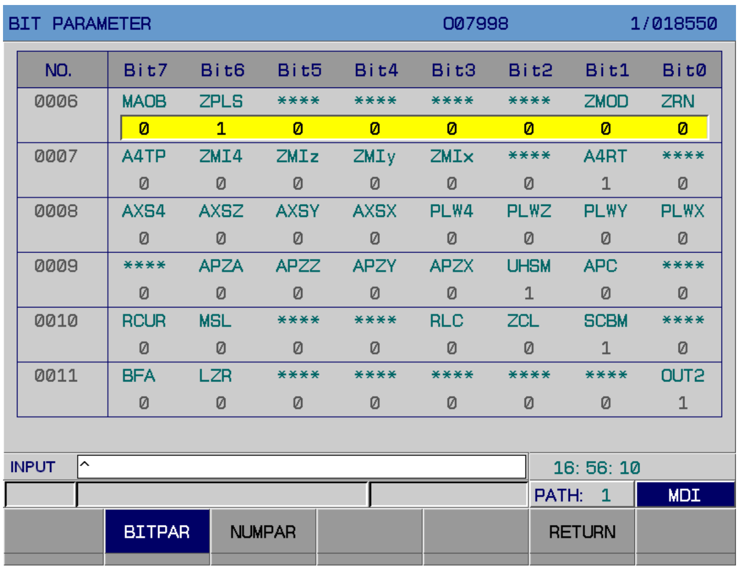


Fig. 3-3-2-1-1

Refer to APPENDIX 1 PARAMETERS for details.

2) Number parameter page Press soft key【NUMPAR】to enter this page. (See fig. 3-3-2-1-2)

Book 2 Operation

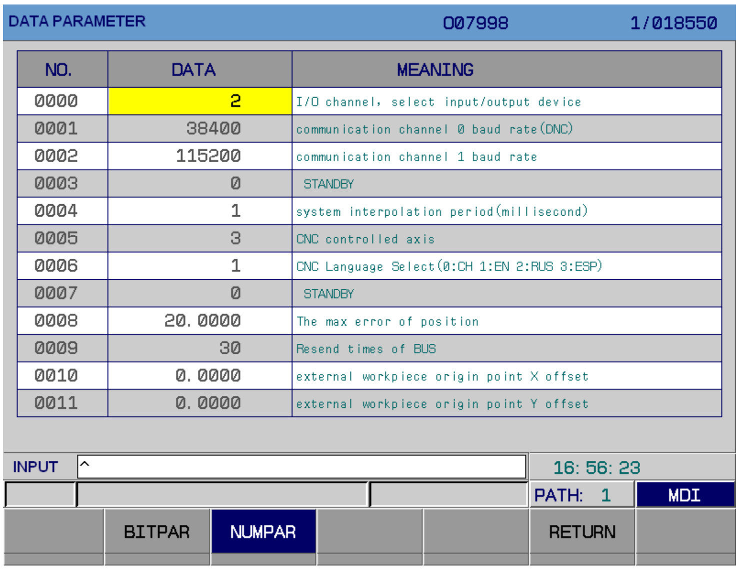


Fig. 3-3-2-1-2

Refer to APPENDIX 1 PARAMETERS for details.

3.3.2.2 Modification and setting for parameter values

1) Select MDI mode;

2) Press key 

to enter <SETTING> page, turn on the parameter switch (set the parameter switch to 1))

3) Press key 

, then the soft key【⊞PARA】to enter parameter display page.

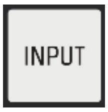
4) Move the cursor to the parameter number to be modified:

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 key 

to search after inputting the parameter number.

5) Input a new parameter value using number keys (corresponding passwords are required for modifying parameters of different levels)

6) Press key 

for confirmation, then the parameter value is input and displayed.

7) Turn off the parameter switch after setting all the parameters.

3.3.3 Display, modification and setting for macro variables

3.3.3.1 Macro variable display

Press soft key【⊞MACRO】 to enter macro variable page, which consists of two subpages:【CUSTOM】 and 【SYSTEM】. Both of them are available to be viewed and modified by corresponding soft keys, as is shown below:

1) User variable page Press soft key【CUSTOMER】to enter this page.

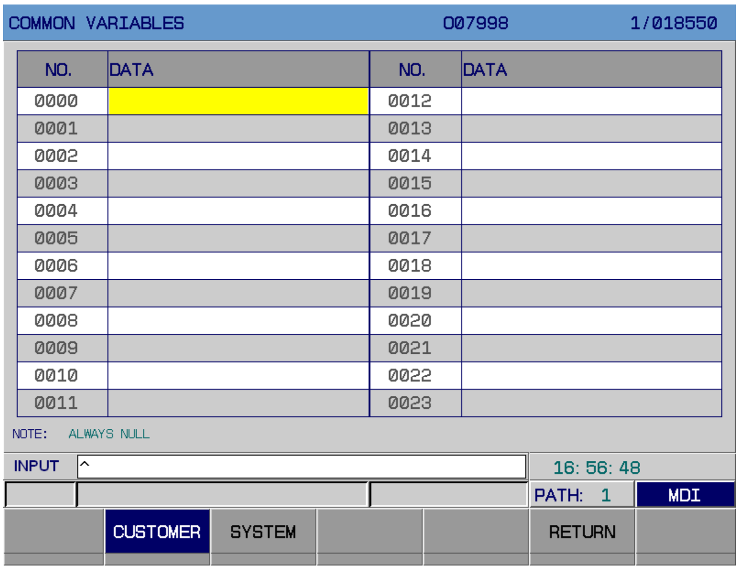


Fig. 3-3-3-1-1

2) System variable page Press soft key【SYSTEM】to enter this page.

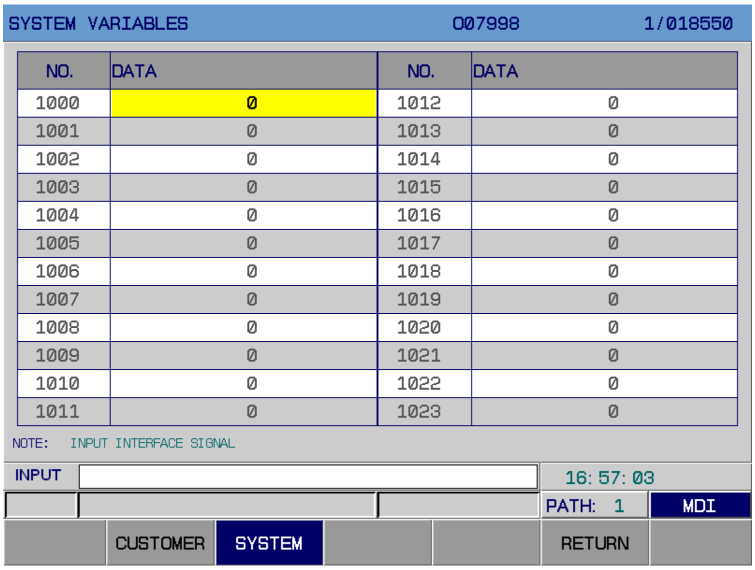


Fig. 3-3-3-1-2

Refer to SECTION 4.7.2 in PROGRAMMING for the explanation and use of macro variables.

3.3.3.2 Modification and setting for macro variables

1) Select <MDI> mode.

2) Press key 

, then soft key【⊞MACRO】to enter macro variable page.

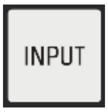
3) Move the cursor to the variable number to be modified.

Method 1: Press page keys to display the page where the variable is to be modified; move the cursor to the variable to be modified.

Method 2: Press key 

to search after inputting the variable number.

4) Input a new value using number keys.

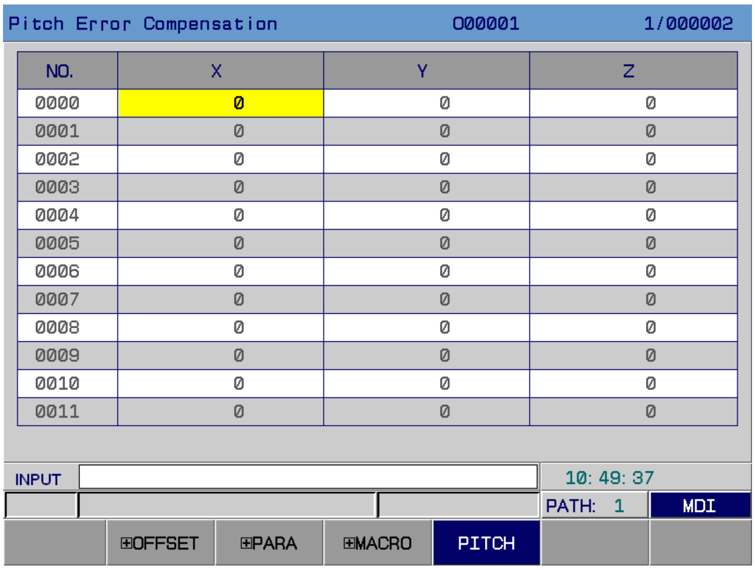
5) Press key 

for confirmation, and then the value will be input and displayed.

3.3.4 Display, modification and setting for screw pitch offset

3.3.4.1 Pitch offset display

Press soft key【PITCH】to enter pitch offset page, which is shown as follows (fig. 3-3-4-1-1):



NO.	X	Y	Z
0000	0	0	0
0001	0	0	0
0002	0	0	0
0003	0	0	0
0004	0	0	0
0005	0	0	0
0006	0	0	0
0007	0	0	0
0008	0	0	0
0009	0	0	0
0010	0	0	0
0011	0	0	0

Fig. 3-3-4-1-1

3.3.4.2 Modification and setting for pitch offset

1) The pitch error offset point for each axis is set by data parameters P221～P224, the pitch error offset interval by data parameters P226～P229, and the pitch error offset multiplier by data parameters P231～P234.

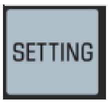
2) In <MDI> mode, input the offset value for each point in turn.

Note: Refer to VOLUME 4 INSTALLATION AND CONNECTION in "GSK218MC CNC System Installation and Connection Manual" for the setting of pitch offset.

3.4 Setting display

3.4.1 Setting page

1. Entering the page

Press key 

to enter the SETTING page. There are four subpages, including 【SETTING】, 【⊞WORK】, 【⊞DATA】and【PASSWORD】. All of them can be viewed or modified by corresponding soft keys. The contents are shown as follows (see Fig. 3-4-1-1):

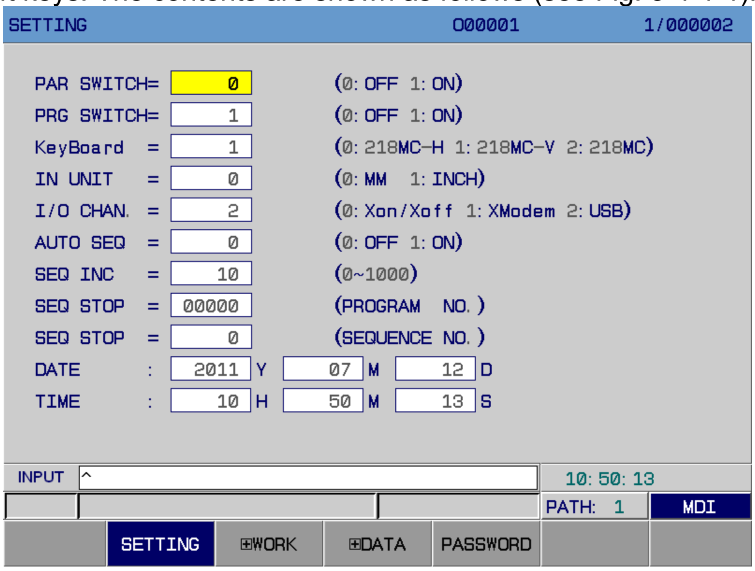


Fig. 3-4-1-1

2. Explanation for 【SETTING】 page

Press soft key 【SETTING】to enter the page shown as Fig. 3-4-1-1. After entering the page, users can view and modify the parameters. The operation steps are as follows:

（**a**） Enter < MDI> mode;

（**b**） Move the cursor to the item to be altered by pressing cursor keys;

（**c**） According to the explanation below, key in 1 or 0, or use left and right keys for modification :

1) Parameter switch

0: Parameter switch OFF 1: Parameter switch ON

When the parameter switch is set to 0, it is forbidden to modify and set the system parameters, meanwhile, an alarm "(0100: parameter writing valid) cancel" is issued. When the parameter switch is set to 1, an alarm "0100: parameter writing valid" is issued. Here, the user can cancel the alarm pressing key 

\+ key 

(This operation is only effective in 【SETTING】 page).

2) Program switch

0: Program switch OFF 1: Program switch ON

When the program switch is set to 0, it is forbidden to edit any program.

3) Keyboard selection

0：218MC-H 1：218MC-V 2：218MC

Note: In any mode, the keyboard selection can be modified if the Emergency Stop button is pressed.

4) Input unit

Set whether the input unit of the program is metric or inch:

0: Metric. 1: Inch.

5) I/O channel

It is set by users as required, e.g., if using U disk to perform DNC machining, set the channel to 2.

0，1：RS232（0 for selecting Xon/Xoff protocol，1 for selecting Xmodem protocol）

2：USB

6) Automatic sequence number

0: The system will not insert the sequence number automatically when the program is input with keyboard in edit mode.

1: When the program is input with keyboard in edit mode, the system will automatically insert the sequence number. The sequence number increment between blocks is set by data parameter P210.

7) Sequence number increment

Set the increment when inserting sequence number automatically. Range: 0~1000.

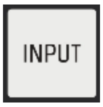
8) Stop sequence number

This function can be used to stop the program execution at a specified block, but it is not effective unless both the program number and block number are specified. E.g. 00060（program number）means program number O00060; 00100 (sequence number) means block number N00100.

Note: When the stop sequence is set to -1, the single block stop is not executed.

9) Date and time

Users can set the system date and time here.

(**d**) Press key 

for confirmation.

3.4.2 Workpiece coordinate setting page

1. Press soft key【⊞WORK】to enter coordinate system setting page, the contents of which are shown as follows:

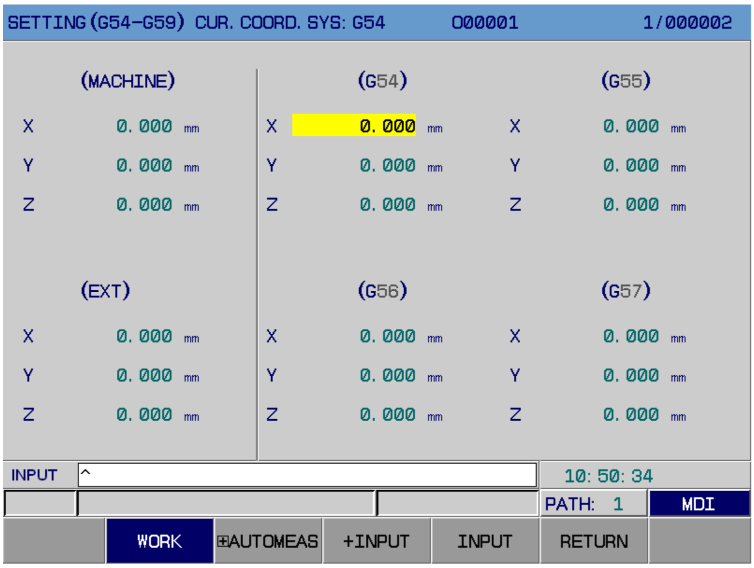


Fig. 3-4-2-1

Another 50 additional workpiece coordinate systems can be used besides the 6 standard workpiece coordinate systems (G54～G59 coordinate systems), as is shown in fig. 3-4-2-2. Each coordinate system can be viewed or modified by page keys. See section 4.2.9 Additional workpiece coordinate system in PROGRAMMING for details about its operation.

Fig. 3-4-2-2

2. There are two ways to input coordinates:

1) After entering this page in any mode, move the cursor to the coordinate system to be altered. Press the axis name to be assigned and then press key INPUT for confirmation, then the values in the current machine coordinate system will be set as the origin of the G coordinate system, e.g. by pressing "X " and then key INPUT, or pressing "X0" and then key INPUT, the X machine coordinate of this point is input automatically by the system; In addition, e.g. if X10 (or X-10) is input and then key INPUT is pressed, the X machine coordinate is +10 (or -10).

2) After entering this page in any mode, move the cursor to the coordinate axis to be altered, input the machine coordinates of the origin of the workpiece coordinate system directly, then press key INPUT for confirmation.

3. Method to search a coordinate system

1) In any mode, press key SEARCH to search after inputting a coordinate system, e.g. inputting "G56".

2) In any mode, by inputting "P6" or "P06" and then pressing key SEARCH, the cursor will be located in the additional workpiece coordinate system "G54 P06".

3.4.2.2 Auto tool setting

Press soft key 【⊞AUTO MEAS】to enter auto tool setting page, the contents of which are shown as follows (fig. 3-4-2-2-1):

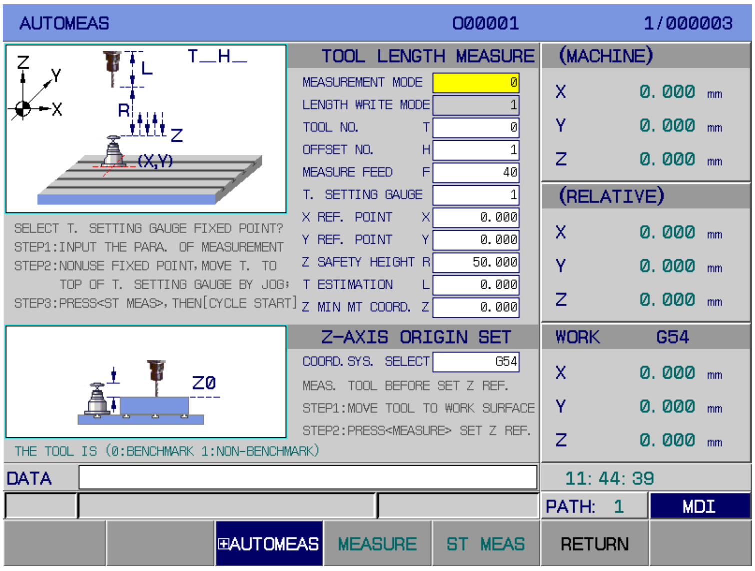


Fig. 3-4-2-2-1

Ⅰ. **Tool setting function**

The tool setting function consists of two parts, including automatic tool length measurement and Z axis workpiece origin setting.

A. Automatic tool length measurement

Automatic tool length measurement is used to measure the tool lengths of different tools by the tool setting gauge fixed on the worktable, and set the length difference between each tool and reference tool to the reference offset or tool offset, thus ensuring correct machining even if tools with different lengths are used in a program.

The basic principle is shown in fig. 3-4-2-2-2:

Reference tool: The tool with its length firstly measured after power-on is defined as the reference tool. The reference tool is usually a fixed one, which is not used for cutting; it can also be defined by users.

Reference tool length Lb: the displacement that the reference tool moves from the machine origin to the tool setting gauge. The length of the reference tool measured is saved in the macro variables till the system power-off. Lb cannot be modified and deleted, it is only for operation.

Current tool length Lc: The displacement that the current tool moves from the machine origin to the tool setting gauge.

Tool length difference ΔL: The tool length difference between current tool and reference tool. ΔL= Lc- Lb

After the reference tool is replaced by another tool and the length of the tool is measured, the obtained length difference between current tool and reference tool is set to the position specified by “measured value write mode”

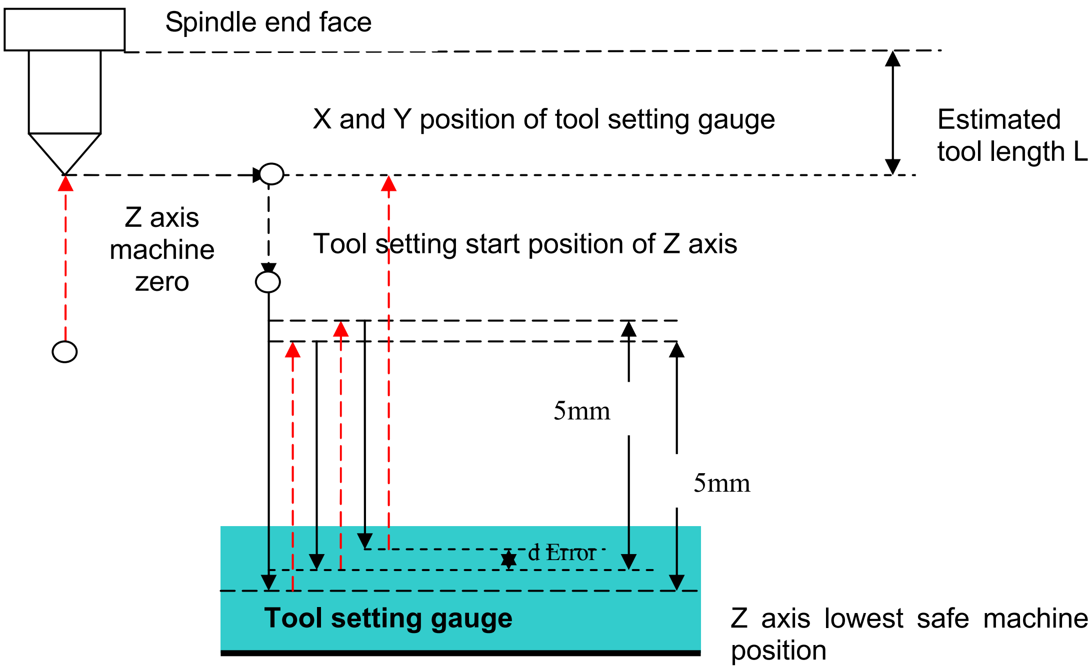


Fig. 3-4-2-2-2

B. Z axis workpiece origin setting

After finishing the tool length measurement, it is required to move the tool to the workpiece surface. Here, press soft key 【MEASURE】 to set the current machine coordinates as the origin to the selected workpiece coordinate system (G54~G59 G54 P1~P50).

Ⅱ Operation

- After system power-on, firstly perform machine zero return operation before the automatic tool length measurement. See Section 9.2 Operation procedure for machine zero return in BOOK 2 OPERATION for details.
- Turn on the parameter switch, set bit parameter NO：1#6 to 1 (tool setting gauge installed), set bit parameter NO：1#7 to 1 (reference point memory), bit parameter NO：2#5 sets whether the tool measured value can be written to reference offset, and bit parameter NO：2#6=1 sets the skip signal SKIP (0: 1, 1: 0) is input as a signal.

1. Auto tool length measurement and reference offset writing

Reference tool setting:

1) Measurement mode selection: 0: reference tool.

2) Measured value write mode: 0: reference offset. The bit parameter NO：2#5 (STME whether the tool measured value is written in reference offset) must be set to 1, or the value cannot be written. When the measured value write mode is 0, neither tool number nor tool offset number can be written.

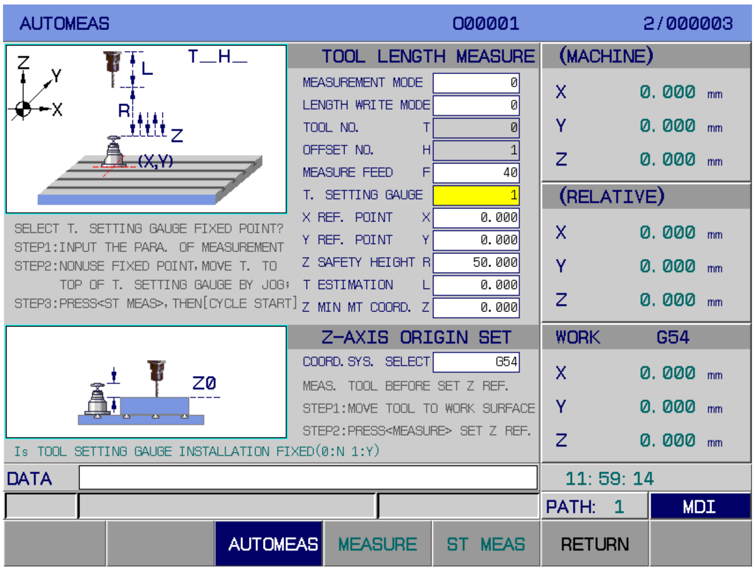


Fig. 3-4-2-2-3

3) Measure feed setting. The measure speed is the traverse speed at which the tool moves from the start point to point R. When it is input in metric, the default speed is 40 mm/min (Range:10~100mm/min); when it is input in inch, the default speed is 2.0 inch/min (0.4~4.0 inch/min).

4) Whether the tool setting gauge installation is fixed: 0: unfixed; 1: fixed. When it is set to unfixed mode, the position of the tool setting gauge on each axis cannot be modified; when it is set to fixed mode, the position of the tool setting gauge on each axis can only be modified with the authority of debugging level or above.

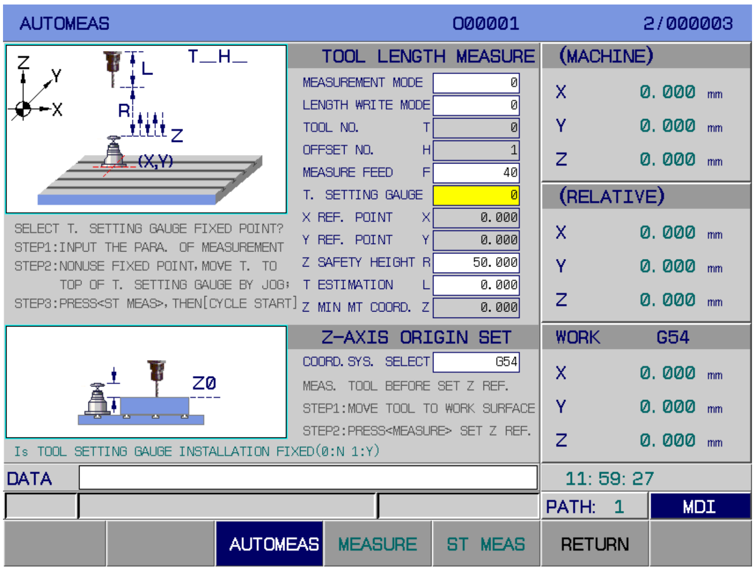


Fig. 3-4-2-2-4

5) Tool setting gauge position X on X axis: The X coordinate of the tool setting gauge in the machine coordinate system.

6) Tool setting gauge position Y on Y axis: The Y coordinate of the tool setting gauge in the machine coordinate system.

7) Start point R on Z axis: The distance (a positive value) the tool moves from the tool nose to the tool setting gauge at the measured speed.

8) Tool length estimation L: The distance (a positive value, metric: mm/min; inch：inch/min)

from the tool nose to the spindle end face.

9) The safe position Z of the tool setting gauge on Z axis: The safe position from the spindle end face to the tool setting gauge plane.

10) Select <AUTO> mode, press soft key 【AUTO MEAS】 to enable the system to call the tool setting macro program automatically, then press key 

to execute Auto Tool Setting.

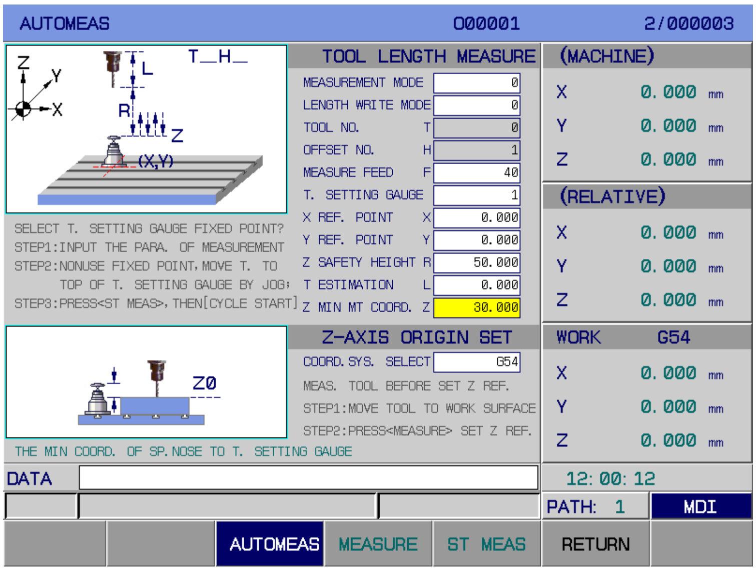


Fig. 3-4-2-2-5

Non-reference tool setting:

After the reference tool setting is finished, the system sets the Measurement Mode Selection to 1 automatically. Users can perform the non-reference tool setting after changing the reference tool: Measure the length of the non-reference tool and write it into the tool length estimation L, select <AUTO> mode, press 【AUTO MEAS】 to enable the system to call the tool setting macro program automatically, then press key 

to execute Auto Tool Setting.

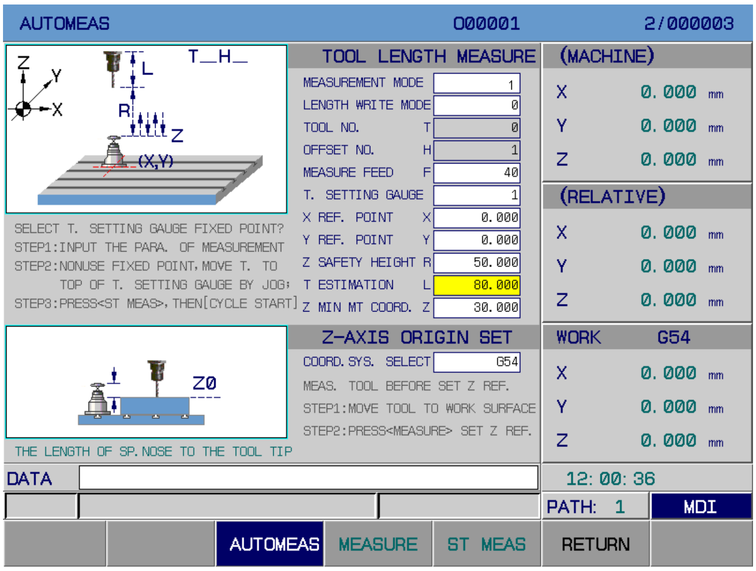


Fig. 3-4-2-2-6

The difference between a tool and reference tool is set to the reference offset.

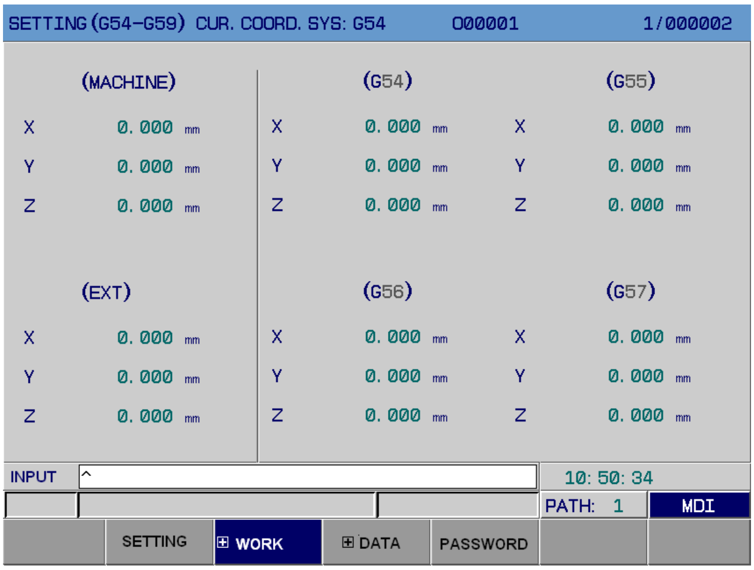


Fig. 3-4-2-2-7

2. Measuring tool length automatically and writing it into tool length offset H

Reference tool setting:

1) Measurement mode selection: 0: reference tool.

2) Measured value write mode: 1: tool offset. Here, modify the bit parameter NO：2#5 (STME whether the tool measured value is written into the reference offset) to 0, then the system will set the measured value write mode to 1 which cannot be modified.

3) Tool number: The purpose of changing the tool number is to change the measured tool number conveniently in the non-reference tool mode in a shorter time. The offset number changes with the change of the tool number. However, during tool setting, the tool measured value is subject to the offset number set in the tool offset number.

4) Tool offset number:

In reference tool mode, when the measured value write mode is set to 1 (tool length offset), users can change the tool offset number as required.

In non-reference tool mode, the tool offset number changes automatically as the tool number changes; the user can also modify the tool offset number as required.

5) Measure feed setting. The measured speed is the traverse speed at which the tool moves from start point R of tool setting to the tool setting gauge. When it is input in metric, the default is 40 mm/min (range: 10~100mm/min); when it is input in inch, the default is 2.0 inch/min (Range: 0.4~4.0inch/min).

6) Whether the tool setting gauge installation is fixed: 0: unfixed; 1: fixed. When the unfixed mode is set, the tool setting gauge position on each axis cannot be modified; when the fixed mode is set, the tool setting gauge position on each axis can only be modified with debugging-level authority or above.

7) Tool setting gauge position X on X axis: X coordinate of the tool setting gauge in the machine coordinate system.

8) Tool setting gauge position Y on Y axis: Y coordinate of the tool setting gauge in the machine coordinate system.

9) Start point R on Z axis: The distance (a positive value) from the tool nose to the tool setting gauge when the tool is moved at the measured speed.

10) Tool length estimation L: The distance (a positive value, metric: mm/min; inch: inch/min) from the tool nose to the spindle end face.

11) Safety height of the tool setting gauge on Z axis: The safe position from the spindle end face to the tool setting gauge plane.

12) Select <AUTO> mode, press soft key【AUTO MEAS】 to enable the system to call the tool setting macro program automatically, then press key 

to execute Auto Tool Setting.

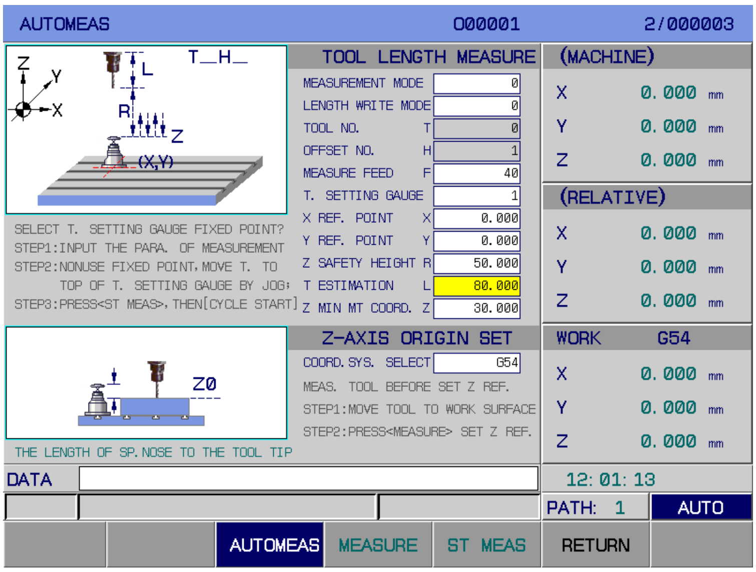


Fig. 3-4-2-2-8

Non-reference tool setting

After the reference tool setting is finished, the system set the Measurement Selection Mode to 1. Users can perform non-reference tool setting after changing the reference tool: Measure the length of the non-reference tool and write it in the tool length estimation L, select <AUTO> mode and press 【AUTO MEAS】 to enable the system to call the tool setting macro program automatically, then press key 

to execute the tool setting.

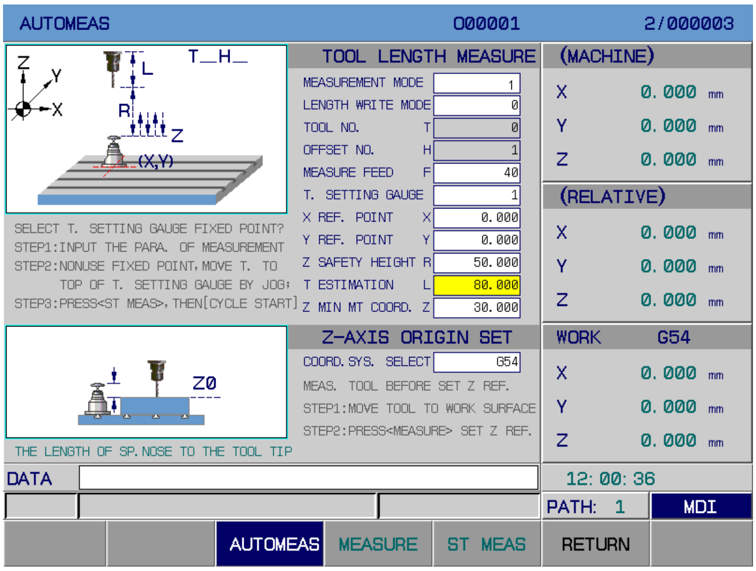
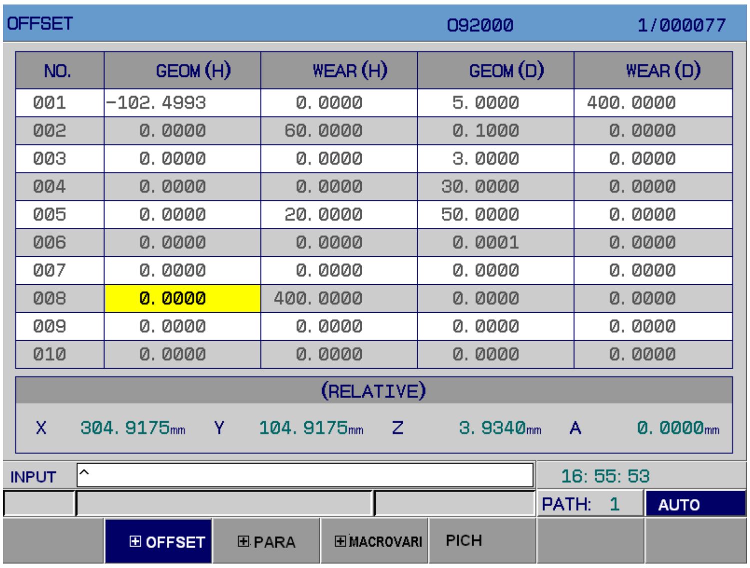


Fig. 3-4-2-2-9

The length difference between a tool and reference tool is set to the tool offset H.



OFFSET O92000 1/O00077

NO.	GEOM (H)	WEAR (H)	GEOM (D)	WEAR (D)
001	-102.4993	0.0000	5.0000	400.0000
002	0.0000	60.0000	0.1000	0.0000
003	0.0000	0.0000	3.0000	0.0000
004	0.0000	0.0000	30.0000	0.0000
005	0.0000	20.0000	50.0000	0.0000
006	0.0000	0.0000	0.0001	0.0000
007	0.0000	0.0000	0.0000	0.0000
008	0.0000	400.0000	0.0000	0.0000
009	0.0000	0.0000	0.0000	0.0000
010	0.0000	0.0000	0.0000	0.0000

(RELATIVE)

X 304.9175mm Y 104.9175mm Z 3.9340mm A 0.0000mm

INPUT ^ 16: 55: 53 PATH: 1 AUTO

OFFSET PARA MACROVARI PICH

Fig. 3-4-2-2-10

Note: After changing a tool, it is required to perform tool length measurement again. Only in this way, the correct length can be offset to the machining program.

3. Z axis workpiece origin setting

Note: Before setting the Z axis workpiece origin, please make sure that automatic tool length measurement has been performed to the current tool, or machining mistakes, tool and equipment damage or even personal injury may occur.

1. Coordinate system selection:

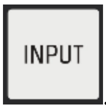
1) Setting range: G54~G59 G54 P1~P50

2) Data input: After automatic tool length measurement, in any mode, move the cursor to the

coordinate system selection item, then input the data in the following format:

a. An integer from 54~59

b. G54~G59；

c. P1~P50, then press key 

.

For example, by inputting “G55”, the system calls the workpiece coordinate system G55 automatically.

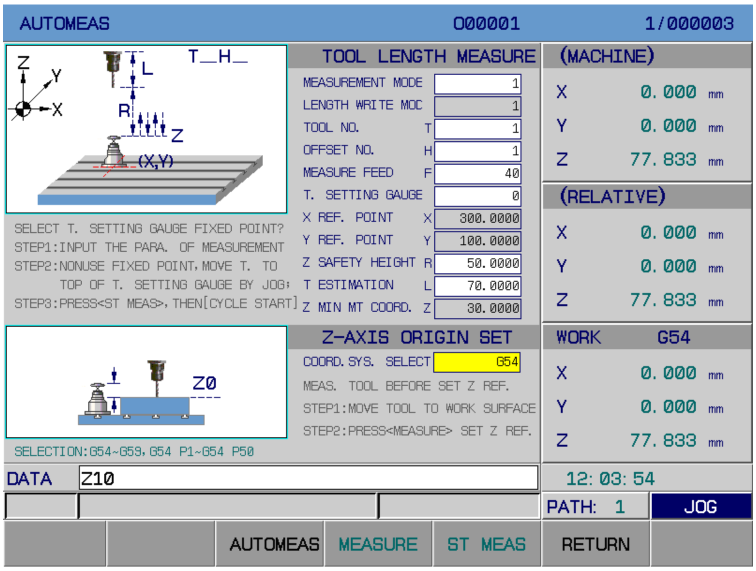


Fig. 3-4-2-2-11

2. Workpiece origin setting

Setting range: -9999.999~9999.999

Data input: After finishing automatic tool length measurement, in any mode, move the cursor to the coordinate system selection item, press soft key 【MEASURE】 directly to set the current machine coordinate value of Z axis to the Z axis of the current selected workpiece coordinate system.

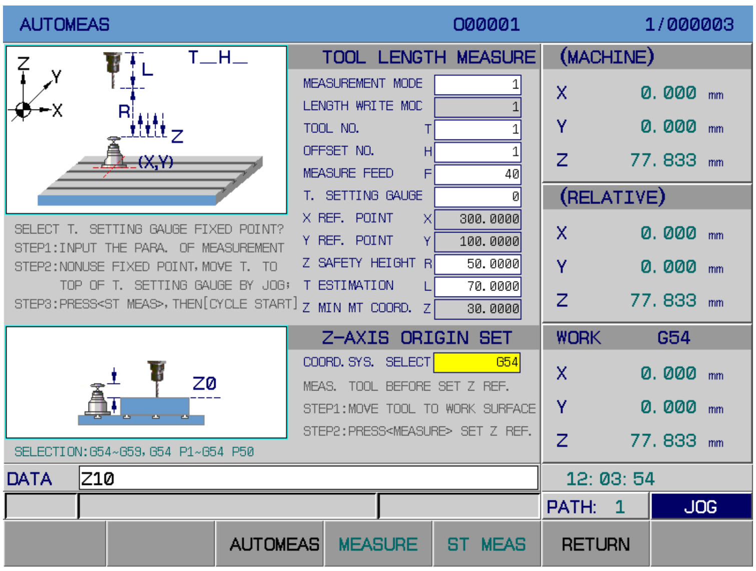


Fig. 3-4-2-2-12

Or input the data in the following format:

Input format: Z+data

Then press soft key 【MESAURE】 to set the current machine coordinate value of Z axis + input data to the Z axis of the current selected workpiece coordinate system.

Example: Input "Z10",

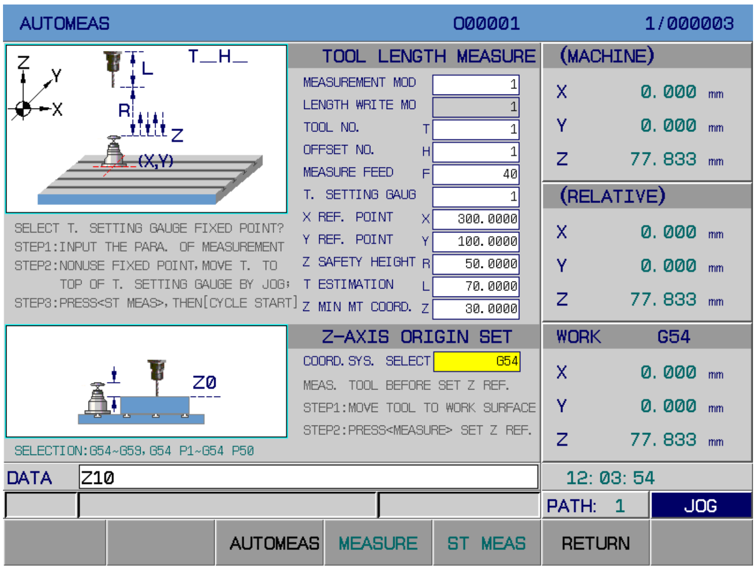


Fig. 3-4-2-2-13

3.4.3 Backup, restoration and transmission for data

Press soft key【⊞DATA】to enter SETTING (DATA DEAL) page. The user data (such as ladder, ladder parameters, system parameter values, tool offset values, pitch offset values, system macro variables, custom macro programs and CNC part programs) can be backup (saved) and restored (read); and the data input and output via PC or U disk are also available in this system. The part programs saved in CNC are not affected during the data backup and restoration.（See Fig.3-5-6-1）

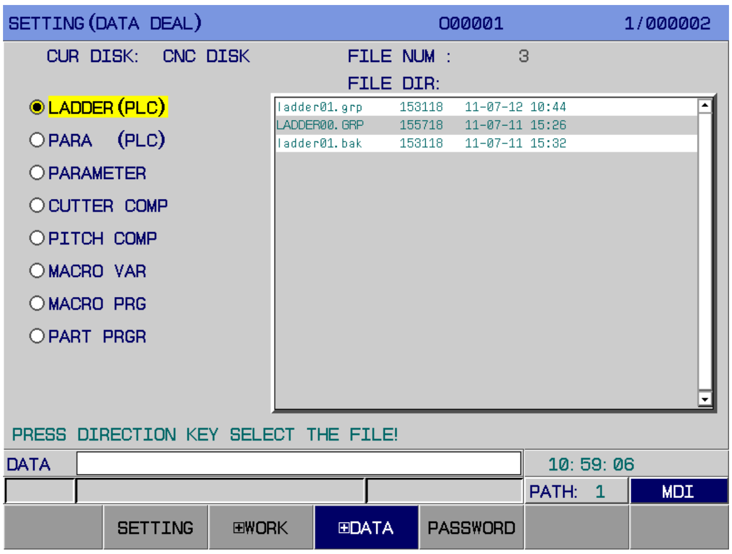


Fig. 3-4-3-1

Operation:

1. Set the password for a corresponding level in password page pressing soft key 【PASSWORD】. The corresponding password levels of the data are shown as follows:

Table 3-4-3-2

Data	Password authority
Ladder (PLC), parameter (PLC), All parameters	Password for machine tool builder level, password for system manufacturer level
System parameters, pitch offset values	Password for machine tool builder level, password for system manufacturer level, password for system debugging level
Custom macro	Password for machine tool builder level, password for system manufacturer level, password for system debugging level, password for end user level
Tool offset values system macro variables, CNC part programs	No password required during data output/input; The password for end-user level or above is required during one key output/one key input.

2. Press soft key【⊞DATA】twice to enter the DATA DEAL page, as is shown below:

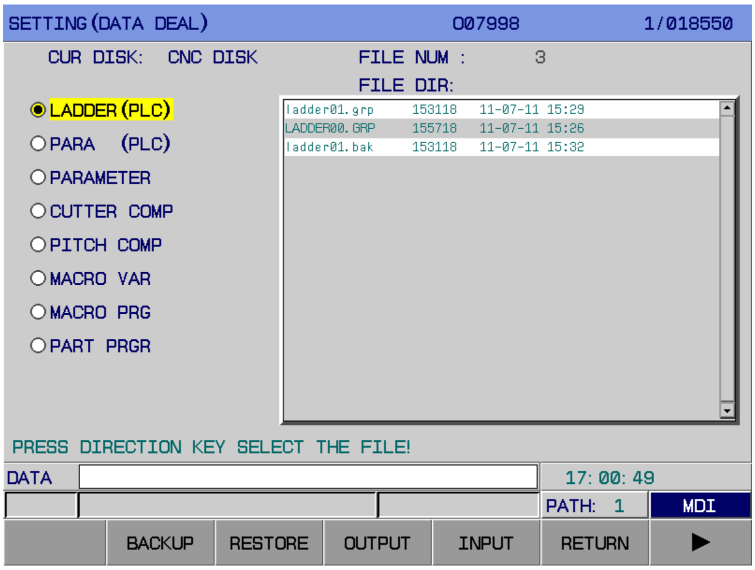


Fig. 3-4-3-3

Press【▶】to enter the next page

The functions of the operations are shown in the table below (table **3-4-3-4**):

Table **3-4-3-4**

Operation item	Explanation
Data backup	It is available to backup the data saved in the system disk such as ladder (PLC), parameters (PLC), system parameter values, tool offset values, pitch offset values, and system macro variables separately. After the backup, the system will create a backup file with file extension .bak.

Data restoration	It is available to restore the data saved in the system disk such as ladder (PLC), parameters (PLC), system parameter values, tool offset values, pitch offset values, or system macro variables separately. The operation reads the backup file saved in the system firstly and then recovers the data.
Data output	This operation can output the data saved in the system disk to the external storage devices.
Data input	This operation can input the data saved in the external storage devices to the system disk.
One key backup (OTB)	It can backup a variety of data items to the system disk simultaneously.
One key restoration (Ghost)	It can restore the backup files of multiple data items simultaneously.
One key output	It can copy multiple data items saved in the system disk to a U disk simultaneously.
One key input	It can copy multiple data items to the system disk from a U disk simultaneously.

3. Press ↑ and ↓ to select the target file, press ← and → to switch between data item directory and file directory.

4. Press corresponding soft keys to perform operations such as backup, recovery, output, input, one key backup, one key recovery, one key output and one key input.

Note:

1) When I/O channel is set to "U Disk", the functions of soft keys Data Output and Data Input are the same.
2) When performing data output/input operation, ensure the setting for the I/O channel is correct. When using a U disk, set the I/O channel to 2; when using transmission software via PC, set the I/O channel to 0 or 1.
3) The contents of One Key Output/Input are determined by password authorities. See table 3-4-3-1 for the correspondence between data items and password authorities.
4) Related parameters

   Bit parameter N0:54#7: for setting whether one key output/input is valid for part programs in debugging-level authority or above.

   Bit parameter N0:27#0: for setting whether the editing for subprograms with program numbers from 80000-89999 is forbidden.

   Bit parameter N0:27#4: for setting whether the editing for subprograms with program numbers from 90000-99999 is forbidden.
5) There are concerned operation prompts in the system during data processing, the contents of which are shown as follows (table 3-4-3-3).

Table 3-4-3-5

No.	Prompt message	Cause	Handling
1	Once key operation completed	Operation succeeded	Transmission is completed
2	One key operation completed, system prompts: Copy after modifying parameters	The input/output operation of the macro program has been performed, but the parameters concerned in the system have not been set.	Skip the input/output operation of this file.
3	One key operation completed, system alarm: Parameters taking effect after power-off are modified.	The update for the ladder and ladder parameters has been executed, which requires power-on again.	Transmission is completed, please turn on the power again.
4	File reading failed	File error	Interrupt the input/output operation
5	File writing failed	File error	Interrupt the input/output operation
6	File copy failed	File error	Interrupt the input/output operation
7	Large file, please use DNC	The part program is greater than 4M	Interrupt the input/output operation
8	Insufficient storage capacity	The storage capacity is not enough.	Interrupt the input/output operation

6) File LADCHI**.TXT is invalid after it is transmitted to the system until the power is turned off and on again.

3.4.4 Setting and modification for password authority

To prevent the part programs and CNC parameters from malicious modification, the password authority setting is available in this GSK218MC system. It is classified into 5 levels, which are the 1st level (system manufacturer), the 2nd level (machine builder), the 3rd level (system debugging), the 4th level (end user) and the 5th level (operator) in descending sequence. The system default level is the lowest one at power-on（See Fig. 3-4-4-1）.

The 1st and the 2nd level: The modifications for state parameters, data parameters, tool offset data and PLC ladder transfer, etc. are allowed in these levels.

The 3rd level: The modifications for CNC state parameters, data parameters, tool offset data etc. are allowed in this level.

The 4th level: The modifications for CNC state parameters, data parameters, tool offset data are allowed in this level.

The 5th level: No password. Modifications for offset data, macro variables and operations using the machine operator panel are available, but the modifications for CNC state parameters and data parameters are unavailable.

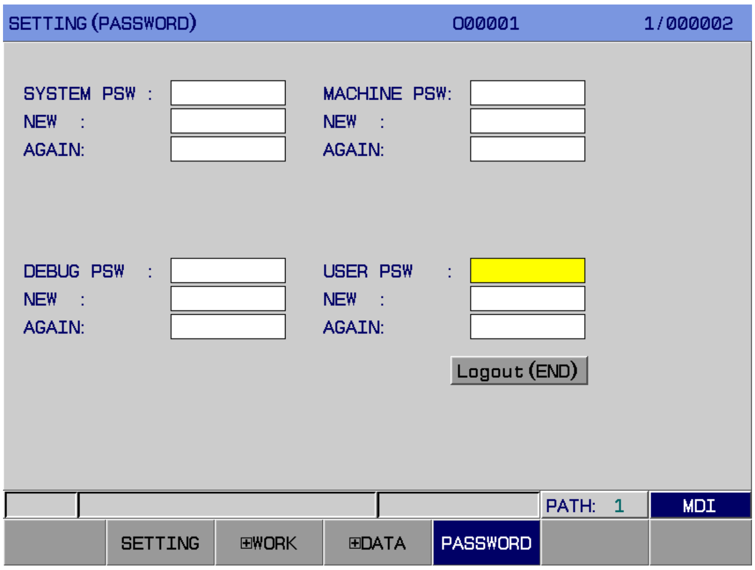
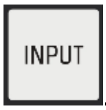
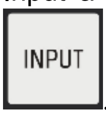
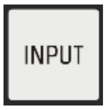


Fig. 3-4-4-1

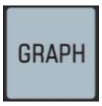
1）After entering this page in MDI mode, move the cursor to the item to be altered;

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.

3) Input a new password of 0-6 digits or letters to modify the system password, then press .

4) After modification, move the cursor to the “END” button by pressing key , then the page prompts "Press INPUT key to confirm the cancellation! "; after key  is pressed, the page prompts "Cancellation is Finished! ", and the cursor returns to the password setting item. The password is also automatically cancelled when the power is turned off.

3.5 Graphic display

Press key  to enter the graphic page which consists of two subpages:【G. PARA】and【⊞GRAPH】. They can be switched between each other by corresponding soft keys.

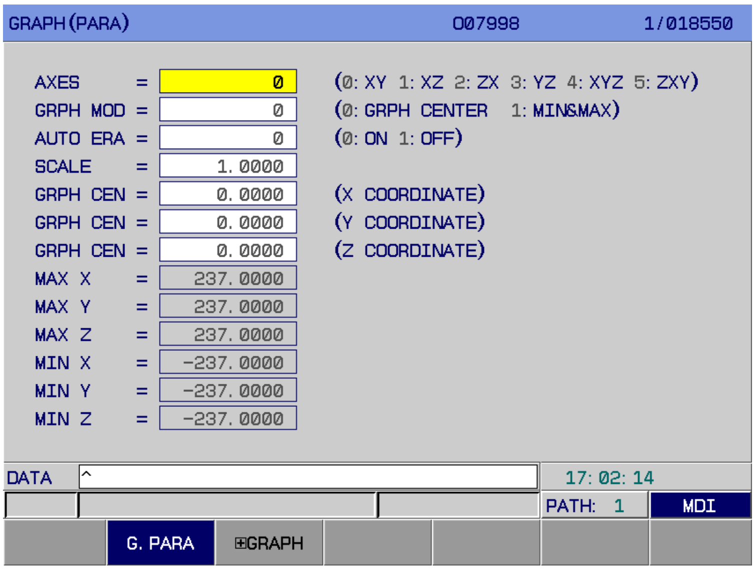


Fig. 3-5-1

1）Graphic parameter page: Press soft key【G. PARA】 to enter this page, see Fig.3-5-1.

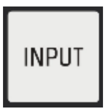
A. Graphic parameter meaning

AXIS: set drawing plane, with 6 selection modes (0-5), as shown in the next line.
Graphic mode: set graphic display mode
Automatic erasion： When it is set to 1, the program graphic is erased automatically at next cycle start-up after the program is finished.
Scale: set drawing ratio
Graphic center: set the coordinates corresponding to the LCD center in workpiece coordinate system
The 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: 0.0001mm / 0.0001inch)
Minimum value of X axis: the minimum value along X axis in graphics
(Unit: 0.0001mm / 0.0001inch)
Maximum value of Y axis: the maximum value along Y axis in graphics
(unit: 0.0001mm / 0.0001inch)
Minimum value of Y axis: the minimum value along Y axis in graphics
(Unit: 0.0001mm / 0.0001inch)
Maximum value of Z axis: the maximum value along Z axis in graphics
(Unit: 0.0001mm / 0.0001inch)
Minimum value of Z axis: the minimum value along Z axis in graphics
(Unit: 0.0001mm / 0.0001inch)

B. Setting steps for graphic parameters:

a. Move the cursor to the parameter to be set;

b. Key in the value required;

c. Press key 

to confirm it.

2) Graphic page　Press soft key【⊞GRAPH】to enter this page (See Fig. 3-5-2):

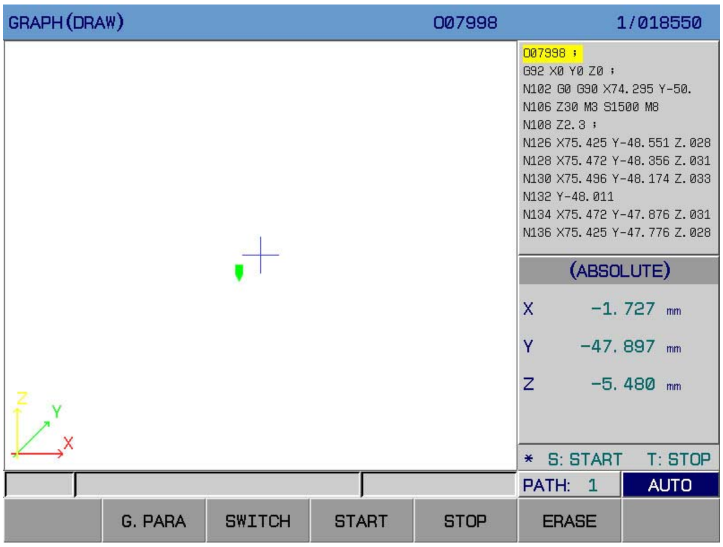


Fig. 3-5-2

The machining path of the program being executed can be monitored in graphic page.

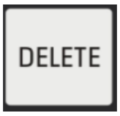
A Press soft key【START】or key 

to enter the DRAW START mode, then sign "*" is placed in front of "S: START";

B Press【STOP】soft key or key 

to enter the DRAW STOP mode, then sign '*' is moved ahead of "T: STOP";

C Press soft key 【SWITCH】 to switch the graph display among coordinates corresponding to **0**～**5**;

D Press soft key【ERASE】or key 

to erase the graph drawn.

3.6 Diagnosis display

The state of DI/DO signals between CNC and machine, the signals transferred between CNC and PLC, PLC internal data and CNC internal state etc. are displayed in the diagnosis page.

Refer to "GSK218MC CNC System Connection and PLC Manual" for the meaning and setting of each diagnosis number.

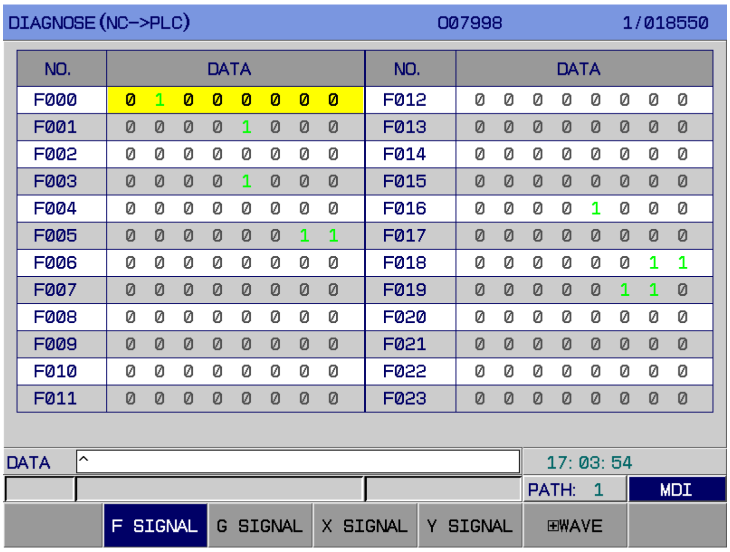
The diagnosis of this part is used to detect the running states of the CNC interface signals and internal signals rather than modifying the states.

3.6.1 Diagnosis data display

Press key 

to enter the Diagnose page, which consists of 5 subpages: 【F SIGNAL】, 【G SIGNAL】, 【X SIGNAL】, 【Y SIGNAL】 and【⊞WAVE】. All of them can also be viewed by pressing the soft keys（See Fig. **3-6-1-1** to Fig. **3-6-1-5**）.

1. F signal page　Press soft key 【F SIGNAL】in <DIAGNOSIS> page to enter diagnosis (NC→PLC) page. See figure 3-6-1-1.

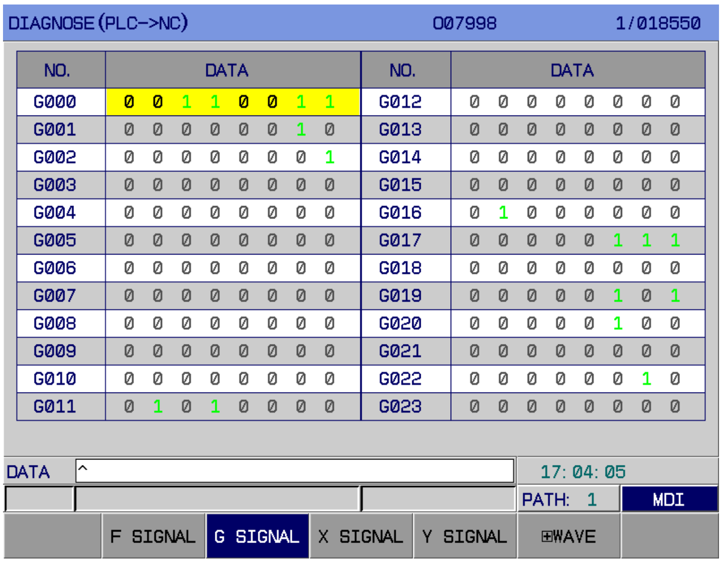


NO.	DATA	NO.	DATA
F000	0 1 0 0 0 0 0 0	F012	0 0 0 0 0 0 0 0
F001	0 0 0 0 1 0 0 0	F013	0 0 0 0 0 0 0 0
F002	0 0 0 0 0 0 0 0	F014	0 0 0 0 0 0 0 0
F003	0 0 0 0 1 0 0 0	F015	0 0 0 0 0 0 0 0
F004	0 0 0 0 0 0 0 0	F016	0 0 0 0 1 0 0 0
F005	0 0 0 0 0 0 1 1	F017	0 0 0 0 0 0 0 0
F006	0 0 0 0 0 0 0 0	F018	0 0 0 0 0 0 1 1
F007	0 0 0 0 0 0 0 0	F019	0 0 0 0 0 1 1 0
F008	0 0 0 0 0 0 0 0	F020	0 0 0 0 0 0 0 0
F009	0 0 0 0 0 0 0 0	F021	0 0 0 0 0 0 0 0
F010	0 0 0 0 0 0 0 0	F022	0 0 0 0 0 0 0 0
F011	0 0 0 0 0 0 0 0	F023	0 0 0 0 0 0 0 0

Fig. 3-6-1-1

This is the signal sent to PLC by CNC system. See “GSK218M CNC System Connection and PLC Manual” for the meaning and setting of each diagnosis number.

2. G signal page　In <DIAGNOSE> page, press soft key【G SIGNAL】 to enter diagnosis (PMC—>CNC) page, as is shown in Fig. 3-6-1-2.



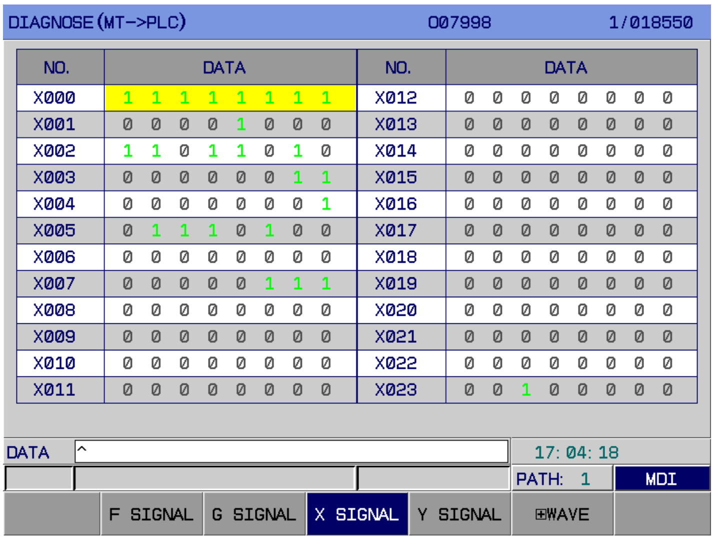
NO.	DATA	NO.	DATA
G000	0 0 1 1 0 0 1 1	G012	0 0 0 0 0 0 0 0
G001	0 0 0 0 0 0 1 0	G013	0 0 0 0 0 0 0 0
G002	0 0 0 0 0 0 0 1	G014	0 0 0 0 0 0 0 0
G003	0 0 0 0 0 0 0 0	G015	0 0 0 0 0 0 0 0
G004	0 0 0 0 0 0 0 0	G016	0 1 0 0 0 0 0 0
G005	0 0 0 0 0 0 0 0	G017	0 0 0 0 0 1 1 1
G006	0 0 0 0 0 0 0 0	G018	0 0 0 0 0 0 0 0
G007	0 0 0 0 0 0 0 0	G019	0 0 0 0 0 1 0 1
G008	0 0 0 0 0 0 0 0	G020	0 0 0 0 0 1 0 0
G009	0 0 0 0 0 0 0 0	G021	0 0 0 0 0 0 0 0
G010	0 0 0 0 0 0 0 0	G022	0 0 0 0 0 0 1 0
G011	0 1 0 1 0 0 0 0	G023	0 0 0 0 0 0 0 0

Fig. 3-6-1-2

This is the signal sent to CNC system by PLC. See “GSK218MC CNC System　Connection and PLC Manual” for the meaning and setting of each diagnosis number.

3. X signal page　Press soft key【X SIGNAL】in <DIAGNOSIS> page to enter diagnosis (MT→PLC) page, as is shown in fig. **3-6-1-3.**

Book 2　Operation

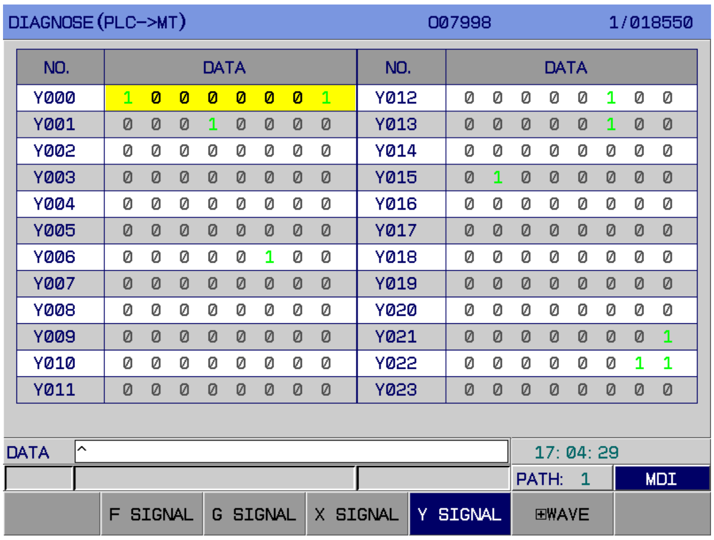

DIAGNOSE (MT->PLC) 007998 1/018550

NO.	DATA	NO.	DATA
X000	1 1 1 1 1 1 1 1	X012	0 0 0 0 0 0 0 0
X001	0 0 0 0 1 0 0 0	X013	0 0 0 0 0 0 0 0
X002	1 1 0 1 1 0 1 0	X014	0 0 0 0 0 0 0 0
X003	0 0 0 0 0 0 1 1	X015	0 0 0 0 0 0 0 0
X004	0 0 0 0 0 0 0 1	X016	0 0 0 0 0 0 0 0
X005	0 1 1 1 0 1 0 0	X017	0 0 0 0 0 0 0 0
X006	0 0 0 0 0 0 0 0	X018	0 0 0 0 0 0 0 0
X007	0 0 0 0 0 1 1 1	X019	0 0 0 0 0 0 0 0
X008	0 0 0 0 0 0 0 0	X020	0 0 0 0 0 0 0 0
X009	0 0 0 0 0 0 0 0	X021	0 0 0 0 0 0 0 0
X010	0 0 0 0 0 0 0 0	X022	0 0 0 0 0 0 0 0
X011	0 0 0 0 0 0 0 0	X023	0 0 1 0 0 0 0 0

Fig. 3-6-1-3

This is the signal sent to PLC by CNC system. See "GSK218MC CNC System Connection and PLC Manual" for the meaning and setting of each diagnosis number.

4. Y signal page Press soft key 【Y SIGNAL】 in <DIAGNOSIS> page to enter （PLC→MT） page, as is shown in fig. 3-6-1-4


DIAGNOSE (PLC->MT) 007998 1/018550

NO.	DATA	NO.	DATA
Y000	1 0 0 0 0 0 0 1	Y012	0 0 0 0 0 1 0 0
Y001	0 0 0 1 0 0 0 0	Y013	0 0 0 0 0 1 0 0
Y002	0 0 0 0 0 0 0 0	Y014	0 0 0 0 0 0 0 0
Y003	0 0 0 0 0 0 0 0	Y015	0 1 0 0 0 0 0 0
Y004	0 0 0 0 0 0 0 0	Y016	0 0 0 0 0 0 0 0
Y005	0 0 0 0 0 0 0 0	Y017	0 0 0 0 0 0 0 0
Y006	0 0 0 0 0 1 0 0	Y018	0 0 0 0 0 0 0 0
Y007	0 0 0 0 0 0 0 0	Y019	0 0 0 0 0 0 0 0
Y008	0 0 0 0 0 0 0 0	Y020	0 0 0 0 0 0 0 0
Y009	0 0 0 0 0 0 0 0	Y021	0 0 0 0 0 0 0 1
Y010	0 0 0 0 0 0 0 0	Y022	0 0 0 0 0 0 1 1
Y011	0 0 0 0 0 0 0 0	Y023	0 0 0 0 0 0 0 0

Fig. 3-6-1-4

This is the signal sent to CNC system by PLC. See "GSK218MC CNC System Connection and PLC Manual" for the meaning and setting of each diagnosis number.

5. Waveform page Press soft key【WAVE】in <DIAGNOSIS> page to enter wave page, as is shown in fig. 3-6-1-5.

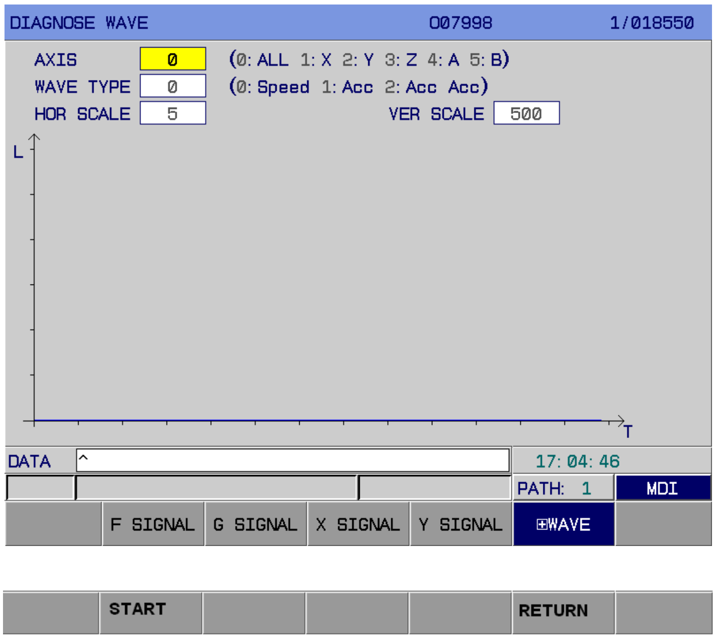
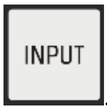


Fig. 3-6-1-5

AXIS: select the axis for WAVE diagnosis.

WAVE: select the waveform type.

HOR SCALE: select the graph ratio.

Data: In any mode, input corresponding data and press key 

.

Using key <START> to monitor signals, key <STOP> to stop monitoring signals.

3.6.2 Signal state viewing

1) Press key 

to select the DIAGNOSE page.

2) The respective address explanation and meaning are shown at the lower left corner of the screen when the cursor is moved left or right.

3） Move the cursor to the target parameter address or key in the parameter address, then press key 

to search.

4） In【WAVE】page, the feedrate, acceleration and jerk 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.

3.7 Alarm display

When an alarm is issued, "ALARM" is displayed at the lower left corner of the LCD. Press key 

to display the alarm page. There are 4 subpages: 【ALARM】, 【USER】, 【HISTORY】 and【OPERATE】, all of which can be viewed by the corresponding soft keys（See Fig.3-7-1 to Fig.3-7-4）. Whether the page is switched to alarm page when an alarm occurs can also be set by bit parameter No: 24#6.

1. Alarm page In <ALARM> page, press soft key【ALARM】to enter this page, as is shown in fig.3-7-1.

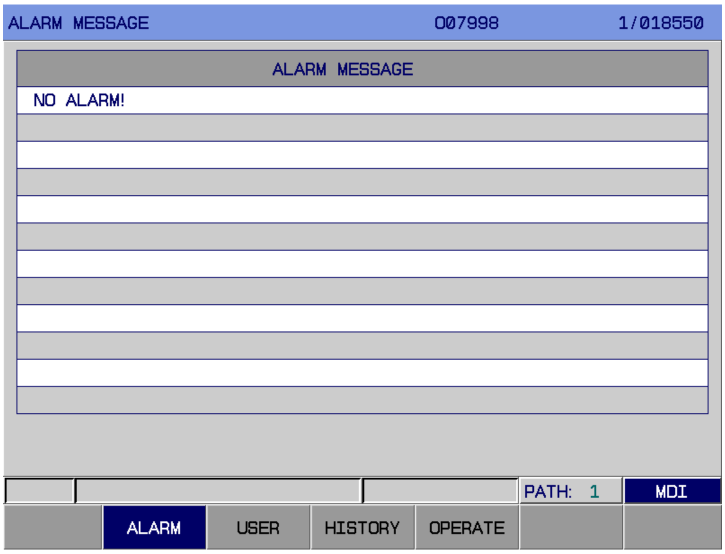


Fig. 3-7-1

In alarm page, the message of current P/S alarm number is displayed. See details about the alarm in Appendix 2.

2. User page In <ALARM> page, press soft key【USER】to enter external alarm page, as is shown in fig. 3-7-2.

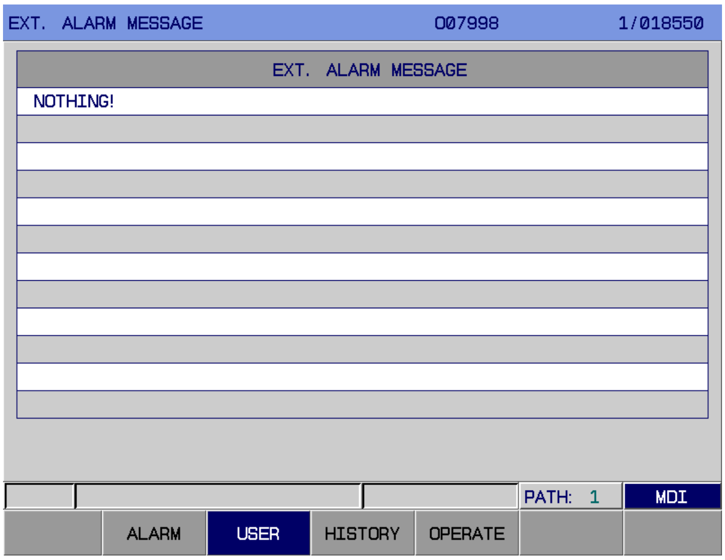


Fig. 3-7-2

See *GSK218M CNC System Connection and PLC manual* for the details about the user alarm.

Note: The external alarm number can be set and edited by users according to the site conditions. The edited contents of the alarm are input into the system via a transmission software. The external alarm is the A of edit file LadChi**.txt, and the two digits behind it are set by bit parameters 53.0~53.3. (The default is 01, i.e. the file name is LadChi01.txt)

3. History page In <ALARM> page, press soft key 【HISTORY】 to enter this page. See fig. 3-7-3:

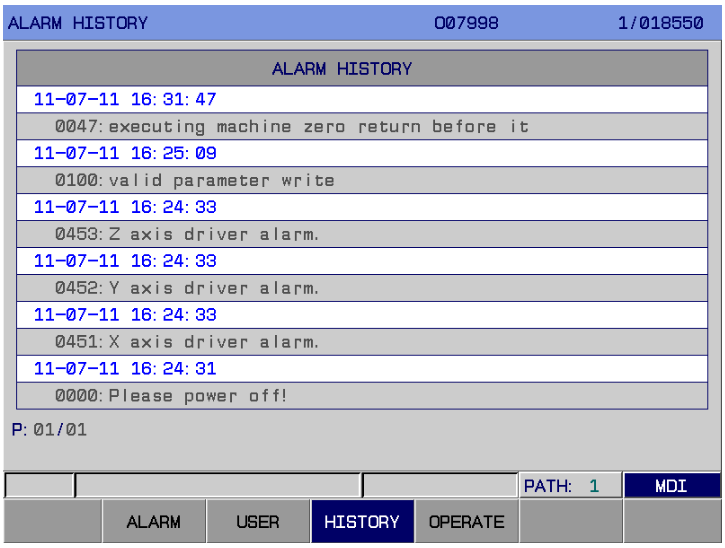


Fig. 3-7-3

In this page, the messages are arranged in chronological order for users' convenience.

4. OPERATE page In <ALARM> page, press soft key【OPERATE】to enter this page, as is shown in Fig. 3-7-4:

The OPERATE page displays the modification messages applied to the system parameters and ladders, e.g. content modification and time modification.

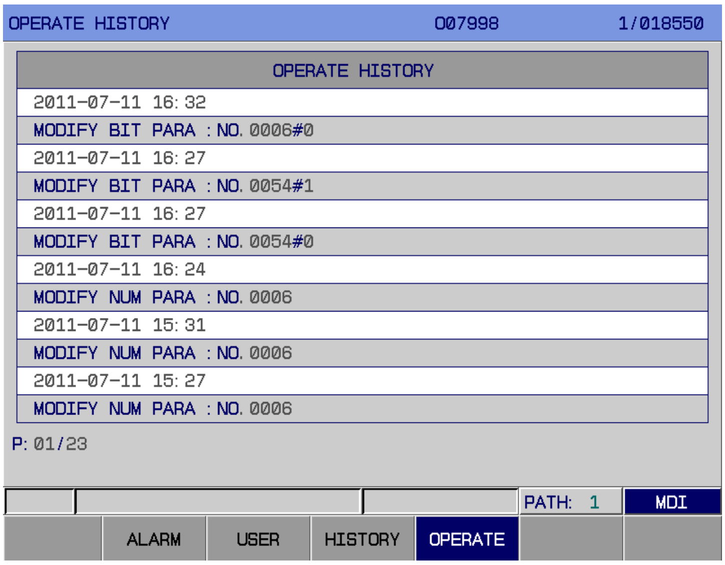
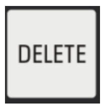


Fig. 3-7-4

OPERATE page can display 34 pages, while HISTORY alarm page can display 9 pages. The alarm time, alarm numbers, alarm messages and page numbers can be viewed using page keys.

The records of the HISTORY and OPERATE can be deleted by pressing key 

(system debugging level or above required).

3.8 PLC display

Press the key  to display the PLC page. There are 5 subpages, including 【INFO】, 【⊞PLCGRA】, 【⊞PLCPAR】, 【PLCDGN】and【⊞PLCTRA】, which can be viewed by the corresponding soft keys（See Fig.3-8-1 to Fig.3-8-5）.

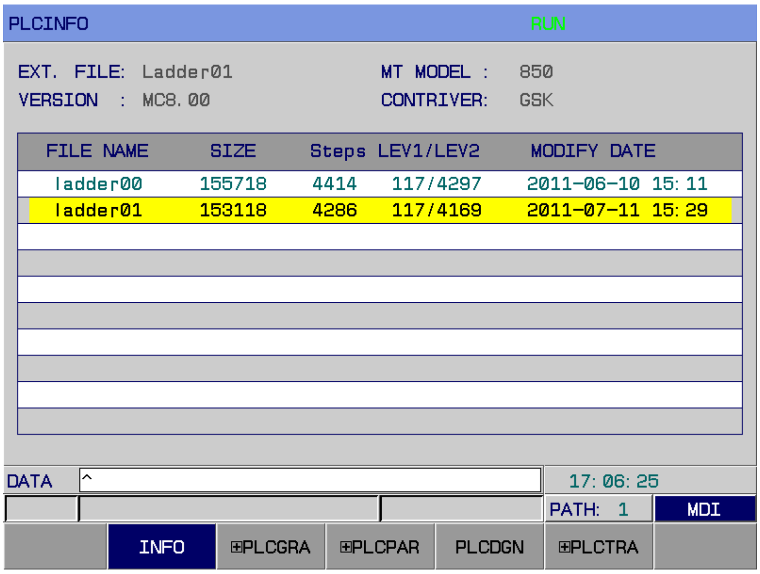


Fig. 3-8-1

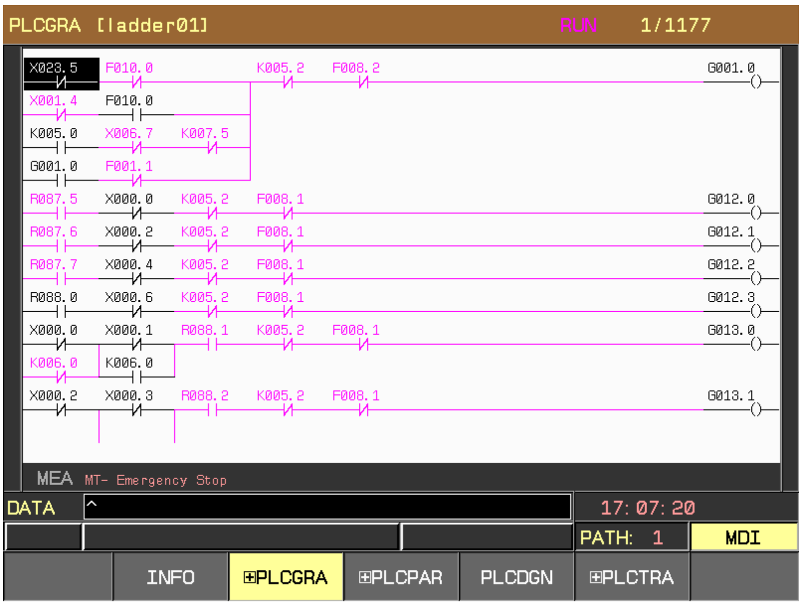


Fig. 3-8-2

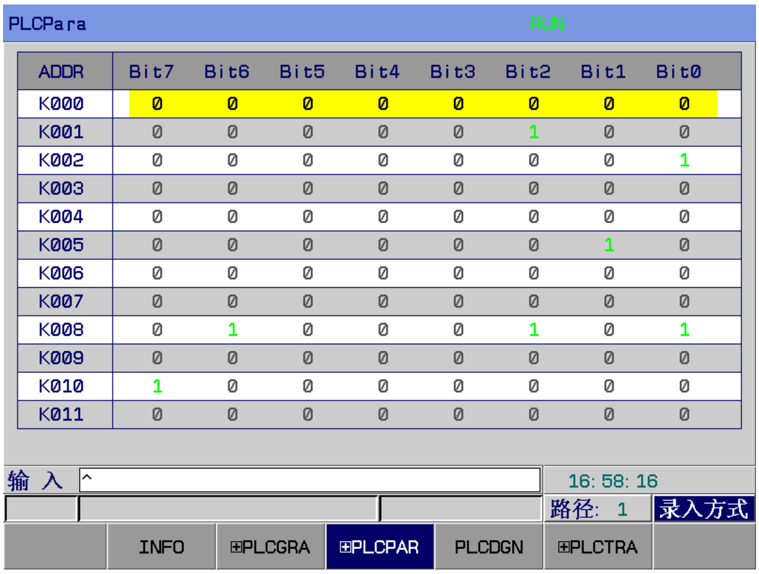


Fig. 3-8-3

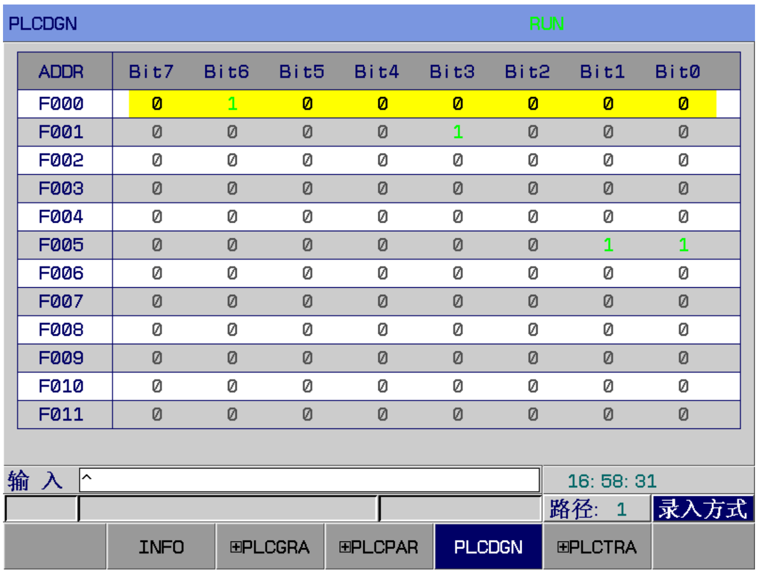


Fig. 3-8-4

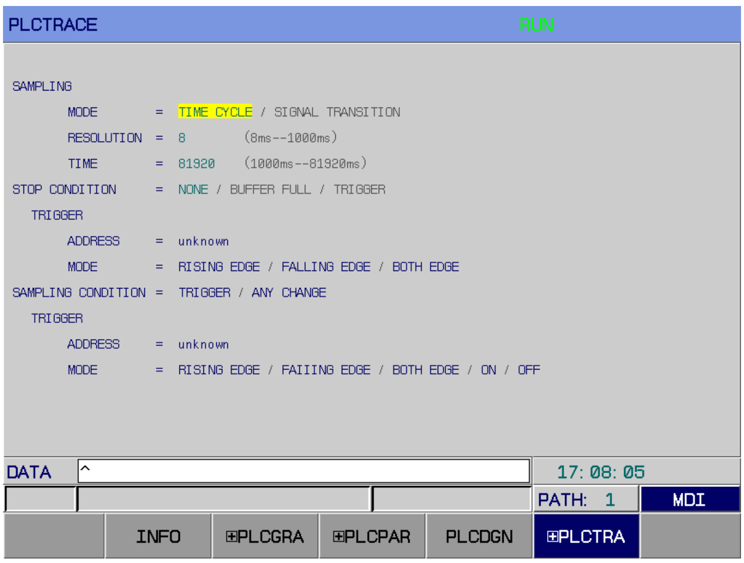


Fig. 3-8-5

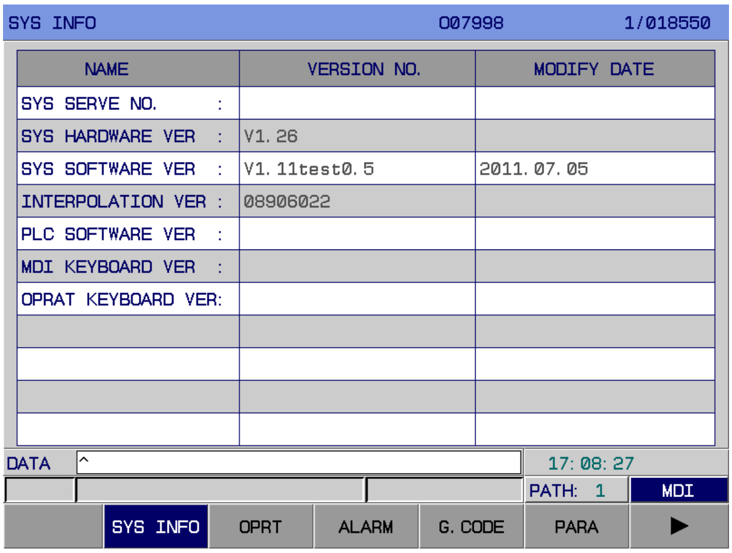
Note: Refer to *GSK218M CNC System Connection and PLC manual* for the PLC ladder modification and relevant messages.

3.9 Help display

Press key 

to display help page. There are 8 subpages, including 【SYS INFO】,【OPRT】, 【ALARM】, 【G CODE】, 【PARA】,【MACRO】,【⊞PLC.AD】and【CALCULA】. All of them can be viewed by corresponding soft keys（See Fig. 3- 9- 1～3- 9- 12）.

1. System information page In <HELP> page, press soft key【SYS INFO】to enter system information page (See fig. 3-9-1)



2. OPRT page In <HELP> page, press soft key【OPRT】to enter this page, as is shown in Fig. 3-9-2:

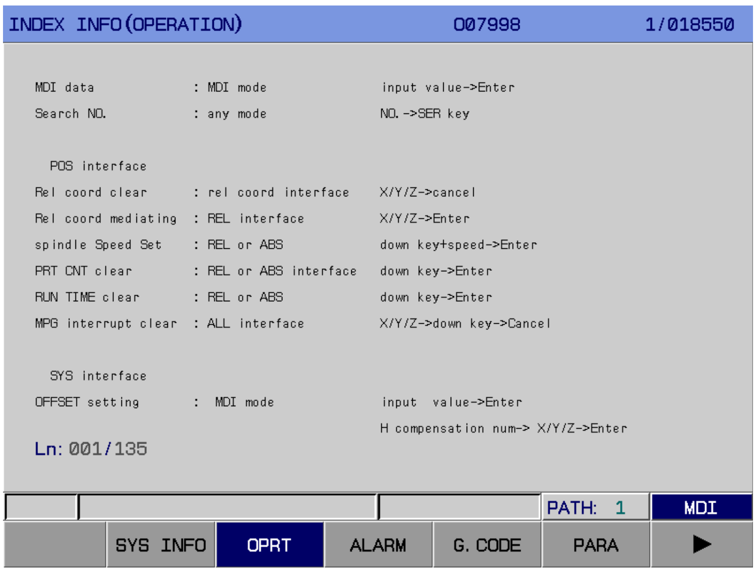


Fig. 3-9-2

The various operation steps on different pages are described in <HELP> **(**OPRT**)** page, you can get help in the HELP page if you are unfamiliar with some operations.

3. ALARM page In <HELP> page, press soft key【ALARM】to enter this page. See fig. 3-9-3:

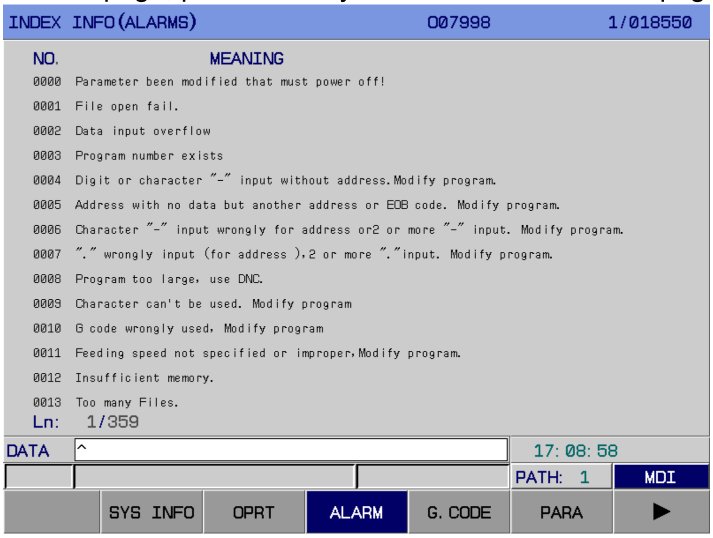
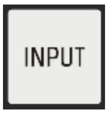


Fig. 3-9-3

The meaning and handling for each alarm number is described in this page.

4. G code page In <HELP> page, press soft key【G. CODE】to enter this page. See fig. 3-9-4:

Fig. 3-9-4

The definitions of G codes used in system are shown in G code page. Move the cursor to the G code to be viewed, then its definition is shown at the lower left corner of the page (fig. **3-9-4**). If you need to know the format and usage of a G code, press key 

on the panel after selecting the G code. Press key 

to return. See fig. 3-9-5:

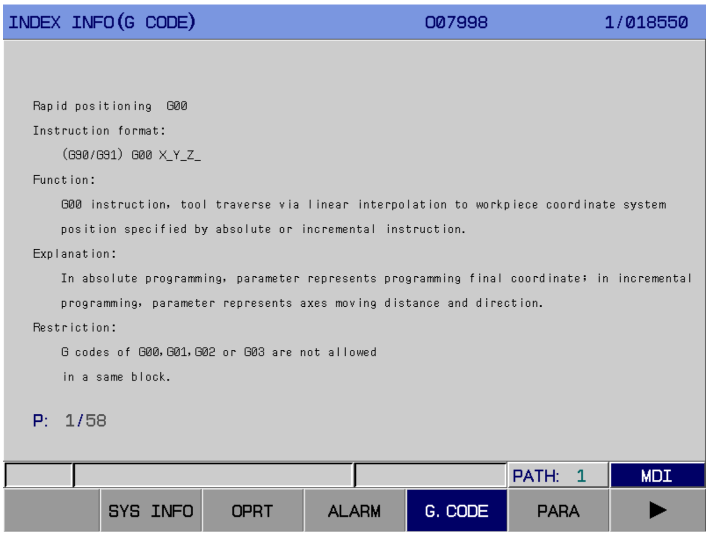


Fig. 3-9-5

The formats, functions, explanations and restrictions of instructions are introduced in this page. You can find the corresponding information here if you are unfamiliar with these instructions.

5. Parameter page In <HELP> page, press soft key【PARA】to enter this page, as is shown in Fig.3-10-5:

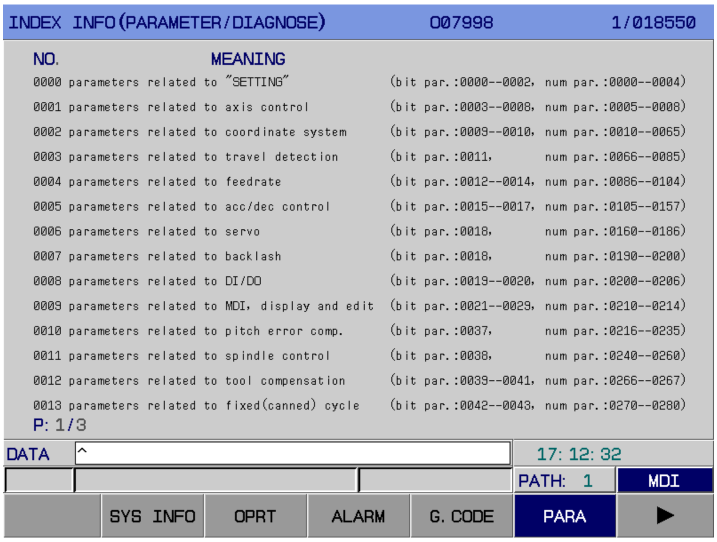


Fig. 3-9-6

The parameter setting for each function is described in the page. If you are not familiar with the setting, you can find corresponding information here.

6. Macro page In <HELP> page, press soft key【MACRO】 to enter this page, as is shown in Fig.3-10-7:

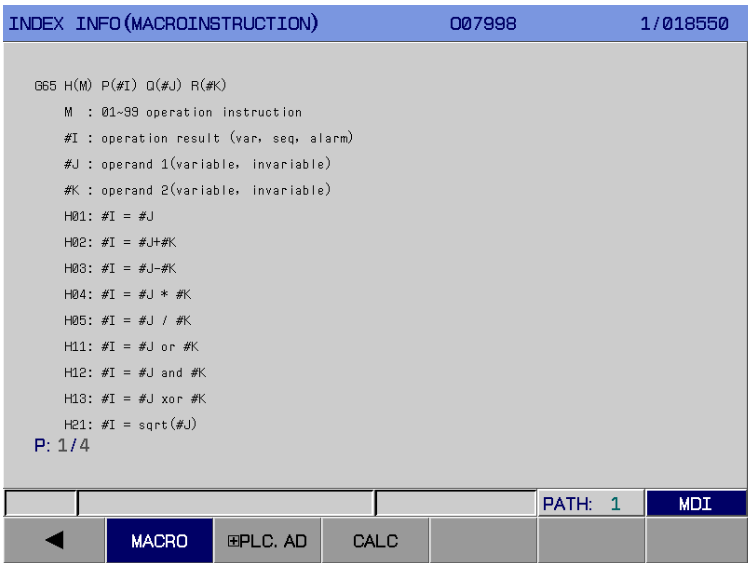


Fig.3-10-7

The formats and a variety of operation codes of the macro instructions are described in this page, and the setting ranges for local variable, common variable and system variable are also given. If you are unfamiliar with the macro instruction operations, you can get corresponding information here.

7. PLC.AD page In <HELP> page, press soft key【PLC.AD】to enter this page. There are four subpages, including【F. ADDR】,【G. ADDR】,【X. ADDR】and【Y. ADDR】, as is shown in figures 3-9-8～3-9-11:

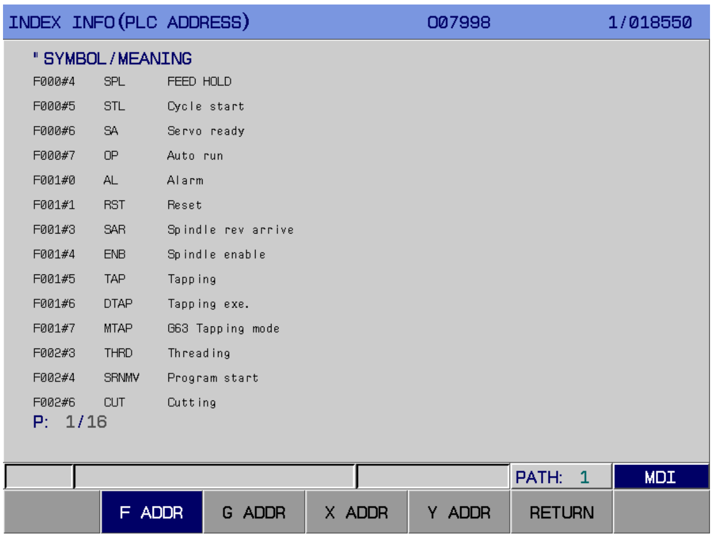


Fig. 3-9-8

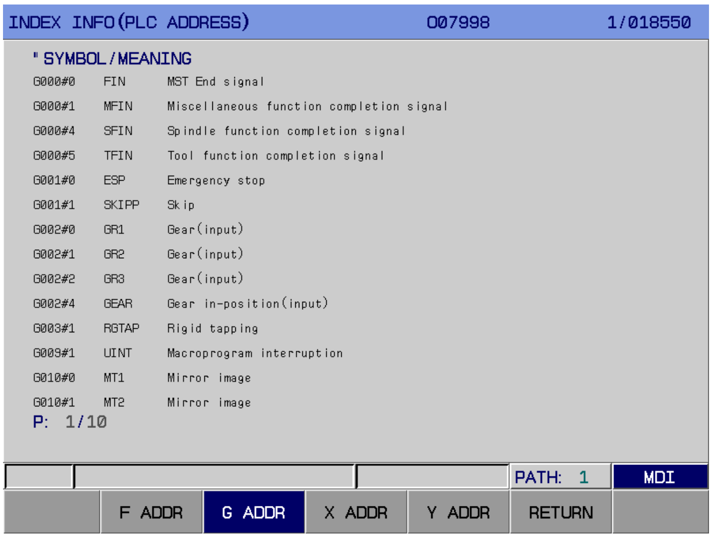


Fig. 3-9-9

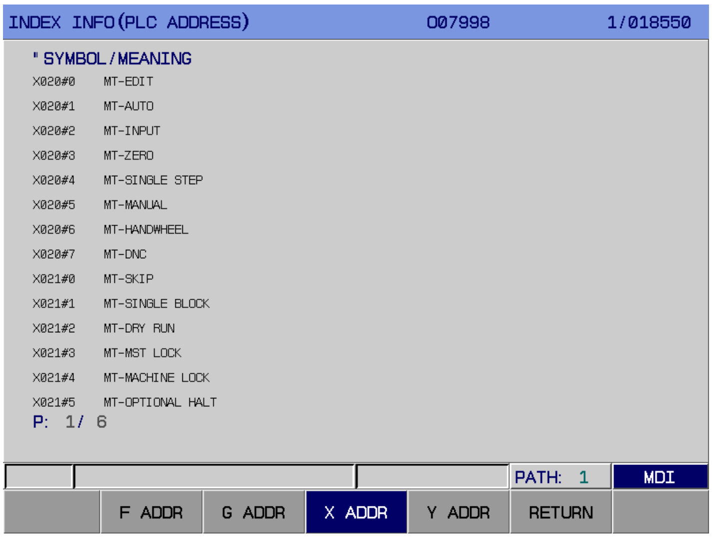


Fig. 3-9-10

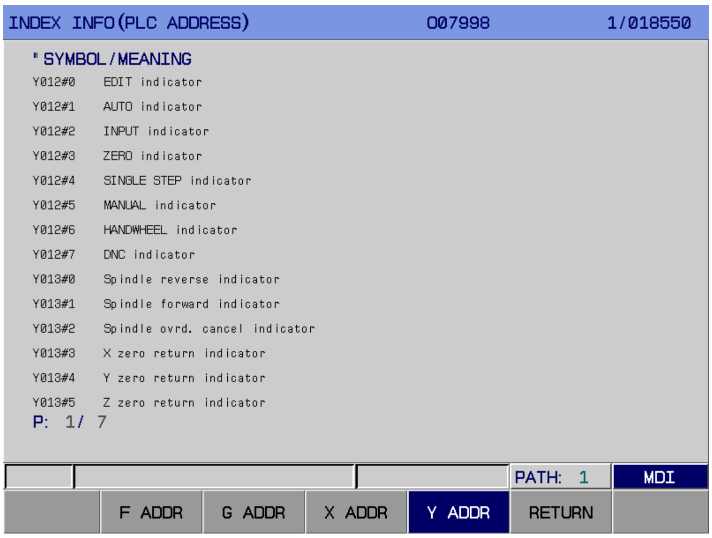


Fig. 3-9-11

The PLC addresses, signs, meanings are described in this page, and you may get the corresponding information here if you are unfamiliar with these addresses.

8. CALCULA page　In <HELP> page, press soft key【CALCULA】to enter this page. See fig. 3-9-12：

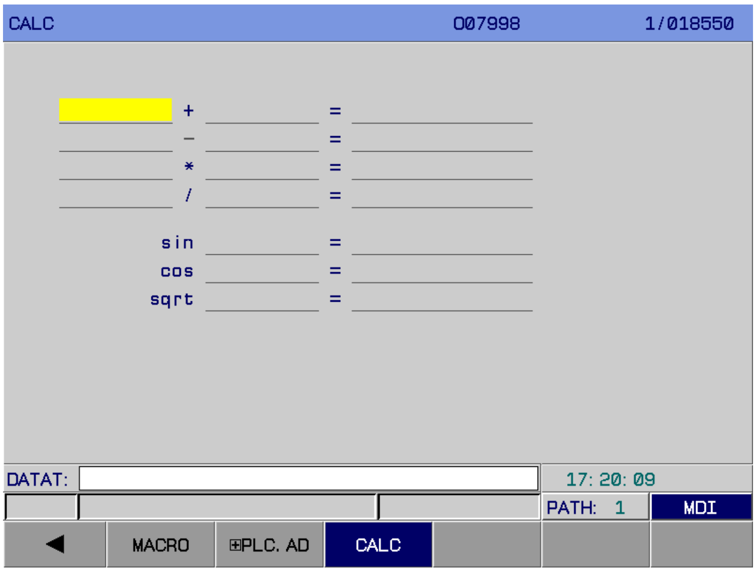
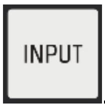


Fig. 3-9-12

The operation formats of addition, subtraction, multiplication, division, sine, cosine, extraction are shown in this page. You can move the cursor to the blank space where the data is to be input, then input the data and press key 

. After the data is input, the system will calculate automatically and output the result to the blank behind sign “=”. If the user needs to input data to calculate again, press key 

to clear all the data in the page.

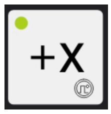
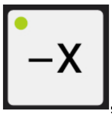
CHAPTER 4 MANUAL OPERATION

Press key  to enter Manual mode, which includes manual feed, spindle control and machine panel control, etc.

4.1 Coordinate axis movement

In Manual mode, each axis can be moved at MANUAL feedrate or manual rapid traverse speed separately.

4.1.1 Manual feed

X axis can be moved in the positive or negative direction by pressing and holding key  or key , and the feedrate can be changed by feedrate override. If the key is released, the X axis movement is stopped. That of the Y and Z axes are the same as X axis. The three axes simultaneous moving is not available in this system, but the three axes simultaneous zero return is supported by the system.

Note: The manual feedrate of each axis is set by parameter P98.

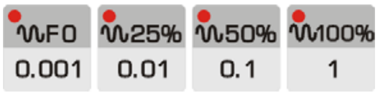
4.1.2 Manual rapid traverse

Press key  to enter Rapid Traverse state with its indicator lighting up. Then press manual feeding keys to move each axis at the rapid traverse speed.

Note 1: The manual rapid speeds are set by the parameter P170~ P173.
Note 2: Whether manual rapid traverse is effective before reference point return is set by the bit parameter N0:12#0.

4.1.3 Manual feedrate and manual rapid traverse speed selection

The manual feedrate override, which can be selected by the band switch, is divided into 21 gears (0%--200%) in MANUAL feed .

In manual rapid traverse, press keys 

to select the override of the manual rapid traverse speed. The rapid override is divided into four gears, including Fo，25％，50％ and 100％ (The speed of F0 is set by data parameter P93).

Note: 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%, the actual speed is 3m/min.

4.1.4 Manual intervention

While a program being executed in Auto, MDI or DNC mode is shifted to MANUAL mode after a dwell operation, the manual intervention is available. Move the axes manually, then shift the mode to the previous one after the intervention. When key 

is pressed to run the program, each axis returns to the original intervention point rapidly by G00, and the program execution continues.

Explanation:

1. If the single block switch is turned on during return operation, the tool performs single block stop at the manual intervention point.
2. If an alarm or resetting occurs during the manual intervention or return operation, this function will be cancelled.
3. Use machine lock, mirror image and scaling functions carefully during manual intervention.
4. Machining and workpiece shape should be taken into consideration prior to the manual intervention to prevent tool or machine damage.

The manual intervention operations are shown in the following figure:

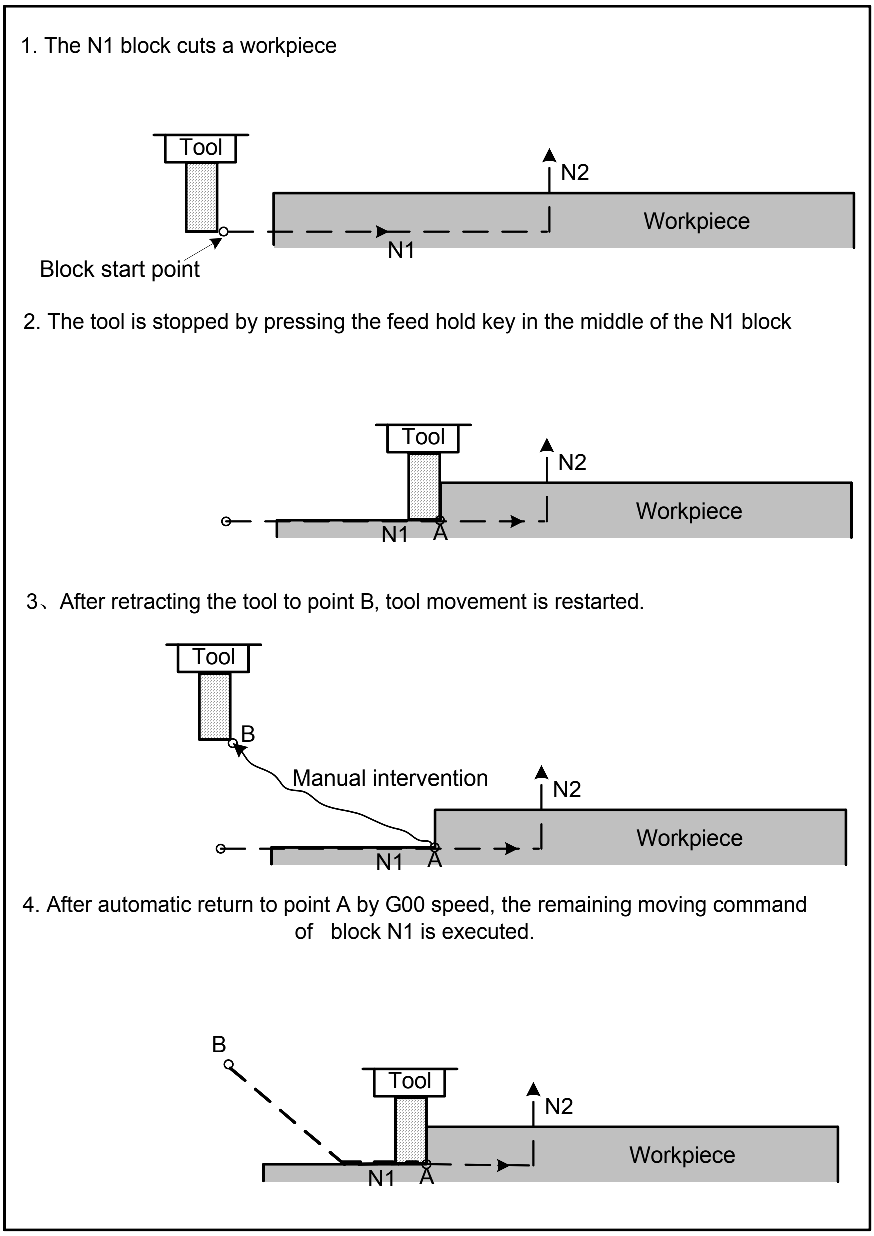


Fig. 4-1-4-1

4.1.5 Workpiece alignment

To ensure the machining precision (size, shape and position precision) and surface quality, the alignment positioning must be performed to the workpiece and fixture clamping workpiece.

The common methods for alignment are: alignment by drawing lines, alignment by trial cutting, etc. For GSK218MC system, an operation method for alignment using a tool is specially designed.

Example: Using the method for alignment by trial cutting and halving (also called halving alignment) to position the center in XY plane of a square workpiece. Operation steps are as follows:

1) Start the spindle at a certain speed.

2) Shift the system to relative coordinate display page. First perform alignment in X direction: Operate each moving axis and position them to X positive direction side of the workpiece in Manual mode, move down Z axis to make the tool nose position lower than the workpiece surface, and then move the tool towards the negative direction of the workpiece at a low speed

(usually using MPG feed mode), stop the tool when it just cuts to the workpiece. Here, press key X U on the edit panel area, and then press key CANCEL to set the X coordinate to 0. (Use the same method to set X coordinate to other values, e.g. input “x20” and press key INPUT)

3) Similarly, move the tool to the negative direction side of the workpiece, and press key X U after positioning, then press key INPUT to complete halving operation. Note that halving setting does not change the absolute coordinates and machine coordinates.

4) Move the tool to the position where the relative coordinate of the axis is 0. The position is the center in X direction.

5) In the “SETTING” page, select “WORKPIECE COORDINATE” subpage, press key X U and then key INPUT to finish the zero point setting for X axis.

6) At the center (i.e. the positioned point where the relative coordinates of X and Y are 0 on the machine) of XY, the floating coordinate system can be established by G92, and the XY machine coordinates of this point can also be written to the parameters of G54～G59 workpiece coordinate systems for system use.

7) Then the operation using trial cutting and halving method to align the center of the square workpiece is finished.

With the assignment for the relative coordinate and halving function setting, the assignment speed is increased and the operation is more convenient.

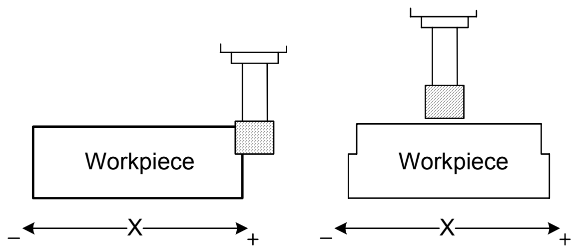


Fig. 4-1-5-1

Note 1: This system can only set and input the coordinates displayed at the relative position. (All the places where the offset value is modified can set the positions of the relative coordinates)

Note 2: Bearing operation function. The displayed coordinates can be set after addition or subtraction operation is performed to it.

Note 3: After the coordinate system is set, the coordinate system set by G92 will be lost due to mechanical zero return or G54~G59 workpiece coordinate system calling, but the one of which the machine coordinates are written to the G54～G59 workpiece coordinate systems by parameters will not be lost. It is recommended to use the latter method.

4.2 Spindle control

4.2.1 Spindle CCW

: Specifies S speed in MDI mode; in Manual/MPG/Step mode, press this key to rotate the spindle counterclockwise

4.2.2 Spindle CW

: Specifies S speed in MDI mode; in Manual/MPG/Step mode, press this key to rotate the spindle clockwise

4.2.3 Spindle stop

: In Manual/MPG/Step mode, press this key to stop the spindle.

4.2.4 Spindle automatic gear shift

Whether the spindle is frequency conversion control or gear control is set by bit parameter No:1#2. If parameter No:1#2=1, the spindle auto gear shift is controlled by PLC. Three gears (gear 1 to gear 3) are available in this system, and the maximum speed of each gear is set by parameters（P246,P247and P248）respectively. The corresponding gear can be output by modifying the ladder. In MANUAL or Auto mode, the increase or decrease for the corresponding spindle gear can be adjusted for the spindle CCW or CW rotation by pressing positive/negative override keys. In MDI mode, the system will automatically select the corresponding gear after the specified speed is input.

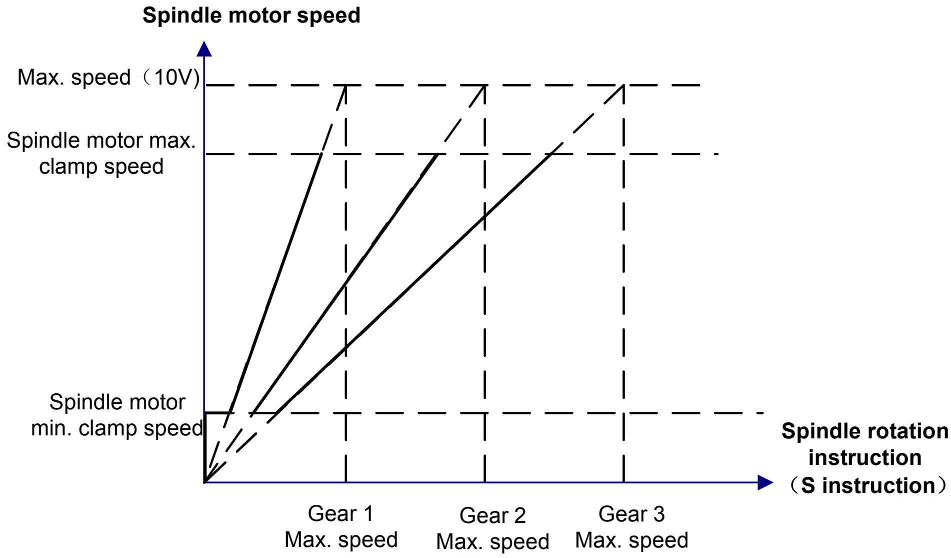


Fig. 4-2-4-1

Note: When the spindle auto gear shift 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 Coolant control

: A compound key, used to switch between coolant ON and OFF. ON: the indicator lights up; OFF: the indicator goes out.

4.3.2 Lubricant control

: A compound key, used to switch between lubricant ON and OFF. ON: the indicator lights up; OFF: the indicator goes out.

4.3.3 Chip removal control

: A compound key, used to switch between chip removal ON and OFF. ON: the indicator lights up; OFF: the indicator goes out.

Book 2 Operation

4.3.4 Working light control

: A compound key, used to switch between working light ON/OFF. ON: the indicator lights up; OFF: the indicator goes out.

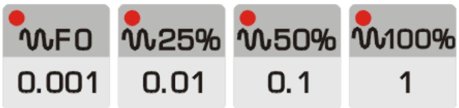
CHAPTER 5 STEP OPERATION

5.1 Step feed

Press key 

to enter the STEP mode. In this mode, the machine moves by the step defined by the system each time.

5.1.1 Selection of moving amount

Press any of keys 

to select a moving increment, then the increment will be shown on the screen. E.g. If key 

is pressed, a step of 0.100 (See Fig. 5-1-1-1) is displayed in <POSITION> page:

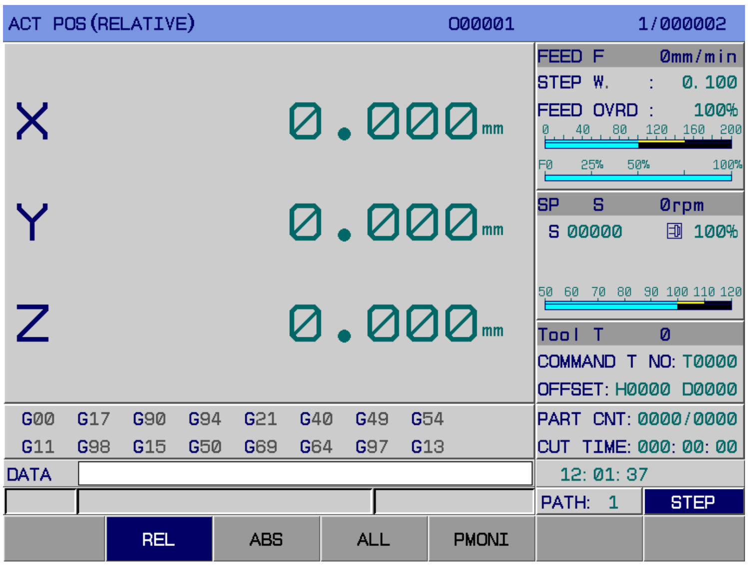
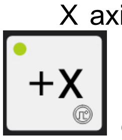
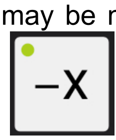


Fig. 5-1-1-1

By press moving key each time, the corresponding axis on the machine is moved 0.1 mm.

5.1.2 Selection of moving axis and direction

X axis may be moved in the 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. The operation for Y or Z axis is identical with that of X axis. Simultaneous manual moving for 3 axes is unavailable in this system, but simultaneous zero return for 3 axes is available.

5.1.3 Step feed explanation

The step feed max. clamp speed is set by data parameter P155.

The step feedrate is beyond the control of the feedrate and rapid override.

5.2 Step interruption

While the program running in Auto, MDI or DNC mode is shifted to Step mode after a dwell operation, the control will execute the step interruption. The coordinate system of step interruption is consistent with that of MPG, and its operation is also the same as that of MPG (MPG for manual pulse generator, i.e. handwheel, similarly hereinafter). See Section 6.2 Control in MPG Interruption for details.

5.3 Auxiliary control in Step mode

It is the same as that of Manual mode. See Sections 4.2 and 4.3 in this manual for details.

CHAPTER 6　MPG OPERATION

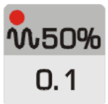
6.1　MPG feed

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 on the position page if any of keys 

is pressed, e.g. if key 

is pressed, the MPG increment: 0.100（See Fig.6-1-1-1）is displayed in <POSITION> page:

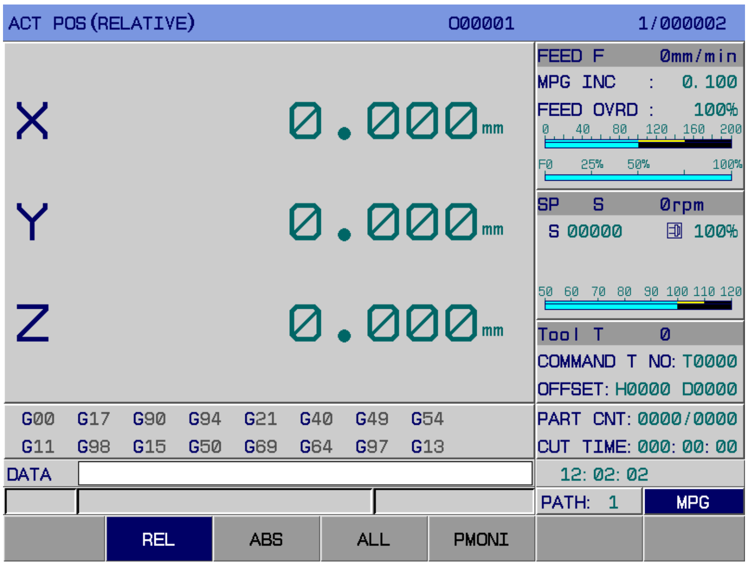
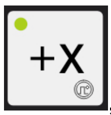


Fig. 6-1-1-1

6.1.2　Selection of moving axis and direction

In MPG mode, select the moving axis to be controlled by the handwheel, and press the corresponding key, then you can move the axis by the handwheel.

In MPG mode, if X axis is to be controlled by the handwheel, press key 

, then you can move the X axis by rotating the handwheel.

The feed direction is controlled by handwheel rotation direction. See the manual provided by the machine tool builder for details. In general, handwheel CW rotation indicates the positive feed, while CCW rotation indicates the negative feed.

6.1.3 MPG feed explanation

1. The relationship between handwheel scale and machine moving amount is as follows:

Table 6-1-3-1

	Moving amount per MPG scale		
MPG increment (mm)	0.001	0.01	0.1
Machine moving amount (mm)	0.001	0.01	0.1

2. The values in the table above vary with the mechanical transmission. See the manual provided by the machine tool builder for details.
3. The rotation speed of the handwheel cannot exceed 5r/s, otherwise, the scale and the moving amount may be inconsistent.

6.2 Control in MPG interruption

6.2.1 MPG interruption operation

The MPG interruption operation can overlap the automatic movement in Auto mode.

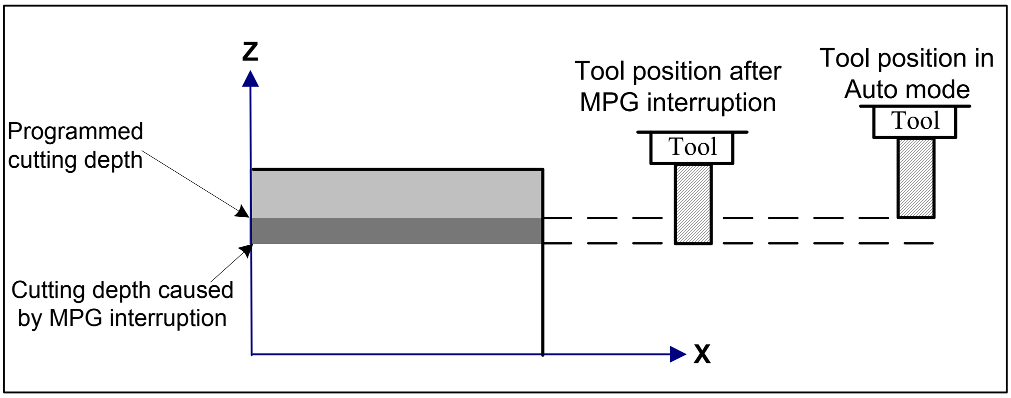


Fig. 6-2-1-1

The operations are as follows:

1）After the dwell operation, switch the program being executed in Auto mode to MPG mode.

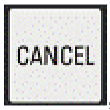
2）Move the tool by the handwheel to modify the coordinate system, such as moving Z axis upward and downward, moving X and Y axes horizontally, or rotating A axis.

3）After the control is switched to Auto mode, the workpiece coordinates remain unchanged till the machine zero return operation is performed again. After the operation, the coordinates restore to their actual values.

Note: Whether MPG/Step interruption function is used is set by bit parameter NO:56#3.

As the program being executed in Auto, MDI or 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.6-2-1-2.

Fig. 6-2-1-2

Steps to clear MPG interruption coordinate system: Press key X, move the cursor upward and downward till the MPG interruption coordinate X flickers, and press key 

, then the coordinate system is cleared. The operations for Y and Z axes are the same as above; when the zero return operation is performed, the coordinate system is cleared automatically too.

Note: When the MPG interruption function is used to adjust the coordinate system, if an alarm or resetting occurs, the function is cancelled.

6.2.2 Relationship between MPG interruption and other functions

Table 6-2-2-1

Display	Relationship
Machine lock	After machine lock is effective, the machine movement by using MPG interruption is ineffective.
Absolute coordinate value	MPG interruption does not change the absolute coordinate values.
Relative coordinate value	MPG interruption does not change the relative coordinate values.
Machine coordinate value	The change amount of the machine coordinate value is the displacement amount caused by MPG rotation.

Note: The moving amount of MPG interruption is cleared when the manual reference point return is performed for each axis.

6.3 Auxiliary control in MPG mode

The auxiliary operation in MPG mode is identical with that in JOG mode. See Sections 4.2 and 4.3 for details.

6.4 Electronic MPG drive function

Operation method:

Enable the electronic MPG drive function by setting bit parameter NO:59#1. In Auto mode, turn on Dry Run, press key <CYCLE START>, and control the execution of the part program by rotating the MPG. The execution speed of the program becomes faster as the MPG is rotated faster, and vice versa. This function is usually used for workpiece trial cutting and machining program detection.

Note 1: The Dry Run is ineffective after the electronic MPG drive function is enabled.
Note 2: Single block stop execution is effective in single block mode.

CHPATER 7 AUTO OPERATION

7.1 Selection of the auto run programs

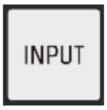
1. Program loading in auto mode

(a) Press key 

to enter the Auto mode;

(b) Press key 

to enter the 【DIR】page, move the cursor to find the target program;

(c) Press key 

for confirmation.

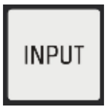
2. Program loading in Edit mode

(a) Press key 

to enter the Edit mode;

(b) Press key 

to enter the【DIR】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 selecting the program using the two methods in section 7.1 above, press key 

to execute the program automatically. The execution of the program can be viewed by switching to <POSITION>, <MONI>, <GRAPH> etc. pages.

The program execution is started from the line where the cursor is located, so it is recommended to check whether the cursor is located at the program to be executed and whether the modal values are correct before pressing key 

. If the cursor is not located at the start line from which the program is started, press key 

, and then key 

to run the program automatically from the start line.

Note: The workpiece coordinate system and reference offset values cannot be modified during program execution in Auto mode.

7.3 Auto run stop

In Auto run, to stop the program being automatically executed, the system provides five methods:

1. Program stop (M00)

After the block containing M00 is executed, the auto running pauses and the modal message is saved. After key 

is pressed, the program execution continues.

2. Program optional stop (M01)

If key 

is pressed before the program execution, the automatic running pauses and the modal message is saved when the block containing M01 is executed in the program. After key 

is pressed, the program execution is continued.

3. Pressing key 

If key 

is pressed during the auto running, the machine states are as follows:

1) Machine feeding slows down and stops;

2) Dwell continues if Dwell (G04 instruction) is executed;

3) The other modal message is saved;

4）The program execution continues after key 

is pressed.

4. Pressing key 

See Section 2.3.1 in this manual.

5. Pressing Emergency Stop button

See Section 2.3.2 in this manual.

In addition, if the control is switched to other mode from Auto mode, DNC mode or MDI page of MDI mode in which the program is being executed, the machine can also be stopped.

The steps are as follows:

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 MANUAL, 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 any block

This system allows the auto run to start from any block of the current program. The steps are shown as follows:

1. Press key 

to enter Manual mode, start spindle and other miscellaneous functions;
2. Execute the modal values of the program in MDI mode, and ensure the modal values are correct;
3. Press key 

to enter Edit mode, and press key 

to enter program page, then find the program to be machined in【DIR】.
4. Open the program, and move the cursor to the block to be executed;
5. Press key 

to enter Auto mode;
6. Press key 

to execute the program automatically.

Note 1: Before execution, confirm the current coordinate point is the end position of the last block (confirmation for 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 for tool change operation, etc, ensure no interference and collision occur between the current position and workpiece in a bid to prevent machine damage and personnel hurt.

7.5 Dry run

Before the machining by a program, use “Dry Run” (usually in combination with “M.S.T. Lock” or “Machine Lock”) to check the program.

Press key 

to enter Auto mode, and press key 

(that the indicator on the key lights up means Dry Run state is entered).

In rapid feed, the program speed equals to Dry Run speed × rapid feed override.

In cutting feed, the program speed equals to Dry Run speed × cutting feed override.

Note 1: The Dry Run speed is set by data parameter P86;

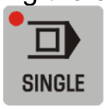
Note 2: In rigid taping, whether the Dry Run is effective is set by bit parameter NO:12#5;

Note 3: In cutting feed, whether the Dry Run is effective is set by bit parameter NO:12#6;

Note 4: In rapid positioning, whether the Dry Run is effective is set by bit parameter NO: NO:12#7.

7.6 Single block execution

"Single Block" can be selected for checking the execution of a block.

In Auto, DNC or MDI mode, press key 

(that the indicator on the key lights up means single block execution state is entered). In single block execution, the system stops after the execution of a single block. Press key 

to execute the next block, and perform the operation like this repeatedly till the whole program is executed.

Note: In G28 mode, the single block stop can be performed at an intermediate point.

7.7 Machine lock

In <AUTO> mode, press key 

(that the indicator on the key lights up means the current Machine lock state is entered). In this mode, the axes on the machine do not move, but the position along each axis changes on the display as if the tool were moving. In addition, M, S and T functions can be executed. This function is for checking a program.

Note: The machine position and coordinate position are inconsistent after key 

is pressed to execute the program. Therefore, it is required to perform machine zero return operation after the execution.

7.8 MST lock

In <AUTO> mode, press key 

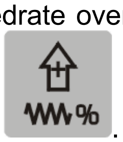
(that the indicator on the panel lights up means MST lock state is entered). In this state, M, S and T codes are not executed. This function is used together with Machine Lock to check a program.

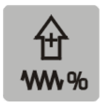
Note: M00, M01, M02, M30, M98, M99 are executed even in MST lock state.

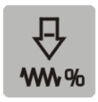
7.9 Feedrate and rapid speed override in Auto run

In <AUTO> mode, the feedrate and rapid traverse speed can be overridden by the system.

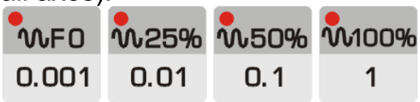
In auto run, the feedrate override, which is divided into 21 gears, can be selected by pressing keys 



. Press key 

once, the feedrate override increases by one gear (10%) till 200%; Press key 

once, the feedrate override decreases by one gear (10%). If the override is set to F_O, whether the axes are stopped is set by bit parameter NO：12#4, and If the axes are not stopped when the override is set to 0, the actual rapid traverse speed is set by data parameter P93 (common to all axes).

In auto run, press keys 

to select the rapid traverse speed with gears Fo, 25％, 50％ and 100％.

For 218MC-H and 218MC-V CNC systems, the feedrate is selected by the feedrate override band switch 

with 21 gears.

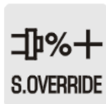
Note 1: Value specified by F in feedrate override program
The actual federate = Value specified by F X feedrate override
Note 2: The rapid traverse speed overridden by data parameter P88, P89, P90 and rapid override is calculated as follows:
Actual rapid traverse speed along X axis= Value specified by P88 X rapid override
The calculation methods for Y and Z axes are the same as that of X axis.

7.10　Spindle speed override in auto run

In auto run, the spindle speed can be overridden if it is controlled by analog quantity.

The spindle override, which is classified into 8 gears from 50％～120％, can be adjusted by pressing spindle override keys 

in auto mode.

The spindle speed override increases by one gear (10%) till 120％ by pressing key 

each time.

The spindle speed decreases by one gear (10%) by pressing key 

once. When it decreases to 50%, the spindle stops.

The spindle speed of 218MC-H/-V CNC system is overridden by spindle override band switch 

with 8 gears from 50%~120%.

The actual spindle speed=speed specified in the program × spindle override. The maximum spindle speed is set by data parameter P258. If the spindle speed exceeds it, it is taken as the actual speed.

7.11 Background edit in auto run

The background edit function during processing is supported in this system.

During the program execution in Auto mode, press key <PROGRAM> to enter the program page, then press soft key【◆PRG】to enter the background edit page, as is shown in Fig.7-11-1:

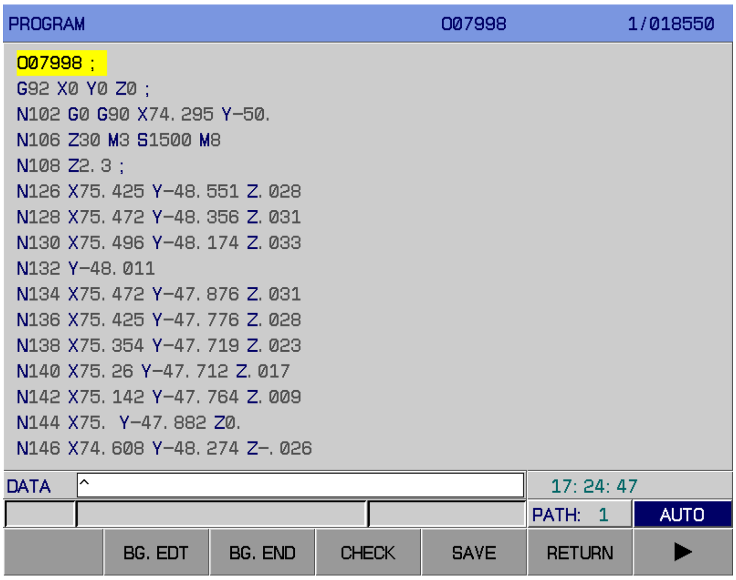


Fig. 7-11-1

Press soft key【BG.EDT】to enter the program background edit page. The program editing operation is the same as that in Edit mode (Refer to Chapter 10 PROGRAM EDIT in this manual). Press soft key【BG.END】to save the edited program and exit this page.

Note: It is suggested that the file size in background edit be not more than 3000 lines, otherwise the processing effect will be affected.

CHAPTER 8 MDI OPERATION

Besides the input and modification for parameters and offsets, the MDI operation function is also provided in MDI mode. The instructions can be input directly using this function. The data input, parameter and offset modification etc. are described in “CHAPTER 3 PAGE DISPLAY AND DATA MODIFICATION AND SETTING”. This chapter will describe the MDI operation function in MDI mode.

8.1 MDI instruction input

The input in MDI mode is classified into two types:

1. By【MDI】type , multiple blocks can be input consecutively.
2. By【CUR/MOD】type, only one bock can be input.

The input in 【MDI】type is identical with the program input in Edit mode. See “CHAPTER 10 PROGRAM EDIT” in this manual for details. The input in【CUR/MOD】type is introduced below:

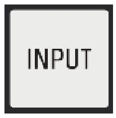
Example: Inputting a block “G00 X50 Y100” in 【CUR/MOD】page. The steps are:

1). Press key 

to enter MDI mode;

2). Press key 

to enter program page, then press soft key 【CUR/MOD】 to enter【CUR/MOD】page (See fig. **8-1-1**)

3). After inputting block “G00X50Y100” in sequence with the keyboard, press key 

for confirmation, then the program is displayed on the page;

As is shown in the figure below (Fig. 8-1-1):

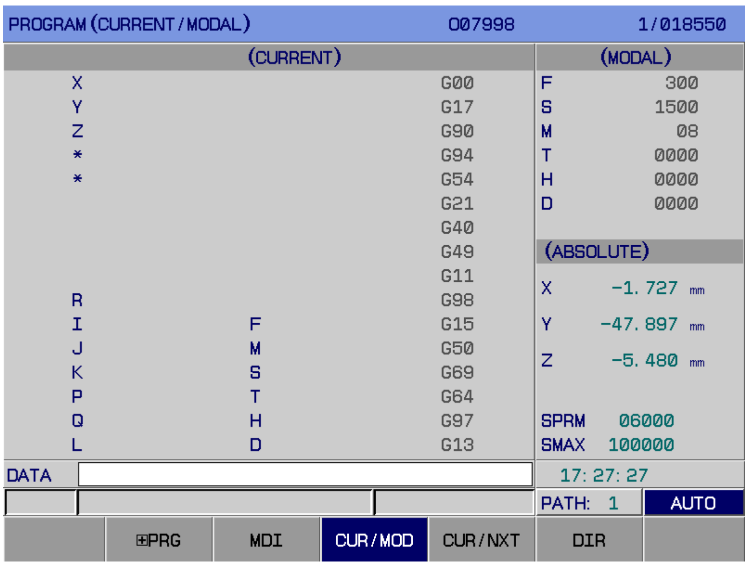


Fig. 8-1-1

8.2 MDI instruction execution and stop

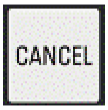
After the instructions are input according to the steps in section 8.1, press key 

to execute them in MDI mode. During the execution, the instruction execution can be stopped by pressing key 

.

Note 1: MDI execution must be performed in MDI mode**.**
Note 2: The program input in【CUR/MOD】page is executed prior to that input in MDI mode.

8.3 Word value modification and deletion of MDI instruction

If a mistake occurs during the input, press key 

to cancel it; if a mistake is detected after the input, re-input the contents to replace the wrong ones or press key 

to delete all the contents and then input them again.

8.4 Operation modes conversion

In Auto, MDI or DNC mode, when the control is converted to MDI, DNC, Auto or Edit mode during the program execution, the system stops the execution of the program after the current block is executed.

When the control is switched to Step mode by a dwell during the program execution in Auto, MDI or DNC mode, the step interruption is executed (See section 5.2 Step interruption) . If the control is switched to MPG mode by a dwell, the MPG interruption is executed (See section 6.2 MPG interruption). If the control is switched to MANUAL mode by a dwell, the manual intervention is executed (See section 4.1.4 Manual interruption).

When the control is directly switched to Step, MPG, MANUAL or Zero Return mode during the program execution in Auto, MDI, DNC mode, the system will execute deceleration and stop.

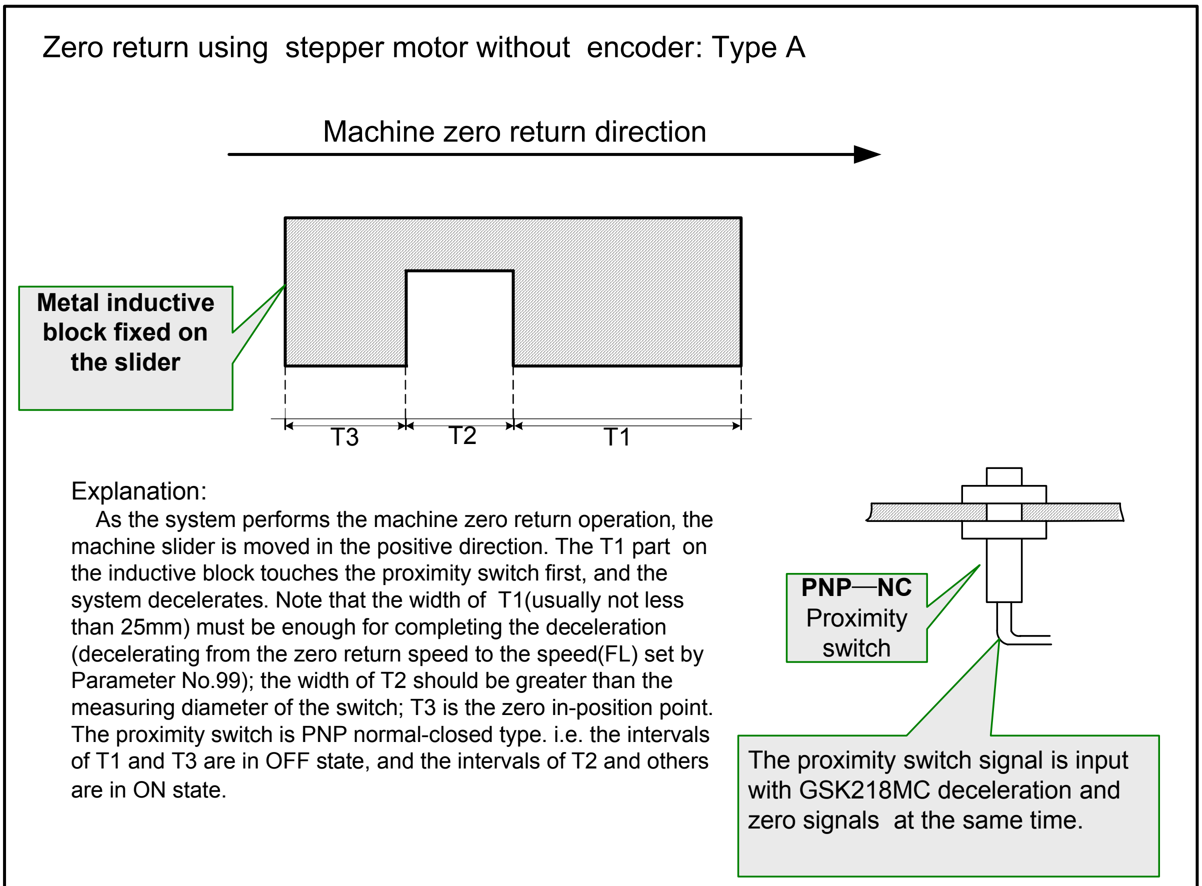
CHAPTER 9 ZERO RETURN OPERATION

9.1 Concept of mechanical zero (machine zero)

The machine coordinate system is the inherent coordinate system of the machine. The origin of the machine coordinate system is called mechanical zero (or machine zero), which is also called **reference point** in this manual. It is usually fixed at the maximum stroke point of X axis, Y axis and Z axis. This origin is determined as a fixed point after the design, manufacture and adjustment of the machine. As the machine zero is unknown at power-on, the auto or manual machine zero return is usually performed.

There are two types of zero return: 1. with one-revolution signal; 2. without one-revolution signal, both of which are set by bit parameter **N0：6#6**.

The zero return without one-revolution signal in the motor is classified into type A and type B zero return, which are set by bit parameter **N0：6#7.**



GSK218MC series machine zero return- Type A

Fig. 9-1-1

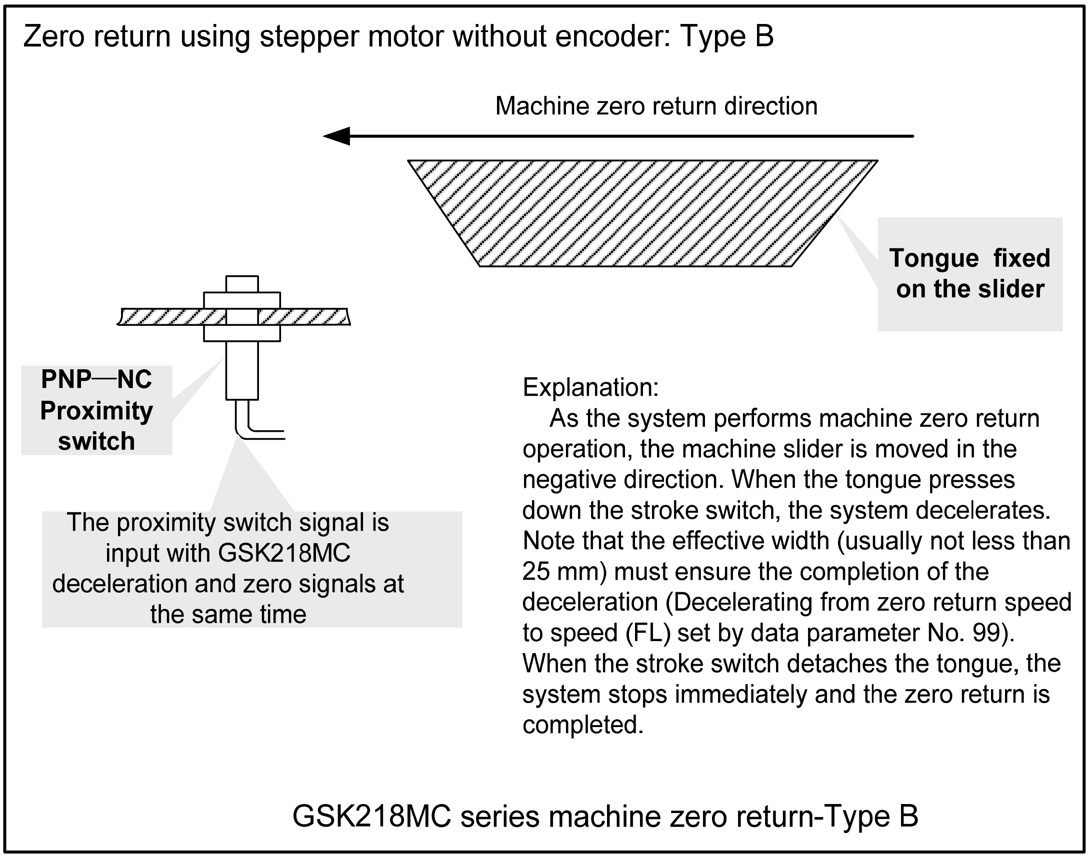
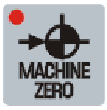


Fig. 9-1-2

9.2 Steps for machine zero return

1. Press 

to enter Machine Zero Return mode, then “machine zero return” will be displayed at the lower right corner of the LCD screen;
2. Select axis X, Y, or Z for machine zero return, the direction of which is set by bit parameter **No.:7#3～N0:7#5**;
3. When it moves towards the machine zero, the machine traverses rapidly (traverse speed set by data parameter **No.100～No.103**) before the deceleration point is reached. After the deceleration switch is touched, it moves to the machine zero point (i.e. reference point) at a speed of FL(set by data parameter **P99**). As the machine zero is reached, the coordinate axis movement stops and the Machine Zero indicator lights up.

9.3 Steps for machine zero return using instructions

The zero return specified by G28 is available after bit parameter **NO：4#3** is set to 0**.** Since it detects the stroke tongue, this instruction is equivalent to manual machine zero.

Note 1: If no machine zero is fixed on your CNC machine, do not perform the machine zero return operation.
Note 2: The indicator of the corresponding axis lights up when the machine zero return is finished**.**
Note 3: The indicator goes out when 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).

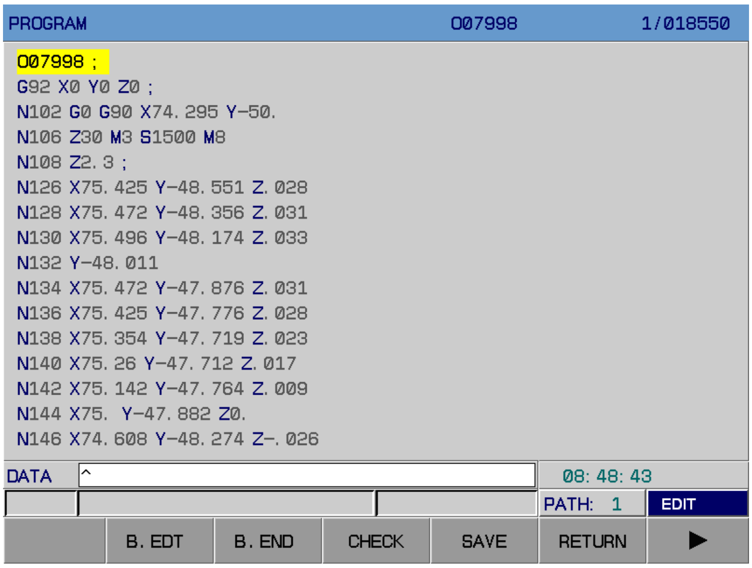
CHAPTER 10 EDIT OPERATION

10.1 Program edit

The edit for part programs should be operated in Edit mode. Press key 

to enter Edit mode; Press key 

to enter program page, and press soft key 【⊞ PROGRAM】 to enter the program editting and modification page (see fig. **10-1-1**).



Press 【▶】 to enter the next page

Press 【▶】 to enter the next page

Press 【◀】 to return to the last page

Fig. 10-1-1

The replacement, cut, copy, paste, reset operations, etc. can be done by pressing the corresponding soft keys.

The program switch must be turned on before program editting. See Section 3.5.2 Parameter and program switch page in this manual for its operation.

Note 1: A program contains no more than 200,000 lines.

Note 2: As is shown in fig. 10-1-1, if there is more than 1 sign “/” ahead of a block, the system will skip the block even if the block skip function is not turned on.

Note 3: It is forbidden to switch the control to other mode when the Check function is performed in Auto mode, or unexpected results will occur.

During Check in Auto mode, if there is a sign "/" ahead of a block, the Check function is performed for this block regardless of whether the skip function is ON.

10.1.1 Program creation

10.1.1.1 Sequence number automatic creation

Set the "AUTO SEQ" to 1 according to the method described in Section **3.5.1.** See fig. 10-1-1-1-1.

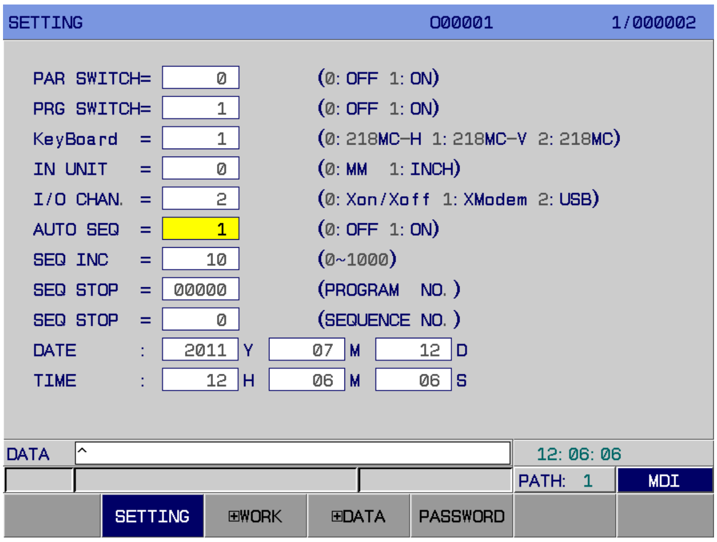


Fig. 10-1-1-1-1

In this way, the sequence number will be automatically inserted into the blocks during program editing. The incremental amount of the sequence number is set by its corresponding parameter.

10.1.1.2 Program input

1. Press key 

to enter Edit mode;

2. Press key 

to enter program page. See fig. **10-1-1-2-1**:

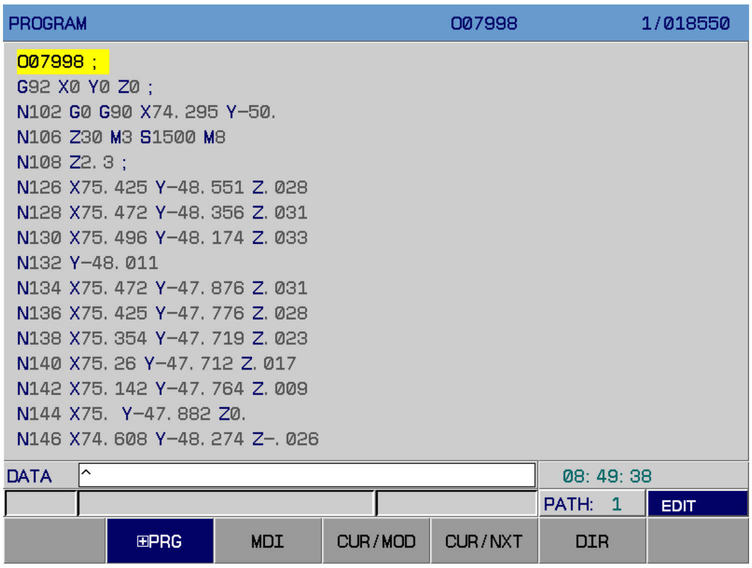


Fig. 10-1-1-2-1

3. Press address key  , and key in numerical keys , , ,  and  in sequence (an example for setting up a program name of O00002 here), then O00002 is displayed behind the DATA column（See Fig. 10-1-1-2-2）:

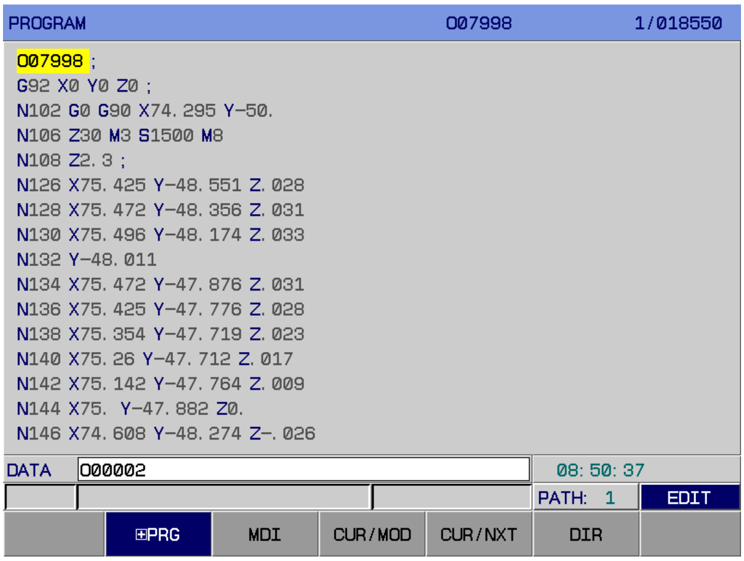


Fig. 10-1-1-2-2

4. Press key  to set up the new program name, as is shown in the fig.10-1-1-2-3:

Book 2　Operation

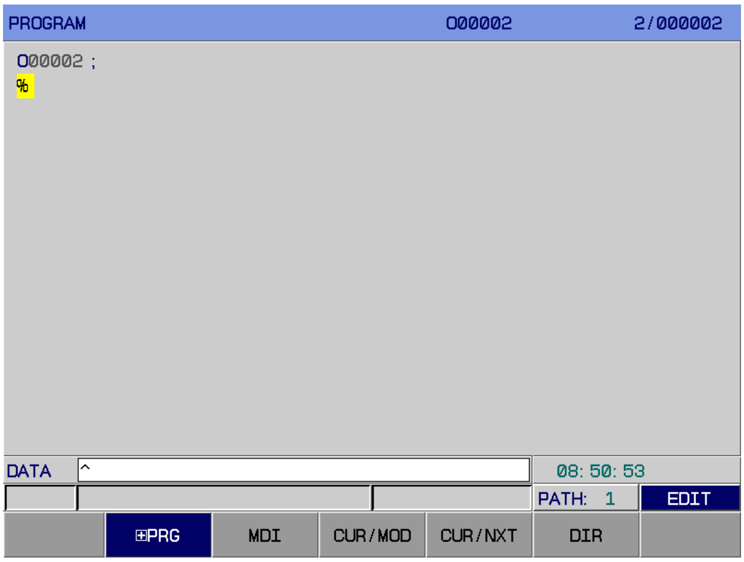


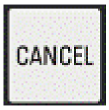
Fig. 10-1-1-2-3

5. Input the written program word by word. After the input, the program will be saved automatically when the control is switched to other operation modes. However, if the control needs to be switched to other pages (e.g. 

page), first press key 

to save the program and then finish the input of the program.

Note 1: Pure numerical value input is unavailable in Edit mode.

Note 2: If a wrong instruction word is detected during program inputting, press key 

to cancel the instruction.

Note 3: No more than 74 characters can be input in one block each time.

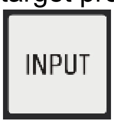
10.1.1.3 Search of sequence number, word and line number

The sequence number search operation is used to search for a sequence number from which the program execution and edit are usually started. Those blocks skipped because of the search have no effect on the CNC state (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).

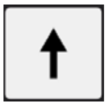
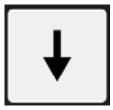
If the execution is started from a block searched in a program, it is required to check the machine and CNC states. The execution can only be performed when both the states are consistent with its corresponding M, S, T codes and coordinate system setting, etc (set in MDI mode).

The word search operation is used to search a specific address word or number , and it is usually used for editing a program.

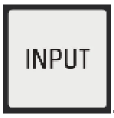
Steps for the search of sequence number, word and line number in a program:

1. Select mode: <Edit > or <Auto>
2. Look up the target program in 【DIR】page;
3. Press key 

to enter the target program;

4. Key in the word or sequence number to be searched and press key  or  to search for it.

5. When needing to search a line number in a program, press key 

, and input the line number to be searched, then press key 

.

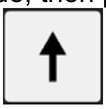
Note 1 The search function is automatically cancelled when the search for sequence number and word is performed to the end of a program.

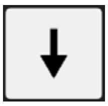
2 The searching for sequence number, word and line number can be performed in either【AUTO】or【EDIT】mode, but in【AUTO】mode, it can only be performed in the background edit page.

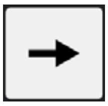
10.1.1.4 Location of the cursor

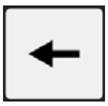
Select Edit mode, then press key 

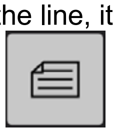
to display the program.

a) Press key  to move the cursor upward a line, if the column where the cursor is located exceeds the end column of the last line, the cursor moves to the end of the last line.

b) Press key  to move the cursor downward a line. If the column where the cursor is located 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 one column to the right. If it is located at the end of the line, the cursor moves to the beginning of the next line.

d) Press key  to move the cursor one 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 screen upward to move the cursor to the last screen.

f) Press key  to move the screen downward to move the cursor to the next screen.

g) Press key 

to move the cursor to the beginning of the line where it is located.

h) Press keys SHIFT + HOME to return the cursor to the beginning of the program.

i) Press key END to move the cursor to the end of the line where it is located.

j) Press keys SHIFT + END to move the cursor to the end of the program.

10.1.1.5 Insertion, deletion and modification of a word

Select <EDIT> mode, press key PROGRAM to display the program, then locate the cursor to the position to be edited.

1. Word insertion

After inputting the data, press key INSERT to insert the data to the left of the cursor.

2. Word deletion

Locate the cursor to the word to be deleted, press key DELETE to delete the word where the cursor is located.

3. Word modification

Move the cursor to the place to be modified, input the new contents, then press key ALTER to replace the old contents by the new ones.

10.1.1.6 Single block deletion

Select <EDIT> mode, then press key PROGRAM to display the program. Locate the cursor to the beginning of the block to be deleted. Press keys N/Q + DELETE to delete the block where the cursor is located.

Note: Regardless of whether there is a sequence number in the block, the user can press key N/Q to delete it (The cursor should be located at the beginning of the line).

10.1.1.7 Deletion of multiple blocks

Blocks deletion from the current displayed word to the block of which the sequence number is specified.

N100 X100.0 M03 S2000; ..... N2233 S02 ; N 2300 M30 ;

Cursor current position

Area to be deleted

Fig. 10-1-1-7-1

Select <EDIT> mode, press key PROGRAM to display the program. Locate the cursor to the beginning of the target position to be deleted (as the position of word N100 in the figure above), then key in the last word of the multiple blocks to be deleted, e.g. **S02** (as Fig.10-1-1-7-1 above), finally press key DELETE to delete the blocks from the current cursor location to the address specified.

Note 1: 200,000 lines of blocks can be deleted at most.
Note 2: If the last word to be deleted occurs many times in a program, the system will delete the blocks till 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) Enter program display page. There are two ways to delete a program:

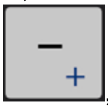
1. Key in address key O P; key in the program name (e.g. for program O0002, key in number Keys 0 *, 0 *, 0 *, 2 #); press key DELETE, the corresponding program in memory will be deleted.

2. Select【DIR】subpage in program page, and select the program name to be deleted by moving the cursor, then press key DELETE. Here, “Delete the current file?” is prompted on the system state column, press key INPUT again, then “Deletion succeeded” is prompted and the program selected is deleted.

Note: If there is only one program file, by pressing key Delete, its name will be changed to O00001 first and then the contents be deleted in Edit (DIR) page regardless of whether it is O00001 or not; if there are multiple program files, the contents of program O00001 as well as its program name are deleted.

10.1.3 Deletion of all programs

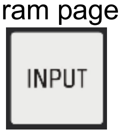
The steps for deleting all programs in memory are as follows:

a) Select <EDIT> mode;
b) Enter the program page;
c) Key in address ;
d) Key in address keys , , , , ,  in sequence;
e) Press key 

to delete all the programs saved in memory.

10.1.4 Copy of a program

Steps for copying the current program and saving it with a new name:

a) Select <EDIT> mode;
b) Enter the program page; select the program to be copied using the cursor in【DIR】subpage, and press key 

to enter the program display page;
c) Press address key , and input the new program name;
d) Press soft key 【COPY】to finish the file copying and enter the edit page for the new program;
e) Return to【DIR】can view the new copied program name.

The copy of a program can also be done in the program edit page (shown in fig. 10-1-1):

1. Press address key  and key in the new program number;
2. Press soft key【COPY】to finish the file copying and enter the edit page for the new program.
3. Return to 【DIR】page to view the new copied program name.

10.1.5 Copy and paste of blocks

Steps for copying and pasting blocks:

a）Locate the cursor to the beginning of the blocks to be copied;

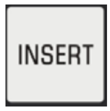
b）Key in the last character of the blocks to be copied;

c）Press keys 

+ 

, the blocks from the cursor to the character keyed in will be copied.

d）Locate the cursor to the position to be pasted, press keys 

+ 

or soft key 【PASTE】 to complete the paste.

The copy and paste of the blocks can also be done in the program edit page (see fig. 10-1-1):

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 soft key【COPY】to finish copying the blocks from the cursor to the character keyed in.
4. Locate the cursor to the position to be pasted, press soft key 【PASTE】to complete the paste.

Note 1: If the last character keyed in occurs many times in the program, the system will copy the blocks till the word nearest to the cursor location.

Note 2: If the blocks are copied with method N+sequence number, the blocks from the cursor to the N + sequence number are copied.

Note 3: 10,000 lines of blocks can be copied at most.

10.1.6 Cut and paste of blocks

Steps for cutting blocks are as follows:

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 soft key【CUT】to cut the block into clipboard.

e) Locate the cursor to the position to be pasted, and press soft key【PASTE】to finish block pasting.

Note 1: If the last character keyed in occurs many times in the program, the system will cut the blocks from the cursor to the word nearest to the cursor.

Note 2: If the blocks are cut with method N+sequence number, the blocks from the cursor to the N sequence number are cut.

10.1.7 Block Replacement

Steps for replacing a block are as follows:

a）Enter the program edit page(Fig.10-1-1);

b）Locate the cursor to the character to be replaced;

c）Key in the new character;

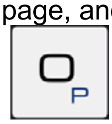
d）Press soft key【REPLACE】to replace the character where the cursor is located as well as other identical characters in the block by the new one.

Note: This replacement operation is only for characters, but not for an entire block.

10.1.8 Rename of a program

Step for renaming the current program to another one:

a) Select <EDIT> mode;
b) Enter the program page, and specify a program name with the cursor;

c) Press address key 

to key in the new name;

d) Press key 

to complete the renaming.

10.1.9 Program restart

The function is used in the event of an accident such as tool fracture, system restarting after power-off or emergency stop during program execution. After the accident is eliminated, the system returns to the program breakpoint by program restart to continue the program execution, and then it retracts to original point by Dry Run.

Steps for program restart are as follows:

1. Solve the machine accident such as tool change, offset changing, machine zero return.

2. In <AUTO> mode, press key 

on the panel.

3. Press key 

to enter the program page, then press soft key【RSTR】to enter program restart subpage (Fig.10-1-9-1)

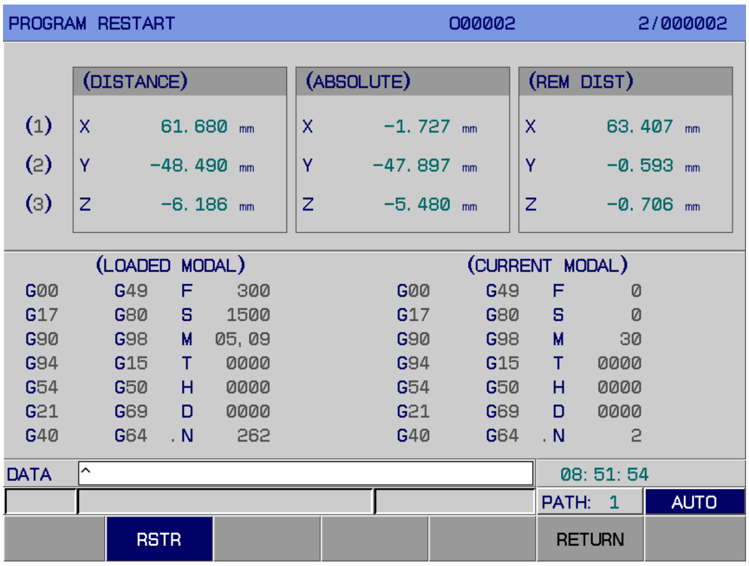


Fig. 10-1-9-1

4. In 【CUR/MOD】page, input corresponding modes according to the pre-loaded modal values in Fig.10-1-9-1.

5. Return to <AUTO> mode, press key 

, and then key 

on the panel. Then the

program moves to the start point (i.e. the end point of the last block) of the interrupted block at the dry run speed and the execution continues . The operation can be restarted anywhere.

Note 1: The “(1), (2), (3)” ahead of the coordinates in the figure above are the sequence in which the axes moves to the program restart position. They are set by data parameter P376.
Note 2: Check whether the collision occurs when the tool moves to the program restart position. If such a possibility exists, move the tool to the place where no obstruction occurs and then perform restart.
Note 3: When the coordinate axis restarts the position moving to 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, the reference point return must be performed before the restart after power-on.
Note 5: Do not perform the resetting during the program execution from block research at restarting to restarting, or the restarting must be done from the first step.
Note 6: The restart function of the system does not support the program containing subprograms currently.

10.2 Program management

10.2.1 Program directory search

Press key 

, then press soft key【DIR】to enter the program directory page（See Fig.10-2-1-1）:

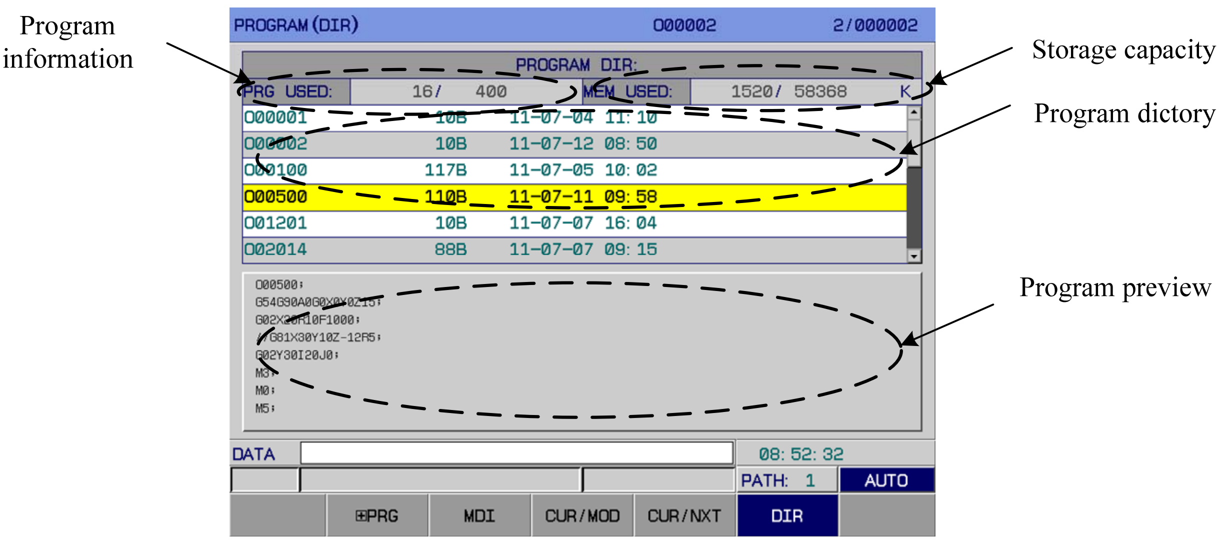


Fig. 10-2-1-1

1) Open a program
Open a specified program: O+sequence number+ key ENTER (or key EOB), or sequence number + key ENTER (or key EOB)
In Edit mode, if the sequence number input does not exist, a new program will be created.

2) Deletion of a program:
1. In Edit mode, press key DEL to delete the program where cursor is located.

2. In Edit mode, press O+ sequence number + DEL, or sequence number + DEL

10.2.2 Number of stored programs

Not more than 400 programs can be stored in this system. The number of the stored programs can be viewed in the program directory page (program information) in fig. **10.2.1.**

10.2.3 Storage capacity

The storage capacity can be viewed in the program directory page (storage capacity) in fig. **10.2.1.**

10.2.4 Viewing of program list

One program directory page can display 6 CNC program names at most. If there are more than 6 names, it is unavailable to display them all in one page. Here, you can press the PAGE key to display the remaining names on the next page. If the Page key is pressed repeatedly, all the CNC program names will be displayed circularly on LCD.

10.2.5 Program lock

The program switch is provided in this system to prevent the user programs from being modified by unauthorized personnel. After the program editing, turn off the program switch to lock the program, thus disabling the program edit. See Section 3.4.1 for details.

CHAPTER 11 SYSTEM COMMUNICATION

This system can communicate with PC or USB via its own interfaces to realize data transmission and DNC on-line machining.

11.1 Serial communication

Preparation for serial communication

1. Connect the PC serial port and system RS232 interface using a serial line.
2. Open GSK Com serial communication software on PC side.

Note: GSK Com serial communication software uses Windows-like interfaces. It can run in Win98, WinMe, WinXP and Win2000.

3. Setting for GSK Com serial communication software:
   (1) Select “Suitable for GSK218MC”;
   (2) Click “Series Port” menu, and set baudrate in “Serial Setting” dialog. For data transmission, select the baudrate of 115200 (corresponding to the default set by data parameter P002); For DNC on-line machining, select the baudrate of 38400 (corresponding to the default set by data parameter P001)

11.1.1 Program start

Run program Comm218MC.exe directly. The page is as follows:

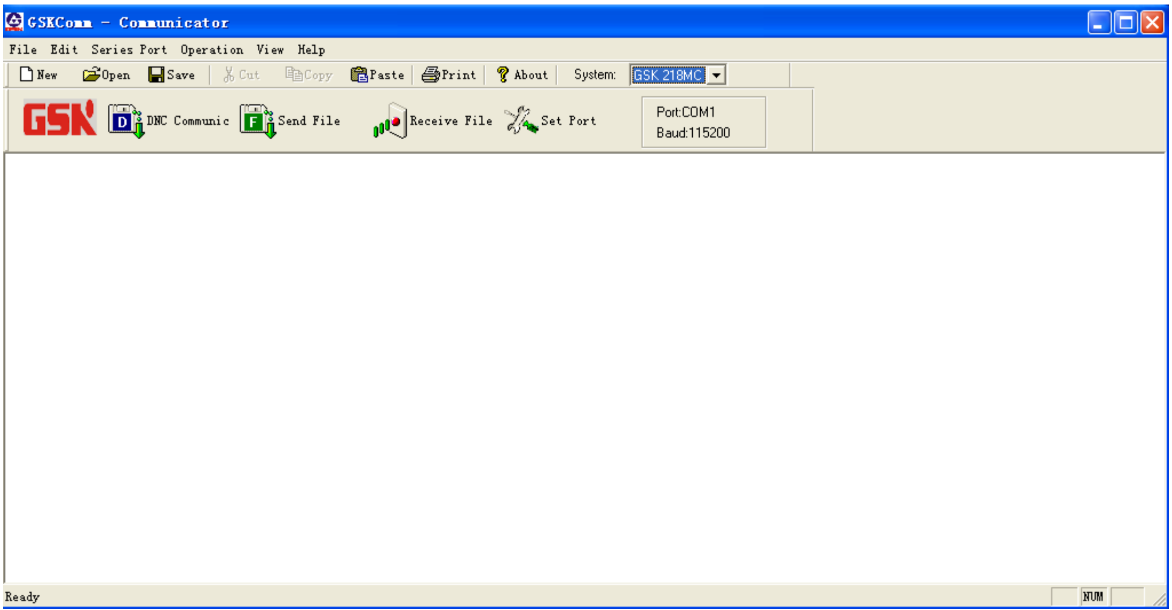


Fig. 11-1-1-1

11.1.2 Functions

1. File menu

The file menu involves functions of New, Open, Save, Print and Print setting and the latest file list etc.

2. Edit menu

The edit menu involves functions such as Cut, Copy, Paste, Undo, Find and Replace.

3. Serial port menu

It is mainly used for opening and setting the serial port.

4.Transfer/Operation menu

It consists of three transmission types: DNC, file sending and file receiving.

5. View menu

It is used for hiding and displaying the tool bar and status bar.

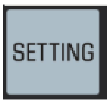
6. Help menu

It is used to view the software version.

11.1.3 Serial port data transmission

Steps are shown as follows:

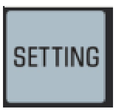
1) Select <MDI> mode;

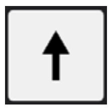
2) Press key 

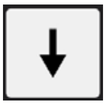
to enter setting page, set the I/O channel to 0 or 1. (With I/O channel set to 0, select Xon/Xoff for DNC protocol; with I/O channel set to 0, select XModem for DNC protocol)

3) Press soft key 【PSW】to enter Setting (Password) page, and then input corresponding password authority.

Target file	Password authority
Ladder（PLC）	Password for machine tool builder level, password for system manufacturer level
Parameter（PLC）	Password for machine tool builder level, password for system manufacturer level
System parameter value	Password for system debugging level, password for machine tool builder level, password for system manufacturer level
Tool offset value	Without a password
Pitch offset value	Password for system debugging level, password for machine tool builder level, password for system manufacturer level
System macro variable	Without a password
Custom macro program	Password for system debugging level, password for machine tool builder level, password for system manufacturer level
CNC part program	Without a password

4) Press key 

to enter SETTING (DATA DEAL) page, then press key  or

 to move the cursor to the target position.

A. Data output （CNC→PC）

1. Press system soft key 【OUTPUT】, then the system prompts “transfer waiting”

2. Click button 

on GSK Com serial communication software, then “Receive File” dialog pops up, as is shown in fig. 11-1-3-1.

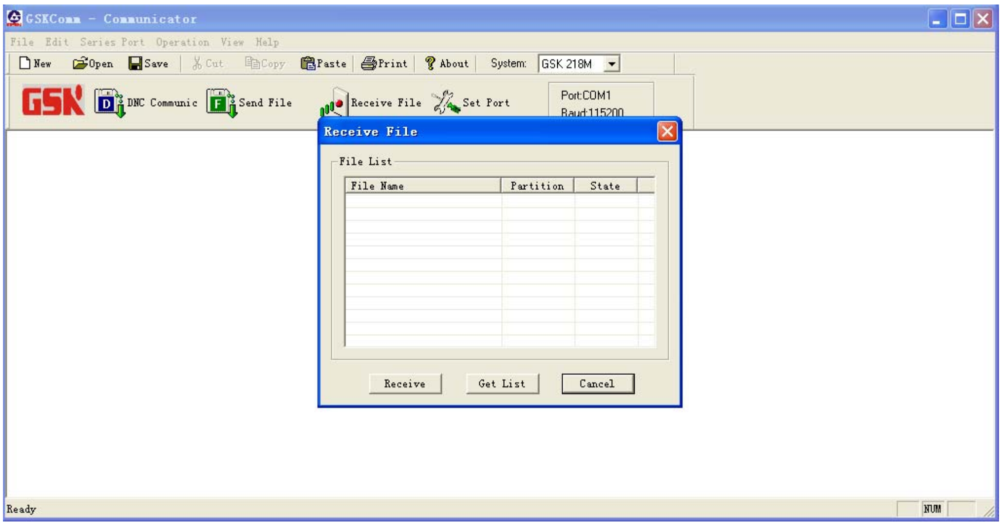


Fig. 11-1-3-1

3. Click button 

in Receive File dialog to obtain the CNC file list, as is shown in fig. 11-1-3-2:

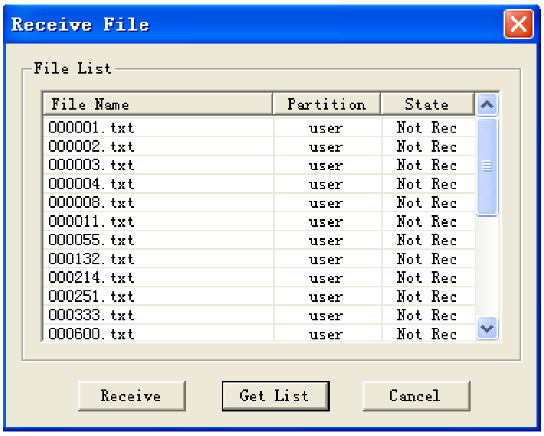


Fig. 11-1-3-2

4. Select the file (or multiple files) to be received, then press button 

to start the file receiving, as is shown is fig. 11-1-3-3:

Book 2 Operation

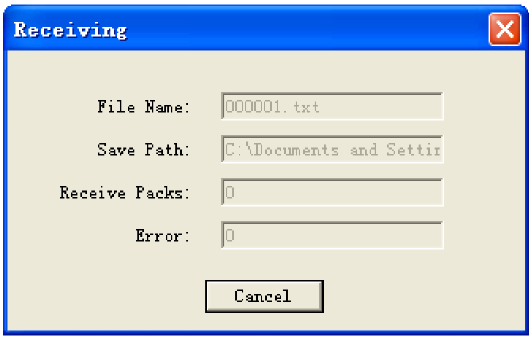


Fig. 11-1-3-3

5. After the file receiving, the status bar of the dialog displays “Received”, as is shown in fig. 11-1-3-4

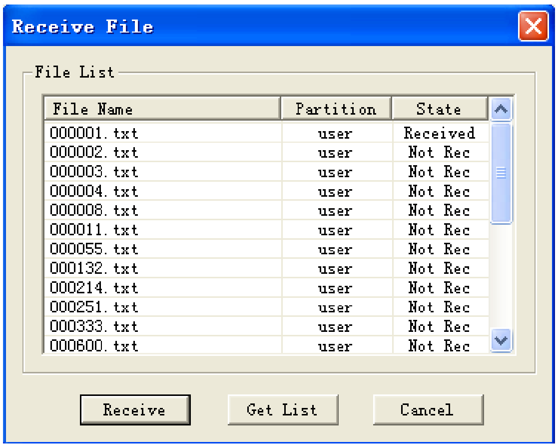


Fig. 11-1-3-4

B. Data input（PC→CNC）

1. Press system soft key 【IINPUT】, then the system prompts “input waiting”

2. Click button 

(or press “Send File” in the down menu of “OPERATION”) to pop up Send File Dialog in the GSK com serial communication software, as is shown in fig. 11-1-3-5.

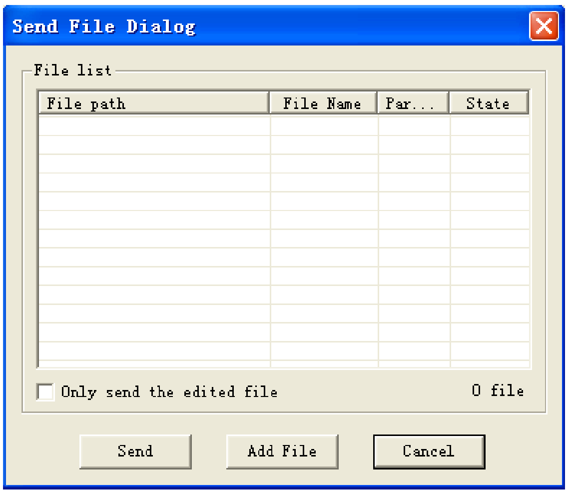


Fig. 11-1-3-5

3. Click button 

in the “Send File” dialog, then the “Select Part Dialog” pops up as in fig. 11-1-3-6.

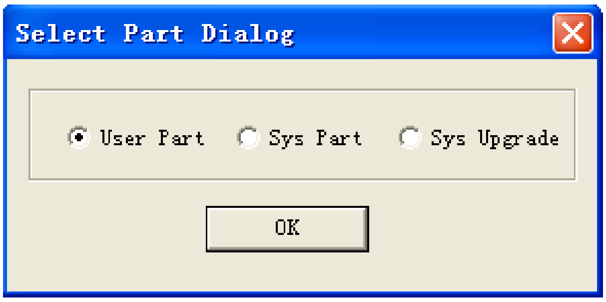


Fig. 11-1-3-6

4. In the "Select Part Dialog":

   Select "User Part" when sending CNC part programs and custom macro programs; select "System Part" when sending files such ladder (PLC), parameters (PLC), system parameter values, tool offset values, pitch offset values and system macro variables.

5. After selecting the partition, select the file (or multiple files) to be sent, and click button 

to start the file sending, as is shown in fig. 11-1-3-7.

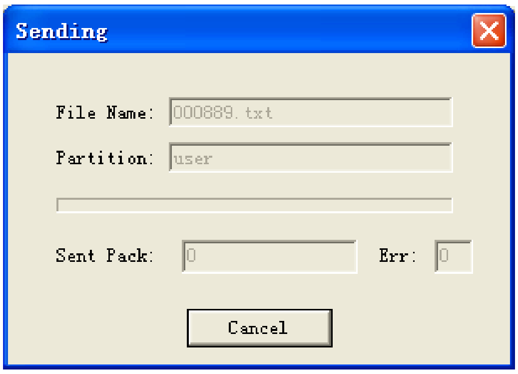


Fig. 11-1-3-7

6. After sending the file/files, "Sent" is displayed in the dialog, as is shown in fig. 11-1-3-8.

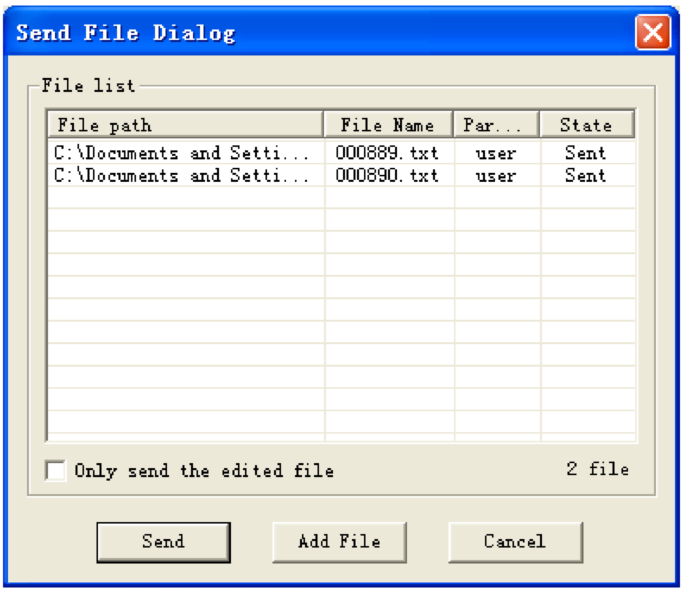


Fig. 11-1-3-8

Note 1: Make sure the baudrate is correctly set and the serial line is reliably connected before data transmission,

Note 2: It is forbidden to switch operation modes or pages during data transmission, or critical errors will occur.

Note 3: File LADCHI**.TXT is ineffective when transferred to the system unless the power is turned off.

11.1.4 Serial port on-line machining

Operation steps

1. Setting for CNC side:

1) Press key 

to enter setting page, and set I/0 channel to 0 or 1.

2) Select <DNC> mode; then the system prompts “DNC state ready, press key INPUT after sent by PC”

2. Setting for serial communication software

1) Click menu “Series Port”, set the baudrate to 38400 in Serial Port Setting Dialog.

2) When the system I/O channel is set to 0, select Xon/Xoff in the pull-down menu “DNC Protocol” of Menu “Operation”.

When the system I/O channel is set to 1, select XModem in the pull-down menu “DNC Protocol” of Menu “Operation”

3. Open CNC program files. Open the program files by pressing button “Open” in menu “File” or button 

in the toolbar, as is shown in fig.11-1-4-1 below (further edit for the program files by serial communication software)

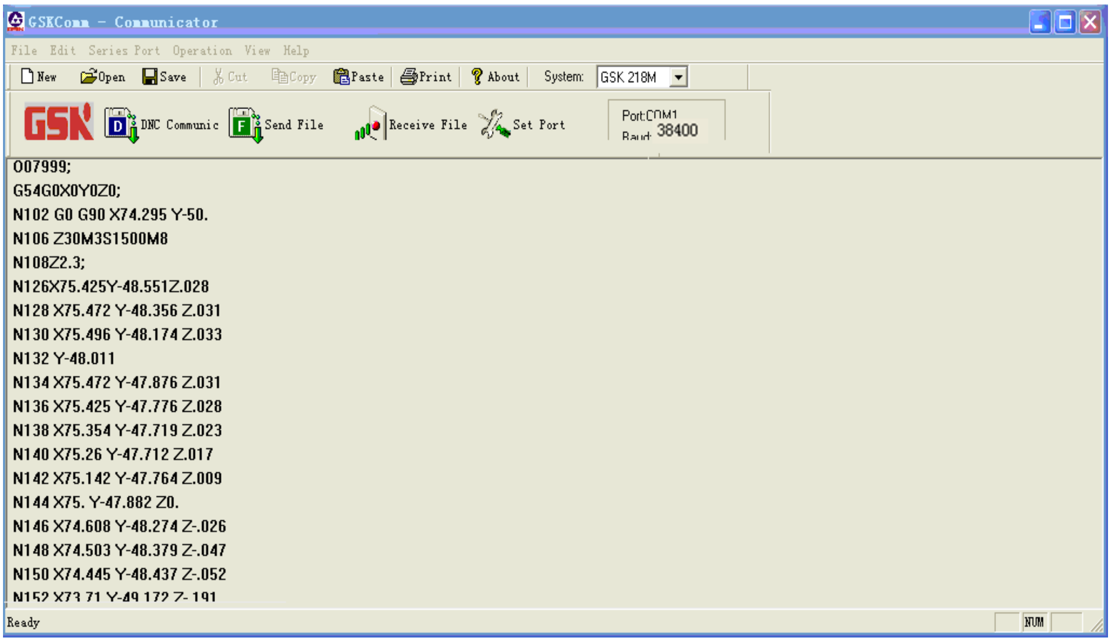


Fig. 11-1-4-1

4. DNC transmission. Click 

in the toolbar or pull-down menu “DNC Communic” in menu “Operation” to send the data. When the system I/O channel is set to 0, PC sends the files directly in a common way, then “DNC COMMUNICATION dialog displays the states of file sending, including the file name, sent bytes, sent lines as well as sent time and speed (byte/s), as is shown in fig. 11-1-4-2. When the system I/O channel is set to 1, PC sends the files by pack, and the dialog displays the states such as sent pack and retransmission times, as is shown in fig. 11-1-4-3:

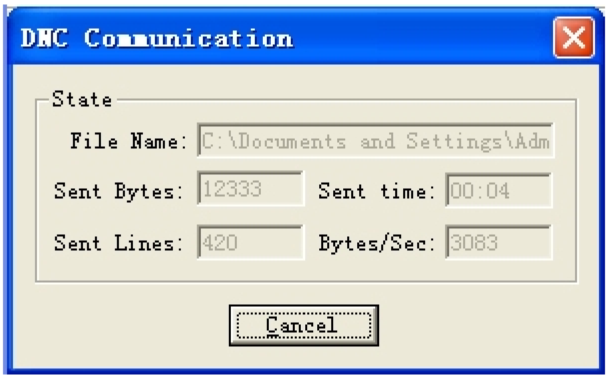


Fig. 11-1-4-2 System I/O channel set to 0

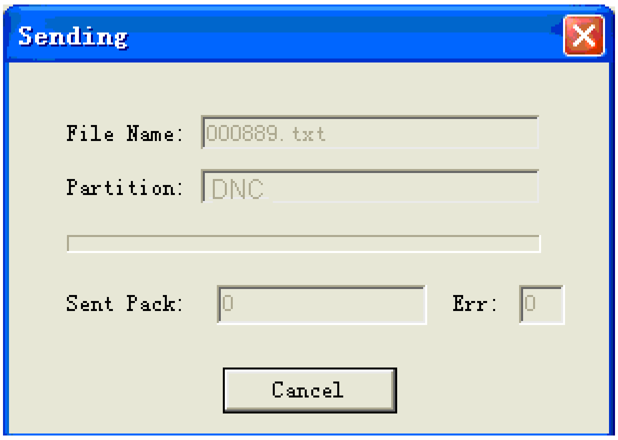
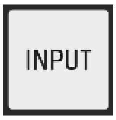


Fig. 11-1-4-3 System I/O channel set to 1

5. Press key 

on the CNC panel to receive data, and then press button 

on the panel to start the machining.

Note 1: Do not operate the serial communication software during DNC transmission except for ending the transmission.

Note 2: M99 is processed as M30 in DNC mode.

Note 3: Press key 

to cancel the operation after the machining is completed.

11.2 USB communication

11.2.1 Overview and precautions

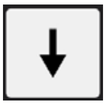
Precautions:

1. Set I/O channel to 2 in <SETTING> page.
2. The CNC programs should be stored in the root directory of the U disk with file extension .txt, .nc

or .CNC, or they cannot be read by the system.

3. After the USB communication is finished, pull out the U disk when its indicator does not flicker (or after a moment is waited for) to ensure the completion of the data transmission.

11.2.2 Operations steps for USB part programs

In <MDI> mode, enter the SETTING (DATA DEAL) page, press direction key  or  to move the cursor to “PART PRGR”. Press soft key 【OUTPUT】or【INPUT】to enter the page shown as follows (fig. 11-2-2-1):

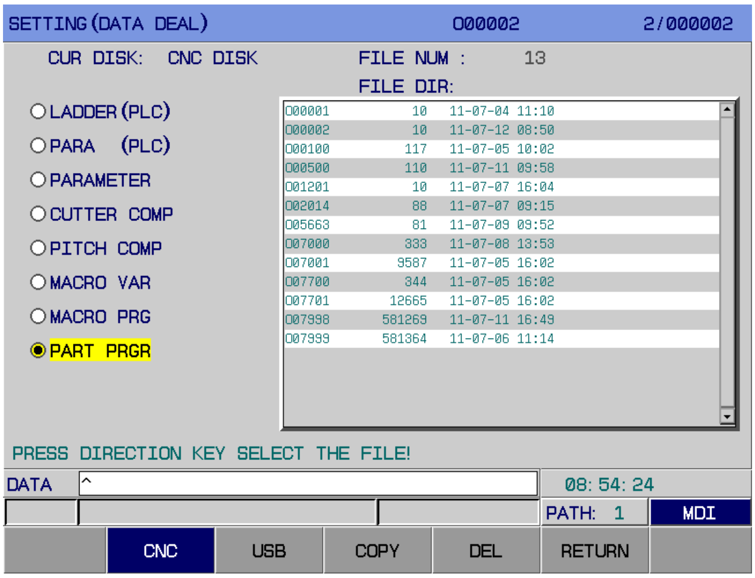
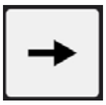
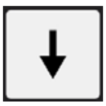


Fig. 11-2-2-1

1. To copy CNC program files to U disk from the system disk:

a. Press key  to switch the cursor to the file directory.

b. Press key  or  to move the cursor to select the CNC program files to be copied in the system disk.

c. Press soft key 【COPY】, then the systems prompts “COPY TO USB DISC? New Name”, as is shown in fig. 11-2-2-2.

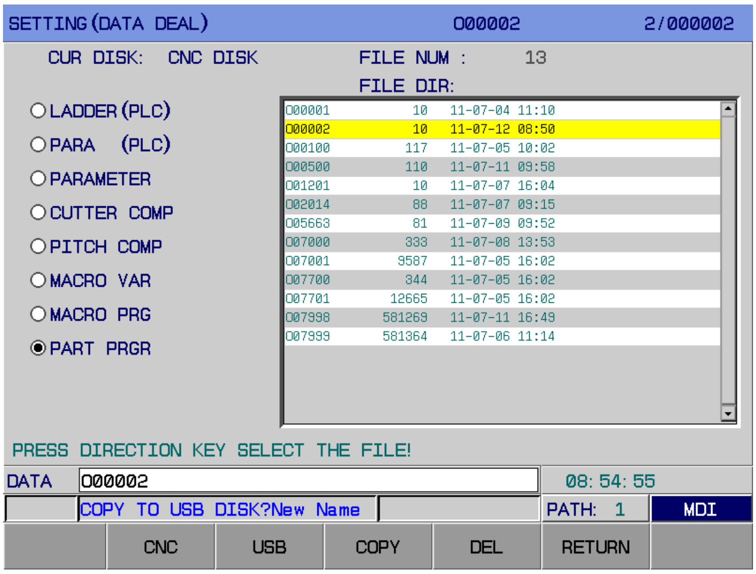


Fig. 11-2-2-2

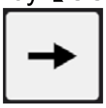
d. If renaming for CNC program files is not required, press key <INPUT> to copy the CNC program files directly.

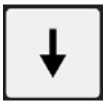
Renaming required, press key <CANCEL> to input the new program number (e.g. O10 or O100), and then press key <INPUT> to copy the program files.

If the program name already exists in the U disk, the system prompts “Please rename the file” . Here, input the new program number (e.g. O10 or O100) and then press key <INPUT> to copy the CNC program files.

2. To copy CNC program files to system disk from U disk:

a. Press soft key【USB】to switch to USB file directory page;

b. Press key  to switch the cursor to the file directory.

c. Press key  or  to move the cursor to select the CNC program files to be copied in the U disk.

Press soft key 【COPY】, then the system prompts “COPY TO CNC DISC? New Name”, which is shown in fig. 11-2-2-3:

Book 2 Operation

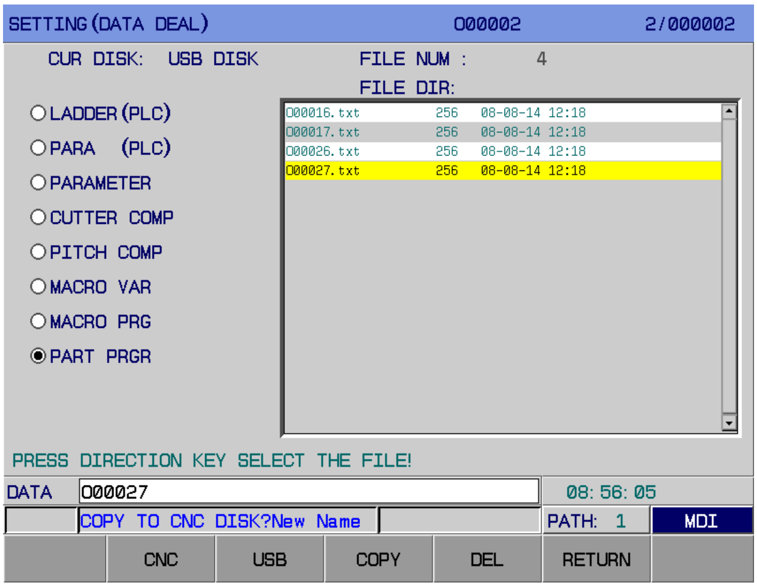


Fig. 11-2-2-3

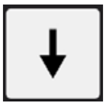
d. If renaming for CNC program files is not required, press key <INPUT> to copy the CNC program files directly.

Renaming required, press key <CANCEL> to input the new program number (e.g. O10 or O100), and then press key <INPUT> to copy the program files.

If the same program name already exists in the system disk, the system prompts “Please rename the file” . Here, input the new program number (e.g. O10 or O100) and then press key <INPUT> to copy the CNC program files.

Note: File LADCHI**.TXT is ineffective after transmitted to the system unless the power is turned off.

3. To delete files from system disk/U disk

a. Press key  or  to move the cursor to select the CNC program files to be deleted in the system disk/U disk.

b. Press soft key【DEL】, then “DELETE CURRENT FILE?” is prompted at the bottom of the page. Press key <CANCEL> to cancel the file deletion; press key <ENTER> to delete the file.

11.2.3 USB DNC machining operation steps

1. In <SETTING> page, set I/O channel to 2. See Section 3.4.1 in OPERATION for details.
2. Insert the U disk.
3. Press key <DNC> to switch the system to DNC mode, then “Please select machining files in program directory page” is prompted at the bottom of the screen. Press key 【◆PRG】 to enter program page, and press soft key 【DIR】 to display the USB program directory; move the cursor to select the program to be machined, and then press <INPUT> to open the program, finally press key <CYCLE START> to execute DNC machining.

Note: In USB program directory page, if the name of a program contains 6 characters or less, the beginning of the program can be previewed; if the name contains more than 6 characters, the beginning cannot be previewed; if the name contains 8 characters or more, the system displays an abbreviation for it, and the beginning cannot be previewed neither.

11.2.4 Exiting U disk page

1. Pull out the U disk as its indicator does not blink.
2. Press soft key 【RETURN】to return to【DATA】subpage in <SETTING> (DATA DEAL) page.

APPENDIX

APPENDIX I GSK218MC SERIES PARAMETER LIST

Explanation:

The parameters are classified into following patterns according to the data type:

- 2 data types and data value range

Data type	Effective data range	Remark
Bit	0 or 1	The default value is given by the CNC, and user can modify the setting by requirement.
Data	Specified according to the parameter range	The default value is given by the CNC, and user can modify the setting by requirement.

1. For bit and axis parameters, the data are comprised by 8 bits with each bit having different meaning.
2. The data value range in above table is common effective range. The specific parameter value range actually differs. See the parameter explanation for details.

Example

（1）Meaning of the bit parameters

Data number	**BIT7**	**BIT6**	**BIT5**	**BIT4**	**BIT3**	**BIT2**	**BIT1**	**BIT0**

（（2）Meaning of the data parameters

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 to 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 to 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 to 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		MODE		SEQ	MSP	CPB	INI		PBUS

PBUS =1: Transmission type of the drive unit is bus type
=0: Transmission type of the drive unit is pulse type

INI =1: Inch input
=0: Metric input

If **INI** is set to 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 to 1, in inch input, the basic unit for linear axis is inch、inch/min；that for rotary axis is deg, deg/min.

CPB =1: Pulse port and Ethernet are used simultaneously.
=0: Pulse port and Ethernet are not used simultaneously.

MSP =1: Double-spindle control is used.
=0: Double-spindle control is not used.

SEQ =1: Automatic sequence number insertion
=0: Not automatic sequence number insertion

MODE =1: High-speed and high-precision mode. #15.0 and #17.0 can not be modified, and only X, Y, Z axes can be used.
=0: Common mode. When the high speed and high precision mode is changed into common mode, default setting for #15.0 is 1.

Standard setting：0 1 0 0 0 0 0 0

System parameter number

0	0	1	SJZ	TMES				SPT	UDVP	DRVT

DRVT =1: Bus drive unit is DAH01 series
=0: Bus drive unit is DA98E series

UDVP =1: Automatically update parameter of the drive unit
=0: Not automatically update parameter of the drive unit

SPT =1: I/O point control
=0: Frequency conversion or others

TMES =1: Toolsetting gauge is fixed
=0: Toolsetting gauge is not fixed

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	SIOD	SK0	STME		DEC4	DECZ	DECY	DECX

DECX =1: X-axis deceleration signal is high level active
=0: X-axis deceleration signal is low level active

DECY =1: Y-axis deceleration signal is high level active
=0: Y-axis deceleration signal is low level active

Appendix

DECZ	=1:	Z-axis deceleration signal is high level active
	=0:	Z-axis deceleration signal is low level active
DEC4	=1:	The 4TH axis deceleration signal is high level active
	=0:	The 4TH axis deceleration signal is low level active
STME	=1:	Tool length value can be added to reference offset
	=0:	Tool length value can not be added to reference offset
SK0	=1:	Skip signal SKIP is input when it is 0
	=0:	Skip signal SKIP is input when it is 1
SIOD	=1:	Machine zero return deceleration signal with PLC logical operation
	=0:	Machine zero return deceleration signal without PLC logical operation

Standard setting：0 0 0 0 0 0 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 to 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 to 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:	The 4^{th} axis feeding direction
	=0:	The 4^{th} axis feeding direction reversing
DIR5	=1:	The 5^{th} axis feeding direction
	=0:	The 5^{th} axis feeding direction reversing

Standard setting：0 0 0 1 1 1 0 0

System parameter number

0	0	4						AZR			JAX

JAX	=1:	Synchronous controlled axes for manual reference point mode: one axis
	=0:	Synchronous controlled axes for manual reference point mode: multiple axes
AZR	=1:	For G28 when reference point is not setup: alarm
	=0:	For G28 when reference point is not setup: use tongue

Standard setting：0 0 0 0 0 0 0 0

System parameter number

0	0	5		IPR						ISC	

ISC	=1:	Min. moving unit of 0.0001mm/deg,0.00001inc
	=0:	Min. moving unit of 0.001mm/deg,0.0001inc

Appendix

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**					**ZMOD**	**ZRN**

ZRN =1: When the reference point is not specified, system alarms if instruction other than G28 is specified during auto running
=0: When the reference point is not specified, system doesn't alarm if instruction other than G28 is specified during auto running.

ZMOD =1: Reference return mode selection: in front of the tongue
=0: Reference return mode selection: behind the tongue

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		**A4TP**	**ZMI4**	**ZMIZ**	**ZMIY**	**ZMIX**		**A4RT**	

A4RT =1: Axis rotates with nearst principle
=0: Axis does not rotate with nearest principle

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 the 4th axis reference point return: negative
=0: Direction setting of the 4th axis reference point return: positive

A4TP =1: It is a four-axis coordinate system
=0: It is not a four-axis coordinate system

Standard setting：0 0 0 0 0 0 0 0

System parameter number

0	0	8		**AXS4**	**AXSZ**	**AXSY**	**AXSX**	**PLW4**	**PLWZ**	**PLWY**	**PLWX**

PLWX =1: Pulse width of X-axis is set to 2 microseconds
=0: Pulse width of X-axis is set to 1 microsecond

PLWY =1: Pulse width of Y-axis is set to 2 microseconds
=0: Pulse width of Y-axis is set to 1 microsecond

PLWZ =1: Pulse width of Z-axis is set to 2 microseconds
=0: Pulse width of Z-axis is set to 1 microsecond

PLW4 =1: Pulse width of the 4th axis is set to 2 microseconds
=0: Pulse width of the 4th axis is set to 1 microsecond

Appendix

AXSX =1: X-axis is set as linear axis
=0: X-axis is set as rotation axis

AXSY =1: Y-axis is set as linear axis
=0: Y-axis is set as rotation axis

AXSZ =1: Z-axis is set as linear axis
=0: Z-axis is set as rotation axis

AXS4 =1: The 4th axis is set as linear axis
=0: The 4th axis is set as rotation axis

System parameter number

0	0	9		APZA	APZZ	APZY	APZX	UHSM	APC	

APC =1: Use absolute encoder
=0: Not use absolute encoder

UHSM =1: Machine zero point can be set manually
=0: Machine zero point can not be set manually

APZX =1: Position of the X-axis machine tool are consistent with that of the absolute encoder
=0: Position of the X-axis machine tool are not consistent with that of the absolute encoder

APZY =1: Position of the Y-axis machine tool are consistent with that of the absolute encoder
=0: Position of the Y-axis machine tool are not consistent with that of the absolute encoder

APZZ =1: Position of the Z-axis machine tool are consistent with that of the absolute encoder
=0: Position of the Z-axis machine tool are not consistent with that of the absolute encoder

APZA =1: Position of the 4th axis machine tool are consistent with that of the absolute encoder
=0: Position of the 4th axis machine tool are not consistent with that of the absolute encoder

Standard setting：0 0 0 0 0 1 0 0

Syatem parameter number

0	1	0	RCUR	MSL			RLC	ZCL	SCBM	

SCBM =1: Check the stroke before moving
=0: Not check the stroke before moving

ZCL =1: To cancel local coordinate system when performing manual reference point return
=0: Not cancel relative coordinate system when performing manual reference point return

RLC =1: To cancel relative coordinate system after resetting
=0: Not cancel relative coordinate system after resetting

MSL =1：Start from the line where cursor locates on cycle start of multi-section MDI
=0: Start from the first line on cycle start of multi-section MDI

Appendix

RCUR =1： Cursor returns to the starting position in non-edit mode after reset
=0: Cursor not returns to the starting position in non-edit mode after reset

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 the 2nd stroke is unallowed
=0: Inner area entry of the 2nd stroke is unallowed

LZR =1: To perform travel check before manual reference return after power-on
=0: Not perform travel check before manual reference 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 (system alarm range is 5MM in front of borders of forbidding area)

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 non-linear

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	

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				PIIS		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

PIIS =1: Overlapping interpolation effective in acceleration/deceleration blocks before forecasting
=0: Overlapping interpolation ineffective in acceleration/deceleration blocks before forecasting

Standard setting：0 0 0 0 0 0 0 1

System parameter setting

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 setting

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: lineat

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

Appendix

	=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							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

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	ALM5	ALM4	ALMZ	ALMY	ALMX

ALMX	=1:	High level effective of X-axis driver alarm
	=0:	Low level effective of X-axis driver alarm
ALMY	=1:	High level effective of Y-axis driver alarm
	=0:	Low level effective of Y-axis driver alarm
ALMZ	=1:	High level effective of Z-axis driver alarm
	=0:	Low level effective of Z-axis driver alarm
ALM4	=1:	High level effective of the 4^{th} axis driver alarm
	=0:	Low level effective of the 4^{th} axis driver alarm
ALM5	=1:	High level effective of the 5^{th} axis driver alarm
	=0:	Low level effective of the 5^{th} axis driver alarm
ALMS	=1:	High level effective of spindle driver alarm
	=0:	Low level effective of spindle 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									ITL

ITL	=1:	All axes interlock signal effective
	=0:	All axes interlock signal ineffective

Standard setting：0 0 0 0　0 0 0 0

Appendix

System parameter number

0	2	1									DCHC

DCHC =1: Control function of two paths effective
=0: Control function of two paths ineffective

Standard setting：0 0 0 0 0 0 0 0

System parameter number

0	2	2			DAL						

DAL =1: Add tool length compensation in absolute position display
=0: Not add tool length compensation in absolute position display

Standard setting：0 0 0 0 0 0 0

System parameter number

0	2	3			POSM						

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			NPA						

NPA =1: To switch to alarm page when alarm occurs
=0: Not switch to alarm page when alarm occurs

Standard setting：0 0 0 0 0 0 0 0

System parameter number

0	2	5		ALM	DGN	GRA	SET		SYS	PRG	POS

POS =1: To switch over page by pressing POSITION key in position page
=0: Not switch over page by pressing POSITION key in position page
PRG =1: To switch over page by repressing POSITION key in program page
=0: Not switch over page by repressing POSITION key in program page
SYS =1: To switch over page by repressing PARAMETER key in program page
=0: Not switch over page by repressing PARAMETER key in program page
SET =1: To switch over page by repressing SET key in set page
=0: Not switch over page by repressing SET key in set page
GRA =1: To switch over page by repressing GRAPHIC key in graphic page
=0: Not switch over page by repressing GRAPHIC key in graphic page
DGN =1: To switch over page by repressing DIAGNOSE key in diagnosis page
=0: Not switchover page by repressing DIAGNOSE key in diagnosis page
ALM =1: To switch over page by repressing ALARM key in alarm page
=0: Not switch over page by repressing ALARM key in alarm page

Standard setting：1 1 1 1 1 1 1 1

System parameter number

0	2	6		HELP	PLC			SMDT	SMDI	SPET	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
SPET =1: Turn to program page automatically by pressing PROGRAM in edit mode
=0: Not turn to program page automatically by pressing PROGRAM in edit mode
SMDI =1: Turn to MDI page automatically by pressing PROGRAM in MDI mode
=0: Not turn to MDI page automatically by pressing PROGRAM in MDI mode
SMDT =1: Turn to current/ mode page selection automatically by pressing PROGRAM in MDI mode
=0: Turn to MDI page selection automatically by pressing PROGRAM in MDI mode
PLC =1: To switch over page by repressing PLC key in PLC page
=0: Not switch over page by repressing PLC key in PLC page
HELP =1: To switch over page by repressing HELP key in help page
=0: Not switch over page by repressing HELP key in help page

Standard setting：1 1 0 0 0 0 0 1

System parameter number

0	2	7					NE9				NE8

NE8 =1: Editting of subprogram with 80000 – 89999 unallowed
=0: Editting of subprogram with 80000 – 89999 allowed
NE9 =1: Editting of subprogram with 90000 - 99999 unallowed
=0: Editting of subprogram with 90000 - 99999 allowed

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					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

Standard setting：0 0 0 0 0 0 0 0

System parameter number

0	3	1				G13	G91		G19	G18	G01

G01	=1:	G01 mode at power-on or clearing
	=0:	G00 mode at power-on or clearing
G18	=1:	G18 plane at power-on or clearing
	=0:	Not G01 at power-on or clearing
G19	=1:	It depends on parameter No31#1
	=0:	When G19=1, please set G18 to 0

G19	G18	G17, G18, G19 mode
0	0	G17 mode（X-Y plane）
0	1	G18 mode（Z-X plane）
1	0	G19 mode（Y-Z plane）

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

Standard setting：0 0 1 0 0 0 1 0

Syatem parameter number

0	3	2			AD2						

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			M30		M02		

M02	=1:	To return to block beginning when M02 is to be executed
	=0:	Not to return to block beginning when M02 is to be 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
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 setting

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：1 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 1

System parameter setting

0	3	7						SOC	RSC		SCRW

SCRW =1: To perform pitch compensation
=0: Not perform pitch compensation

RSC =1: To calculate G96 spindle speed according to current coordinate during G0 rapid positioning
=0: To calculate G96 spindle speed according to end point coordinate during G0 rapid positioning

SOC =1: G96 spindle speed clamped behind spindle override
=0: G96 spindle speed clamped before spindle override

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									TLC

TLC =1： Tool length compensation type: B
=0： Tool length compensation type: A

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			CNI	G39			PUIT		OIM

OIM =1: Metric and inch conversion, automatic tool offset change enabled
=0: Metric and inch conversion, automatic tool offset change disabled

PUIT =1: Distance and speed parameters input are consistent with display unit and CNC input unit
=0: Distance and speed parameters units and display unit are metric units

G39 =1: Corner rounding effective in radius compensation
=0: Corner rounding ineffective in radius compensation

CNI =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				RD2	RD1				

RD1 =1: To set the retraction direction of G76，G87: negative
=0: To set the retraction direction of G76，G87: positive

RD2 =1: To set the retraction axis of G76，G87: Y
=0: To set the retraction axis of G76，G87: X

Standard setting：0 0 0 0 0 0 0 0

System parameter number

0	4	3								QZA	

QZA =1: To make alarm if cut-in depth is not specified in peck drilling (G73,G83)
=0: Not to make alarm if cut-in depth is not specified in peck drilling (G73,G83)

Standard setting：0 0 0 0 0 0 1 0

System parameter number

0	4	4				PCP	DOV			VGR	

VGR =1: Arbitrary gear ration of the spindle and position encoder enabled
=0: Arbitrary gear ration of the spindle and position encoder disabled

DOV =1: Override effective during rigid tapping retraction
=0: Override ineffective during rigid tapping retraction

PCP =1: High-speed peck drilling cycle for flexible tapping
=0: Standard peck drilling cycle for flexible tapping

Standard setting：0 0 0 0 0 0 0 0

System parameter number

0	4	5					OVS	OVU	TDR		NIZ

NIZ =1: To perform rigid tapping smoothing
=0: Not perform rigid tapping smoothing

TDR =1: To use the same constant during the rigid tapping advance and retraction
=0: Not use the same constant during the rigid tapping advance and

retraction

OVU	=1:	10% retraction override for rigid tapping
	=0:	1% retraction override for rigid tapping
OVS	=1:	In rigid tapping, selection and cancel signal for federate override enable
	=0:	In rigid tapping, selection and cancel signal for federate override disable

Standard setting：0 0 0 0 0 0 0 0

System parameter number

0	4	6			ORI				SSOG	

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		XSC	SCLz	SCLy	SCLx			RIN

RIN	=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:	Axes scaling override specified by I, J, K
	=0:	Axes scaling override specified by P instruction

Standard setting：0 1 1 1 1 0 0 1

System parameter number

0	4	8								MDL

MDL	=1:	G codes of unidirectional positioning set for modal
	=0:	G codes of unidirectional positioning not set for modal

Standard setting：0 0 0 0 0 0 0 0

System parameter number

0	4	9								RPST

RPST	=1:	Z axis moving by G01 mode at reset
	=0:	Z axis moving by G00 mode at reset

Standard setting：0 0 0 0 0 0 0 0

System parameter number

0	5	0		SIM		G90			REL	

REL =1: Relative position display setting of indexing table: within 360°
=0: Relative position display setting of indexing table: beyond 360°
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 the same block
=0: Do not make alarm if indexing instruction and other axes instructions are in the same block

Standard setting：0 1 0 0 0 0 0 0

System parameter number

0	5	1		MDLY		SBM					

SBM =1: Single block allowed in macro statement
=0: Single block unallowed in macro statement
MDLY =1: Delay is allowed in macro statement
=0: Delay is 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		PLCV				LAD3	LDA2	LAD1	LAD0

LAD0~LAD3 They are binary combination parameters. If they are 0, it uses No. 0 ladder, if they are 1~15, it uses 0~15 ladder diagram.
PLCV =1: Read and display PLC software version number.
=0: Do not read and display PLC software version number

Standard setting：0 0 0 0 0 0 0 1

System parameter number

0	5	4		OPRG							

OPRG =1: Debugging and above authorithies, one key input/output is effective for workpiece program
=0: Debugging and above authorities, one key input/output is ineffective for workpiece program

Standard setting：0 0 0 0 0 0 1 1

Appendix

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				HISR			HPF

HPF =1: To select full running for MPG moving
=0: Not select full running for MPG moving
HISR =1: Use MPG/step pause function
=0: Not use MPG/step pause function
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	8								ROVD

ROVD =1: Band switch is used in rapid operation override debugging
=0: Operation panel keys are used in rapid operation override debugging

Standard setting：0 0 0 0 0 0 0 0

System parameter number

0	5	9		LEDT	LOPT				RHPG	

RHPG =1: Use electronic handwheeel drive function
=0: Not use electronic handwheeel drive function
LOPT =1: Use external operator panel lock
=0: Not use external operator panel lock
LEDT =1: Use external editing lock
=0: Not use external editing lock

Standard setting：0 0 0 0 0 0 0 0

System parameter number

0	6	0			SCL					

SCL =1: Use scaling
=0: Not use scaling

Standard setting：0 0 0 0 0 0 0 0

System parameter number

0	6	1	FALM	LALM	EALM	SALM	AALM			SSC

SSC	=1:	To use constant surface speed control
	=0:	Not use constant surface speed control
AALM	=1:	External user alarm ignored
	=0:	External user alarm not ignored
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 Data Parameter

Parameter number　　Definition　　Default value

0000	I/O channel, input and output device selection	2

Setting range：0～2

It is set to 0 or 1 for communication between CNC and PC via RS232 interface, and set to 2 when CNC connecting with USB flash disk.

0001	Baudrate of communication channel (DNC)	38400

Setting range：0～115200　（unit：BPS）

0002	Baudrate of communication channel (file transfer)	115200

Setting range：0～115200　（unit：BPS）

0004	System interpolation period (1ms,2ms,4ms,8ms)	1

Setting range：1～8

0005	Axes controlled by CNC	3

Setting range：3～5

0006	CNC language selection	0

Setting range：0～3　0: Chinese 1: English 2: Russian 3: Spanish

0008	Allowed value between encoder and machine coordinate system	20.0000

Setting range：0.0010～100.0000

0009	Max. retransmission times of Ethernet bus	30

Setting range：0～255

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)

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）

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）

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）

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）

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）

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）

0045	X coordinate of the 1st reference point in machine coordinate system	0.0000

Setting range：-9999.9999～9999.9999（mm）

0046	Ycoordinate 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）

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）

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）

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)

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)

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	Positive 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)

0086	Dry run speed	5000

Setting range：0～9999 (mm/min)

0087	Cutting federate at power-on	300

Setting range：0～9999 (mm/min)

0088	G0 rapid traverse speed of X axis	5000

Setting range：0～30000 (mm/min)

0089	G0 rapid traverse speed of Y axis	5000

Setting range：0～30000 (mm/min)

0090	G0 rapid traverse speed of Z axis	5000

Setting range：0～30000 (mm/min)

0091	G0 rapid traverse speed of 4TH axis	5000

Setting range：0～30000 (mm/min)

0093	F0 rapid override of axis (for all axes)	30

Setting range：0～1000 (mm/min)

0094	Maximum control speed in rapid positioning (for all axes)	8000

Setting range：300～30000(mm/min)　Maximum control speed in non-forecast mode

0095	Minimum control speed in rapid positioning (for all axes)	0

Setting range：0～300 (mm/min)　Minimum control speed in non-forecast mode

0096	Maximum control speed in cutting feed (for all axes)	6000

Setting range：300～9999 (mm/min)

0097	Minimum control speed in cutting feed (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 return (for all axes)	40

Setting range：1～60 (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	4TH 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：3～400 （ms）

0106	L type time constant of pre-acceleration/deceleration of rapid Y axis	100

Setting range：3～400 （ms）

0107	L type time constant of pre-acceleration/deceleration of rapid Z axis	100

Setting range：3～400 （ms）

0108	L type time constant of pre-acceleration/deceleration of rapid 4Th axis	100

Setting range：3～400 （ms）

0110	S type time constant of pre-acceleration/deceleration of rapid X axis	100

Setting range：3～400 （ms）

0111	S type time constant of pre-acceleration/deceleration of rapid Y axis	100

Setting range：3～400 （ms）

0112	S type time constant of pre-acceleration/deceleration	100

	of rapid Z axis	

Setting range：3～400 （ms）

0113	S type time constant of pre-acceleration/deceleration of rapid 4Th axis	100

Setting range：3～400 （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）

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）

0125	L type time constant of pre-acceleration/deceleration of cutting feed	100

Setting range：3～400 （ms）

0126	S type time constant of pre-acceleration/deceleration of cutting feed	100

Setting range：3～400 （ms）

0127	L type time constant of post acceleration /deceleration of cutting feed	80

Setting range：3～400 （ms）

0128	E type time constant of post acceleration /deceleration of cutting feed	60

Setting range：3～400 （ms）

0129	FL speed of exponential acceleration/deceleration	10

Setting range：0～9999 （mm/min）

0130	Maximum blocks merged in pre-interpolation	0

Setting range：0～10

0131	In-position precision of cutting feed	0.03

Setting range：0.01～0.5 （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：0.01～0.5 （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 spline formation in forecasting	5

Setting range：0～30

0143	Angular condition of spline formation in forecasting	10

Setting range：0～30

0144	Critical angle of two blocks during automatic corner deceleration in forecasting control	5

Setting range：2～178　　(degrees)

0145	Minimum federate 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 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）

Appendix

0157	Exponential acceleration/deceleration time constant of axes JOG feed	120

Setting range：0～400 （ms）

0158	Acceleration clamp time constant of handwheel incomplete running	1

Setting range：0～1000 （ms）

0160	Multiplication coefficient of X axis instruction (CMR)	1

Setting range：1～65536

0161	Multiplication coefficient of Y axis instruction (CMR)	1

Setting range：1～65536

0162	Multiplication coefficient of Z axis instruction (CMR)	1

Setting range：1～65536

0163	Multiplication coefficient of 4TH axis instruction (CMR)	1

Setting range：1～65536

0165	Frequency dividing coefficient of X axis instruction (CMD)	1

Setting range：1～65536

0166	Frequency dividing coefficient of Y axis instruction (CMD)	1

Setting range：1～65536

0167	Frequency dividing coefficient of Z axis instruction (CMD)	1

Setting range：1～65536

0168	Frequency dividing coefficient of 4TH axis instruction (CMD)	1

Setting range：1～65536

0170	X axis manual rapid positioning speed	5000

Setting range：0～30000

0171	Y axis manual rapid positioning speed	5000

Setting range：0～30000

0172	Z axis manual rapid positioning speed	5000

Setting range：0～30000

0173	4TH axis manual rapid positioning speed	5000

Setting range：0～30000

0175	Program name of the 1st axis	0

Setting range：0～8 (0:X 1:Y 2:Z 3:A 4:B 5:C 6:U 7:V 8:W)

0176	Program name of the 2nd axis	1

Setting range：0～8 (0:X 1:Y 2:Z 3:A 4:B 5:C 6:U 7:V 8:W)

0177	Program name of the 3rd axis	2

Setting range：0～8 (0:X 1:Y 2:Z 3:A 4:B 5:C 6:U 7:V 8:W)

0178	Program name of the 4th axis	3

Setting range：0～8 (0:X 1:Y 2:Z 3:A 4:B 5:C 6:U 7:V 8:W)

0179	Program name of the 5th axis	4

Setting range：0～8 (0:X 1:Y 2:Z 3:A 4:B 5:C 6:U 7:V 8:W)

0180	The 1st axis grid/reference point offset amount	0

Setting range：0～50

0181	The 2nd axis grid/reference point offset amount	0

Setting range：0～50

0182	The 3rd axis grid/reference point offset amount	0

Setting range：0～50

0183	The 4th axis grid/reference point offset amount	0

Setting range：0～50

0184	The 5th axis grid/reference point offset amount	0

Setting range：0～50

0189	(X0.0001)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 large radius, in order to make the offset position not to exceed the quadrant, 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：0～0.5 (mm)

0191	Backlash compensation amount of Y axis	0.0000

Setting range：0～0.5 (mm)

Appendix

0192	Backlash compensation amount of Z axis	0.0000

Setting range：0～0.5 (mm)

0193	Backlash compensation amount of 4TH axis	0.0000

Setting range：0～0.5 (mm)

0195	Compensation step of X axis clearance by fixed frequency	0.0030

Setting range：0～0.5 (mm)

0196	Compensation step of Y axis clearance by fixed frequency	0.0030

Setting range：0～0.5 (mm)

0197	Compensation step of Z axis clearance by fixed frequency	0.0030

Setting range：0～0.5 (mm)

0198	Compensation step of 4TH axis clearance by fixed frequency	0.0030

Setting range：0～0.5 (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 by MDI disabled	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：0.0001～0.1000 （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

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 X4TH axis	256

Setting range：0～1000

0226	Pitch error compensation interval of X axis	5

Setting range：0～9999.9999 (mm)

0227	Pitch error compensation interval of Y axis	5

Setting range：0～9999.9999 (mm)

0228	Pitch error compensation interval of Z axis	5

Setting range：0～9999.9999 (mm)

0229	Pitch error compensation interval of 4TH axis	5

Setting range：0～9999.9999 (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

0240	Gain adjustment data for spindle analog output	1

Setting range：0.98～1.02

0241	Compensation value of offset voltage for spindle analog output	0

Setting range：-0.2～0.2

0242	Spindle speed at spindle orientation, or motor speed at spindle gear shift	50

Setting range：0～9999 (r/min)

0243	Maximum setting value to converter	8191

Setting range：4000～8191

0246	Spindle maximum speed to gear 1	6000

Setting range：0～99999 (r/min)

0247	Spindle maximum speed to gear 2	6000

Setting range：0～99999 (r/min)

0248	Spindle maximum speed to gear 3	6000

Setting range：0～99999 (r/min)

0250	Spindle motor speed of gear shifting	50

Setting range：0～1000 (r/min)

0251	Maximum spindle motor speed of shifting	6000

Setting range：0～99999 (r/min)

0254	Axis as counting for surface speed control	0

Setting range：0～4

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	6000

Setting range：0～99999 (r/min)

0261	Spindle encoder lines	1024

Setting range：0～9999

0262	Spindle override lower limit	0.5000

Setting range：0.5～1

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)

0268	Maximum error value of tool radius compensation C	0.0010

Setting range：0.0001~0.0100

0269	Helical infeed radius coefficient in groove cycle	1.5000

Setting range：0.0100～3.0000

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)

0281	Minimum dwell time at the hole bottom	250

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)

0286	Tooth number of spindle side gear (1^{st} gear)	1

Setting range：1～999

Appendix

0287	Tooth number of spindle side gear (2^{nd} gear)	1

Setting range：1～999

0288	Tooth number of spindle side gear (3^{rd} gear)	1

Setting range：1～999

0290	Tooth number of position encoder side gear (1^{st} gear)	1

Setting range：1～999

0291	Tooth number of position encoder side gear (2^{nd} gear)	1

Setting range：1～999

0292	Tooth number of position encoder side gear (3^{rd} gear)	1

Setting range：1～999

0294	Maximum spindle speed in rigid tapping (1^{st} gear)	6000

Setting range：0～9999 (r/min)

0295	Maximum spindle speed in rigid tapping (2^{nd} gear)	6000

Setting range：0～9999 (r/min)

0296	Maximum spindle speed in rigid tapping (3^{rd} gear)	6000

Setting range：0～9999 (r/min)

0298	Linear acceleration/deceleration time constants of spindle and tapping axis (1^{st} gear)	200

Setting range：0～9999 （ms）

0299	Linear acceleration/deceleration time constants of spindle and tapping axis (2^{nd} gear)	200

Setting range：0～9999 （ms）

0300	Linear acceleration/deceleration time constants of spindle and tapping axis (3^{rd} gear)	200

Setting range：0～9999 （ms）

0302	Time constant of spindle and tapping axis in retraction (1^{st} gear)	200

Setting range：0～9999 （ms）

0303	Time constant of spindle and tapping axis in retraction (2^{nd} gear)	200

Setting range：0～9999 （ms）

0304	Time constant of spindle and tapping axis in retraction (3^{rd} gear)	200

Setting range：0～9999 （ms）

0320	Spindle clearance in rigid tapping (1^{st} gear)	0

Setting range：0～99.9999

0321	Spindle clearance in rigid tapping (2^{nd} gear)	0

Setting range：0～99.9999

0322	Spindle clearance in rigid tapping (3^{rd} gear)	0

Setting range：0～99.9999

0323	Spindle instruction multiplication coefficient (CMR) (1^{st} gear)	512

Setting range：0～9999

0324	Spindle instruction multiplication coefficient (CMR) (2^{nd} gear)	512

Setting range：0～9999

0325	Spindle instruction multiplication coefficient (CMR) (3^{rd} gear)	512

Setting range：0～9999

0326	Spindle instruction frequency dividing coefficient (CMD) (1^{st} gear)	125

Setting range：0～9999

0327	Spindle instruction frequency dividing coefficient (CMD) (2^{nd} gear)	125

Setting range：0～9999

0328	Spindle instruction frequency dividing coefficient (CMD) (3^{rd} gear)	125

Setting range：0～9999

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：0.0001～9999.9999

0331	Scaling of X axis	1

Setting range：0.0001～9999.9999

0332	Scaling of Y axis	1

Setting range：0.0001～9999.9999

0333	Scaling of Z axis	1

Setting range：0.0001～9999.9999

0334	Dwell time 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

0341	ARM interpolation point buffer size	36

Setting range：0～99999

0354	DSP unsuccessful start times	0

Setting range：0～999999

0355	CNC successful start times	0

Setting range：0～999999

0356	Workpiece machined	0

Setting range：0～9999

0357	Total workpiece to be machined	0

Setting range：0～9999

0358	Accumulative time of power-on (h)	0

Setting range：0～99999

0359	Accumulative time of days (days)	0

Setting range：0～99999

0360	Accumulative time of cutting (h)	0

Setting range：0～99999

0361	Register parameter of the year	0

Setting range：0～24

0362	Register parameter of the month	0

Setting range：0～12

0363	Register parameter of the day	0

Setting range：0～31

0371	Positioning error allowable for reverse X axis	0.0150

Setting range：0～99.9999 (mm)

0372	Positioning error allowable for reverse Y axis	0.0150

Setting range：0～99.9999 (mm)

0373	Positioning error allowable for reverse Z axis	0.0150

Setting range：0～99.9999 (mm)

0374	Positioning error allowable for reverse 4TH axis	0.0150

Setting range：0～99.9999 (mm)

When the set backlash compensation value (P0190---P0193) of an axis is bigger than the reverse positioning allowable error (P0371---P0374) of this axis, the speed at the end point of a single block reduces to minimum speed before this backlash compensation begins. This will make the other axes move a small distance in the backlash compensation period, and that will ensure the resultant path deviates the real path least.

0376	Axes moving sequence to program beginning	12345

Setting range：0～99999

0380	Date of the record file	0

Setting range：-999.0000～999.0000

0381	Line of the record file	0

Setting range：-999.0000～999.0000

0382	Instalment times	0

Setting range：0～24

0383	Arrived payment	0

Setting range：0～24

0384	Delay times for each instalment	0

Setting range：0～120

0385	Time limit for incorrect password allowed input	0

Setting range：0～5

0387	X axis positioning value for tool setting machine on G53	0

Setting range：-999.0000～999.0000

0388	Y axis positioning value for toolsetting machine on G53	0

Setting range：-999.0000～999.0000

0389	Z axis positioning value for tool setting machine on G53	0

Appendix

Setting range：-999.0000～999.0000

0393	Estimated length from tool nose to tool holder	50

Setting range： 0.0000～999.0000

0395	Offset value of last toolsetting	0

Setting range：-999.0000～999.0000

0396	X axis backup of coordinate system	0

Setting range： 0～0

0397	Y axis backup of coordinate system	0

Setting range： 0～0

0398	Z axis backup of coordinate system	0

Setting range： 0～0

0399	Multiple of interpolation step length	1.5

Setting range： 1.0000～10.0000

0400	Shape matching parameter	10

Setting range：0.0020～99.0000

Shape matching parameter (#400) is to control error in a permissible range through shape error analyzing and shape optimization based on initial spline curve.
The bigger the parameter is, the bigger the shape error will be, and vice versa.

0401	Shape matching limit	15

Setting range：1.0000～999.000

When shape matching limit parameter (#401) is performing velocity matching calculation, the parameter will prevent shape error increasing caused by curvature optimization.

0402	Velocity matching parameter	1

Setting range：0.0020～99.0000

Velocity matching parameter (#402) is to smooth velocity by optimizing curvature, in which curvature is radially distributed along normal direction of each point on the curve.
The bigger the parameter, the lower the optimization, the bigger the acceleration and the shorter the machining time.
The smaller the parameter, the higher the optimization and the longer the machining time.

0403	Fitting segments of small lines	5

Setting range：0.0020～999.0000

The parameter (#403) determines the number of tool location points of the fitting spline curve. The parameter should be controlled in a certain range.
#403 = 1~10 The bigger the parameter, the bigger the calculation amount, and the smaller the shape error.

The smaller the parameter, the smaller the calculation amount, while the bigger the shape error.

0404	Spline coefficient n1	30

Setting range：1.0000～199.0000

0405	Spline coefficient n2	30

Setting range：1.0000～199.0000

0406	Spline coefficient n3	30

Setting range：1.0000～199.0000

An original cubic spline curve is fitted based on spline parameters n1,n2,n3 (#404、#405、#406). The bigger the spline coefficient n1,n2（#404、#405）, the bigger the curve error, while speed is more smooth, and the machine tool is more stable. The smaller the coefficient, the smaller the curve error, while the speed is not smooth and machine tool vibration occurs. The spline coefficient n3（#406）is opposite.

0407	CNC internal parameter 1	0.3000

Setting range：0.0020～99.0000

0408	CNC internal parameter 2	0.3000

Setting range：0.0020～99.0000

0409	Prereading smooth control	2.0000

Setting range：0.0000～30.0000

Prereading smooth control (#409) is used to reduce machining slash caused by CAM program errors through prereading the machining shape, automatically calculating the whole shape.

0： Stop prereading smooth control function

1： Perform smooth processing according to the length

2： Perform smooth processing according to the length and the angle

0410	Precision smooth and balance coefficient	10.0000

Setting range：0.0000～10.0000

To realize high precision control, user only needs to set parameter value of precision smooth and balance coefficient. The parameter, which includes 0-10, 11 grades in total, can control the grade of machining effect.

#410 = 0: indicates high precision control. In-position precision rather than smooth is strictly controlled. It is especially beneficial for machining the materials with high requirements for subtle edges and corners (such as characters).

=1-10: Return to high speed and high precision control. The lower the grade, the better the precision. The higher the grade, the better the smoothness.

The parameter can be adjusted to achieve the best results according to the actual machining situation.

0411	Spline shape control coefficient	50.0000

Setting range：0.0000～50.0000

0412	Fitting precision control of small lines	-1.0000

Setting range：-10.0000～50.0000

0413	Roundness smooth control coefficient n1	3.0000

Setting range：0.0000～50.0000

0414	Roundness smooth control coefficient n2	0.0000

Setting range：0.0000～50.0000

APPENDIX II 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	There is no address but figure or character "-" at the beginning of the block. Modify the program	
0005	There is no appropriate data but another address or EOB code behind the address. Modify the program	
0006	Sign "-" input is wrong (One or more "-" signs are input behind the address where negative sign can not be used). Modify the program	
0007	Decimal point "." input is wrong (One or more "." signs are input in the address where the sign can not be used). Modify the program	
0008	The program file is too large. Please use CNC to transmit it	
0009	Illegal address input. Modify the program.	
0010	G code wrong. Modify the program	
0011	Feedrate is not specified or it is wrong in cutting feed. Modify the program	
0012	Disk space is not enough. Setup or add file is not allowed	
0013	The program files are up to the upper limit. New program can not be setup	
0014	G95 can not be specified, it is not supported by the spindle	
0015	Exceed the number of simultaneously controlled axes	
0016	Current pitch compensation beyond range	
0017	No authority to modify	
0018	Dummy variable and local variable are not allowed to modify. G10 only to modify parameter of user grade	
0019	Scaling function is OFF. Please use bit parameter 60.5 to make it active	
0020	In circular interpolation (G02 or G03), the distance between the start point and the circle center is not equal to the distance between the end point and the circle center. The value beyond the one specified by parameter 214	
0021	In circular interpolation, illegal axis is specified. Modify the program	
0022	In circular interpolation, R (radius), I, J and k (distance from the start point to the center) are not be specified	
0023	In circular interpolation, I, J, K and R are specified together	
0024	Helical interpolation rotation angle is 0	
0025	G12 and other G code can’t be in a same block	
0026	Unsupported file format. It is too large or with above 1024 bytes	
0027	Tool length compensation instruction can not be in the same block with G92. Modify the program	
0028	In plane selection instructions, two or more axes are specified at the same direction. Modify the program	

0029	The compensation value specified by D/H is too big. Modify the program	
0030	Tool length compensation number or tool radius compensation number specified by D/H code is too big. Workpiece coordinate number specified by P is too big	
0031	Illegal P specified in G10	
0032	Compensation value is too big or it is not specified. Modify the program	
0033	The intersecting point of offset C or chamfer is not confirmed. Modify the program	
0034	Set-up or offset cancel are not allowed in circular instruction	
0035	Tool compensation C should be cancelled before M99 instruction	
0036	G31 is specified in tool compensation	
0037	The plane selected by G17, G18 or G19 is changed in tool compensation C	
0038	In tool compensation C, overcutting will occur	
0039	Tool nose positioning error in tool compensation C	
0040	Cancel the tool compensation before changing the workpiece coordinate	
0041	Interference occurs in tool compensation C will lead overcutting	
0042	Ten blocks with stop tool instruction are specified in tool compensation mode. Modify the program	
0043	No authority. Change it in password page	
0044	In canned cycle, one of instruction in G27，G28，G29，G30 is specified	
0045	In canned cycle G73/G83, cutting depth (Q) is not specified or it is 0	
0046	In 2nd, 3rd, 4th reference return instructions, instruction besides P2, P3 and P4 is specified	
0047	Perform machine zero return before executing instructions G28，G30，G53	
0048	In canned cycle, plane Z is higher than plane R	
0049	In canned cycle, plane Z is lower than plane R	
0050	Move it when changing canned cycle mode	
0051	Wrong movement or distance is specified after rounding or chamfering	
0052	Mirror image function can not be used in grooving canned cycle	
0053	Wrong instruction format for rounding or chamfering	
0054	DNC transmission error	
0055	Chamfer failed	
0056	M99 shall not in the same block with macro instruction G65	
0057	File input failed. Cut off the power and reset it	
0058	In block of rounding or chamfering, specified axis is not in the selected plane	
0059	Program number is not found in external program retrieving or it is edited in background. Check program number or external signal, or stop background editting	
0060	Specified sequence number is not found in retrieving. Check sequence number	
0061	The reference point is not in X axis	

0062	The reference point is not in Y axis	
0063	The reference point is not in Z axis	
0064	The reference point is not in 4TH axis	
0065	The reference point is not in 5TH axis	
0066	Cancel canned cycle mode before inputting parameter (G10)	
0067	G10 does not support the set format	
0068	Parameter switch is not switched on	
0069	U-disk operation page should be closed when machining	
0070	Insufficient memory. Delete unneeded programs and try it again	
0071	The address is not found	
0072	Too many programs. 63 (basic), 125 (optional), 200 (optional) or 400 (optional). Delete unnecessary programs	
0073	Program number already in use. Change the program number or delete unneeded program	
0074	Illegal program number (beyond the range 1-99999). Change the program number	
0075	To register a protected program number	
0076	Address P (program name) is not specified in block M98. Modify the program	
0077	Program nesting exceed 5 layers	
0078	In blocks M98, G65, program name specified by address P is not found or macro program called by M06 does not exist	
0079	CNC expires the using date. Please contact the supplier	
0080	Input data is wrong, Max. speed is smaller than Min. speed or Min. speed is bigger than Max. speed	
0081	Subprogram can not be called	
0084	Overtime or short circuit occurs in key	
0085	Overflow occurs when data is transmitted to memory by series port. Baud rate setting or I/O equipment is wrong	
0086	Planes can not be shifted in canned cycle mode	
0087	Alarm NO.0087~0091 are for reference point return unfinished (starting point of reference return is too close to the reference point or the speed is too slow).	
0092	G27(check for reference return) instruction can not return to the reference point	
0093	Motor type error	
0098	After power-on or emergency stop, when the program with G28, program restarts without executing reference return	
0100	On parameter (setting) screen, PWE (parameter input is active) is set to 1. Restart CNC after setting it to 0.	
0101	Memory data disordered after power off, please ensure correct location	
0102	Driver motor does not match CNC	
0103	Bus communication error. Please check reliability of the cable	
0104	Machine zero point setting error	
0105	Time-out error while data is being fetched	

Appendix

0106	Drive unit is not consistent with gear ratio of servo parameter	
0107	Drive unit parameter is not consistent with servo unit parameter	
0108	Please insert U-disk	
0110	Position data exceeds the allowed range. Please reset	
0111	Calculated result exceeds the allowed range (-1047 to -10-29，0 and 10-29 to 1047)	
0112	Zero (including tan900) is specified as a divisor	
0113	Unusable functional instruction is specified in user macro program. Modify the program	
0114	G39 format error. Modify the program	
0115	Variable value can not be specified. O, N can not be specified as variables in user macro program	
0116	A variable is on the left of the assignment statement, while value assignment to it is not allowed. Modify the program	
0117	G10 online modification is not supported by this parameter. Please modify the program	
0118	Nest exceeds the upper limit (5). Modify the program	
0119	Instructions M00,M01,M02,M30,M98,M99,M06 can not in a same block with other M instructions	
0120	Part of setting is restored	
0121	Machine coordinates and encoder feedback values exceed setting value of error	
0122	Called nests of macro program exceed 5 layers. Modify the program	
0123	Macro program is used in DNC operation. Modify the program	
0124	Program end illegally, without M30, M02, M99 or end sign. Modify the program	
0125	Macro program format error. Modify the program	
0126	Program cycle failure. Modify the program	
0127	NC coexists with user macro instruction statement. Modify the program	
0128	Sequence number in branch instruction is not at the range 0-99999, or the number is not found. Modify the program	
0129	The address of argument assignment. Modify the program	
0130	PLC axis control instruction is input to the axis controlled by CNC, or opposite. Modify the program	
0131	5 or more external alarm signals occur. Check the ladder diagram	
0132	The alarm of the external alarm signal does not exist. Check PLC	
0133	The system does not support axis instruction. Modify the program	
0134	Rigid tapping can not be used when CNC controlled axes exceed 3	
0135	Illegal angle instruction. Modify the program	
0136	Illegal axis instruction. Modify the program	
0137	Sequence number to be transferred by skip instruction is in loop body	
0138	Cycle statement is wrong or skip instruction enters loop body	
0139	PLC axis change disabled	

0140	Sequence number does not exist	
0141	MDI presentation module and DNC mode do not support macro instruction skip	
0142	Illegal scaling beyond 1-999999 is specified	
0143	Scaling, moved distance, coordinate value and radius exceed max. instruction value	
0144	Coordinate rotational plane, arc or tool radius compensation C should be the same one	
0145	G28 is specified before defining reference point. Please modify the program or parameter NO.4#3(AZR)	
0148	Illegal data setting	
0160	Arc programming only by R in polar system	
0161	Reference point, plane selection or direction-related instructions can not be executed in polar coordinate mode	
0163	Reference point or coordinate system-related G instructions can not be executed in revolution mode	
0164	Reference point or coordinate system-related G instructions can not be executed in scaling mode	
0165	Please specify revolution, scaling or G10 instructions in a single block	
0166	No axis specified in reference return	
0167	Intermediate point coordinate too large	
0168	The min. dwell time at the hole bottom should be shorter than the max. dwell time	
0170	Tool radius compensation is not cancelled while entering or exiting subprogram	
0172	P is not an integer or less than 0 in a block calling subprogram	
0173	Subprogram call should be less than 9999	
0175	Canned cycle can only be executed in G17 plane	
0176	Spindle speed is not specified before rigid tapping	
0177	Spindle orientation is not supported by IO control in G76 instruction	
0178	Spindle speed is not specified in canned cycle	
0181	Illegal M code	
0182	Illegal S code	
0183	Illegal T code	
0184	Tool selection beyond range	
0185	L is too small: 1) L is smaller than tool radius in rectangular groove fine milling 2) L is smaller than 0 in groove rough milling	
0186	L is too big: 1) L is bigger than tool diameter in inner circular groove rough milling 2) L is bigger than tool diameter in rectangular groove rough milling 3) L is bigger than I in rectangular groove rough milling 4) L is bigger than J in inner circular groove rough milling	
0187	Tool diameter is too big: 1) Tool diameter is bigger than I in inner circular groove rough milling 2) Tool radius is bigger than I-J in inner circular groove rough milling	

	3) Tool radius is bigger than J in outer circular groove fine milling 4) Tool diameter is bigger than I in rectangular groove fine/rough milling 5) Tool diameter is bigger than J in rectangular groove fine/rough milling 6) Tool radius is bigger than U in rectangular groove fine/rough milling 7) Radius coefficient of helical infeed is too big or D is too big. Modify parameter No.269 or radius compensation value	
0188	U is too big: 1) Twice of U in rectangular groove cycle is bigger than I 2) Twice of U in rectangular groove cycle is bigger than J	
0189	U is too small, U should bigger than or equal to tool radius	
0190	V is too small or it is undefined. V should be bigger than 0	
0191	W is too small or it is undefined. W should be bigger than 0	
0192	Q is too small or it is undefined. Q should be bigger than 0	
0193	I is undefined or it is 0	
0194	J is undefined or it is 0	
0195	D is undefined or it is 0	
0198	In constant surface cutting speed control, specified axis error (see parameter No.254)	
0199	Macro instruction modification program is not defined	
0200	In rigid tapping, illegal S instruction	
0201	F value is not found in rigid tapping	
0202	Assigned value of the spindle is too big in rigid tapping	
0203	Position of M code (M29) or S instruction is wrong in rigid tapping	
0204	M29 should be specified in G80 mode	
0205	G84 (or G74) is executed after specifying M code (M29), rigid tapping signal is not 1. Check ladder diagram to find the reason	
0206	Plane shifting is specified in rigid tapping	
0207	The specified distance in rigid tapping is too long or too short	
0208	This instruction can not be executed in G10 mode. Please cancel G10 mode first	
0209	Restart of the program is not supported by scaling, revolution, polar coordinate modes	
0210	Program name error	
0212	Chamfer or R is specified, or other axis is specified in plane	
0213	Tool changing macro program does not support G31 skip	
0214	Tool changing macro program does not support skip operation	
0215	Tool changing macro program does not support modifying coordinate system and tool compensation dynamically	
0216	Scaling, revolution and polar coordinate do not support G31 skip	
0217	Scaling, revolution and polar coordinate do not support skip operation	
0218	Scaling, revolution and polar coordinate do not support modifying coordinate system and tool compensation dynamically	
0219	M06 Tool magazine is not used (parameter is not opened). Tool changing instruction can not be used	

Appendix

0220	Metric/inch switching is not supported by scaling, revolution and polar coordinate mode	
0221	Metric/inch switching is not supported by tool changing macro program	
0224	Reference return is not performed before auto run started	
0231	Parameter format error: 1) N or R is not input 2) Parameter number is not defined 3) Address P is not defined in bit parameter input L50 4) N，P，R exceed the range	
0232	3 or more axes are specified as helical interpolation axis	
0233	Device connected to RS-232-C is being used	
0235	Specified record end sign (%)	
0236	Parameter setting of program restart is wrong	
0237	No decimal point	
0238	Address repetition error,	
0239	An illegal G code is specified in pre-reading control mode. In pre-reading control mode, dividing spindle is specified, max. cutting feeding parameter is set to 0 and interpolation pro-acceleration/deceleration parameter is set to 0	
0241	MPG pulse is abnormal	
0242	Bus connection error	
0250	Axis name repeated, please modify parameter NO.175~179	
0251	Emergency stop alarm, perform zero return again after canceling the alarm	
0252	Program ends illegally (CNC transmission speed is low, please reduce feedrate)	
0261	Pulse instruction of DSP interpolation axis is too big. Perform zero return again after reset	
0262	DSP is not started. Please power on again	
0263	DSP parameter setting error	
0264	DSP alarm. Data is too big	
0265	DSP alarm. The bus can not be connected or bus initialization failure	
0266	Speed of DSP interpolation axis exceeds 200M/MIN. Perform zero return again after reset	
0267	DSP initial sign (5555) is abnormal. Perform zero return again after reset	
0268	DSP pulse output volume per revolution is too big. Perform zero return again after reset	
0269	DSP internal alarm. Perform zero return again after reset	
0270	Length of DSP equally distributed interpolation point is too small	
0271	DSP received interpolation data is too small. Perform zero return again after reset	
0272	DSP received undistinguishable G code	
0273	DSP hardware data interchange is abnormal (instructions)	
0274	DSP hardware data interchange is abnormal (data)	
0275	In high-speed mode, interpolation multiple is 0	

0280	Perform axes zero return before using tool setting function	
0281	Switch to [SET] [Halving] interface before using tool setting function	
0282	Please check whether toolsetting gauge is installed or parameter 1.6 is set	
0283	Z axis exceeds safety position， please check toolsetting gauge or tool length setting	
0286	Automatic tool length measurement is wrong. Please measure it again	
0300	n-axis origin return	
0301	APC alarm: n-axis communication error, data transmission error. Possible reasons: APC error, cable or servo interface module faults	
0302	APC alarm: n-axis overtime, data transmission error. Possible reasons: APC error, cable or servo interface module faults	
0303	APC alarm: n-axis data format error, data transmission error. Possible reasons: APC error, cable or servo interface module faults	
0304	APC alarm: n-axis parity error, data transmission error. Possible reasons: APC error, cable or servo interface module faults	
0305	APC alarm: n-axis pulse error. APC or cable error	
0306	APC alarm: n-axis battery voltage too low	
0307	APC alarm: n-axis battery voltage too low. The battery should be changed	
0308	APC alarm: n-axis battery voltage too low. The battery should be changed	
0309	Machine zero point return is performed before motor run.	
0401	Drive unit alarm 01: speed of servo motor exceeds set value	
0402	Drive unit alarm 02: power of spindle circuit is too high	
0403	Drive unit alarm 03: main circuit power source is too low	
0404	Drive unit alarm 04: value of position deviation counter exceeds set value	
0405	Drive unit alarm 05: motor temperature is too high	
0406	Drive unit alarm 06: speed regulator is saturated for a long time	
0407	Drive unit alarm 07: CCW, CW input prohibition OFF	
0408	Drive unit alarm 08: absolute value of value of position deviation counter exceeds 230	
0409	Drive unit alarm 09: encoder signal error	
0410	Drive unit alarm 10: control power ±15V is too low	
0411	Drive unit alarm 11: IPM intelligent module failures	
0412	Drive unit alarm 12: motor current is too large	
0413	Drive unit alarm 13: servo drive unit and motor overload (instantaneous overheat)	
0414	Drive unit alarm 14: brake circuit fault	
0415	Drive unit alarm 14: encoder counter fault	
0420	Drive unit alarm 20: EEPROM error	
0430	Drive unit alarm 30: encoder Z pulse error	
0431	Drive unit alarm 31: encoder UVW signal error or it does not match encoder	
0432	Drive unit alarm 32: UVW with all high level or with all low level	

0433	Drive unit alarm 33: communication interrupted	
0434	Drive unit alarm 34: encoder speed is abnormal	
0435	Drive unit alarm 35: encoder state is abnormal	
0436	Drive unit alarm 36: encoder counter is abnormal	
0437	Drive unit alarm 37: single circle number of encoder overflow	
0438	Drive unit alarm 38: multi circle number of encoder overflow	
0439	Drive unit alarm 39: encoder battery alarm	
0440	Drive unit alarm 40: no battery in encoder	
0441	Drive unit alarm 41: motor type error	
0442	Drive unit alarm 42: absolute position data abnormal alarm	
0443	Drive unit alarm 43: encoder EPPROM check alarm	
0449	Ethernet initialization failure. Please check hardware	
0450	Drive unit is disconnected. Please check whether connection of hardware is correct	
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(release it manually or MPG moves in +X direction to release it)	
0501	Software overtravel: +X(release it manually or MPG moves in -X direction to release it)	
0502	Software overtravel: -Y(release it manually or MPG moves in +Y direction to release it)	
0503	Software overtravel: +Y(release it manually or MPG moves in -Y direction to release it)	
0504	Software overtravel: -Z(release it manually or MPG moves in +Z direction to release it)	
0505	Software overtravel: +Z(release it manually or MPG moves in -Z direction to release it)	
0506	Software overtravel: -4TH (release it manually or MPG moves in +4TH direction to release it)	
0507	Software overtravel: +4TH (release it manually or MPG moves in -4TH direction to release it)	
0508	Software overtravel: - 5Th (release it manually or MPG moves in +5Th direction to release it)	
0509	Software overtravel: + 5Th (release it manually or MPG moves in -5Th direction to release it)	
0510	Hardware overtravel: -X(release it manually or MPG moves in +X direction to release it)	
0511	Hardware overtravel: +X (release it manually or MPG moves in -X direction to release it)	
0512	Hardware overtravel: -Y (release it manually or MPG moves in +Y direction to release it)	

0513	Hardware overtravel: +Y (release it manually or MPG moves in -Y direction to release it)	
0514	Hardware overtravel: -Z (release it manually or MPG moves in +Z direction to release it)	
0515	Hardware overtravel: +Z (release it manually or MPG moves in -Z direction to release it)	
0516	Hardware overtravel: -4TH (release it manually or MPG moves in +4TH direction to release it)	
0517	Hardware overtravel: +4TH (release it manually or MPG moves in -4TH direction to release it)	
0518	Hardware overtravel: - 5Th (release it manually or MPG moves in +5Th direction to release it)	
0519	Hardware overtravel: + 5Th (release it manually or MPG moves in -5Th direction to release it)	
1001	Address of relay or coil is not set	
1002	Function code of input code does not exist	
1003	Function instruction COM is not used correctly. Corresponding relationship between COM and COME is wrong, or function instruction is used between COM and COME	
1004	User ladder beyond the maximum permissible linage or step number. Reduce NET number	
1005	Incorrect END1 or END2 functional instruction is used	
1006	Illegal output in NET	
1007	PLC communication failure due to hardware failure or system interruption.	
1008	Functional instruction is not linked correctly	
1009	Network horizontal line is not linked	
1010	Editing NET losses due to power-off in ladder editting	
1011	Address or data format is not the one specified by this function	
1012	Address or data is wrongly input	
1013	Illegal character or data defined	
1014	CTR address repeated	
1015	Functional instruction is wrongly used. Correspondence between JMP and LBL is wrong. JMP is used again between JMP and LBL	
1016	Incomplete network structure	
1017	Unsupported network exists. Change the ladder diagram	
1019	TMR address repeated	
1020	No parameter in functional instruction	
1021	PLC stops automatically by CNC when PLC execution overtime	
1022	Please input the name of functional code	
1023	Address or constant of functional instruction parameter is out of range	
1024	Unnecessary relay or coil exists	
1025	Functional instruction output wrongly	
1026	NET link linage beyond the supported range	
1027	Same output address is used in another place	

1028	File format wrong	
1029	File losses from ladder diagram being used	
1030	False vertical line in network	
1031	Message data area is full. Please reduce COD code data list capacity	
1032	First level of ladder diagram is too large to complete execution on time	
1033	SFT instructions beyond the max. allowed number	
1034	Functional instruction DIFU/DIFD address is repeated	
1039	Instruction or network beyond executable area	
1040	Functional instruction CALL or SP is wrongly used. Correspondence between CALL and SP or between SP and SPE is wrong. SP functional instruction is used again between SP and SPE or SP is set before using END2	
1041	Level conducting line in parallel with node network	
1042	PLC system parameter file is not loaded	