Warning and Precaution

Accident may occur by improper connection and operation！This system can only be operated by authorized and qualified personnel.

Please read this manual carefully before operation！

Please read this manual and a manual from machine tool builder carefully before installation, programming and operation, and strictly observe the requirements.

This manual includes the precautions for protecting user and machine tool. The precautions are classified into Warning and Caution according to their bearing on safety, and supplementary information is described as Note. Read these Warnings, Cautions and Notes carefully before operation.

Warning

User may be injured or equipment be damaged if operation instructions and procedures are not observed.

Caution

Equipment may be damaged if operation instructions or procedures are not observed.

Note

It is used to indicate the supplementary information other than Warning and Caution.

Precautions

■ Delivery and storage

- Packing box over 6 layers in pile is unallowed.
- Never climb the packing box, neither stand on it, nor place heavy objects on it.
- Do not move or drag the products by the cables connected to it.
- Forbid collision or scratch to the panel and display screen.
- Avoid dampness, insolation and drenching.

■ Open-package inspection

- Confirm that the products are the required ones.
- Check that the products are not damaged in delivery.
- Confirm that the parts in packing box are in accordance with the packing list.
- Contact us in time if any inconsistence, shortage or damage is found.

■ Connection

- Only qualified personnel can connect the system or check the connection.
- The system must be earthed, and the earth resistance must be less than 0.1Ω. The earth wire cannot be replaced by zero wire.
- The connection must be correct and firm to avoid any fault or unexpected consequence.
- Connect with surge diode in the specified direction to avoid damage to the system.
- Switch off power supply before plugging out or opening electric cabinet.

■ Troubleshooting

- Only competent personnel are supposed to inspect the system or machine.
- Switch off power supply before troubleshooting or changing components.
- Check for fault when short circuit or overload occurs. Restart can only be done after troubleshooting.
- Frequent switching on/off of the power is forbidden, and the interval time should be at least 1 min.

Announcement

- This manual describes various possibilities as much as possible. However, operations allowable or unallowable cannot be explained one by one due to so many possibilities that may involve with, so the contents that are not specially stated in this manual shall be considered as unallowable.

Caution

- Functions, technical indexes (such as precision and speed) described in this user manual are only for this system. Actual function deployment and technical performance of a machine tool with this CNC system are determined by machine tool builder's design, so functions and technical indexes are subject to the user manual from machine tool builder.
- Refer to the user manual from machine tool builder for function and meaning of keys on control panel.

Safety Responsibility

Manufacturer's Responsibility

——Be responsible for the danger which should be eliminated and/or controlled on design and configuration of the provided CNC systems and accessories.
——Be responsible for the safety of the provided CNC systems and accessories.
——Be responsible for the provided information and advice for the users.

User's Responsibility

——Be trained with the safety operation of CNC system and familiar with the safety operation procedures.
——Be responsible for the dangers caused by adding, changing or altering to the original CNC systems and the accessories.
——Be responsible for the failure to observe the provisions for operation, adjustment, maintenance, installation and storage in the manual.

All specifications and designs herein are subject to change without further notice.

Contents

Ⅰ GENERAL

1 GENERAL

About this manual

This manual consists of the following parts:

1. GENERAL

Describes chapter organization, related manuals, and notes for reading this manual.

2. PROGRAMMING

Describes each function: format used to program functions in the NC language, characteristics, and restrictions.

3. OPERATION

Describes the manual operation and automatic operation of a machine, procedures for MDI and editing a program.

APPENDIX

Lists alarm codes.

1.1 General

GSK 25i Milling Machining CNC system (hereinafter referred to as the system) is a new generation of CNC device, developing by our company with full heart. It is featured by high precision, great performance, 5 axes simultaneous control and closed-loop control (half closed-loop control and full closed-loop control) and can be widely applied in CNC milling machine and machining center.

This manual detailedly describes procedures for programming, operation of a machine, and introduction for parameter, and inputting and outputting data.

Optional functions are also described in this manual, but not all of them are involved in the actual device. Look up the optional functions incorporated into your system in the manual written by the machine tool builder.

1.2 Notes for Reading this Manual

The performance of a machine tool not only depends on the CNC system, but also the strong current circuit of machine tool, the servo device, the CNC controller and the machine operation control. However, it’s impossible for us to describe all of the functions and procedures of programming and operation in this manual, only the functions of CNC system is presented in it. For various machining functions of a machine tool, refer to the manual provided by the machine tool builder.

All the items described in this manual are prior to that of the manual written by the machine tool builder.

This manual describes items concerning the operation of the system as much as possible. However, it is impractical and unnecessary to present all the descriptions, and the undescribed ones are explained in this manual accordingly.

This manual makes explanations for some special items in notes.

Ⅱ PROGRAMMING

1 GENERAL

1.1 Definition

To a CNC machine tool, a written program is needed to operate the machine. For example, when machining a part, the tool path and other machining conditions should be programmed in advance, this program is called part program.

1.2 Program Configuration

Program consists of a group of blocks while a block consists of several words. Each block is separated by end-of-block code "; "(LF in the ISO code and CR in the EIA code).

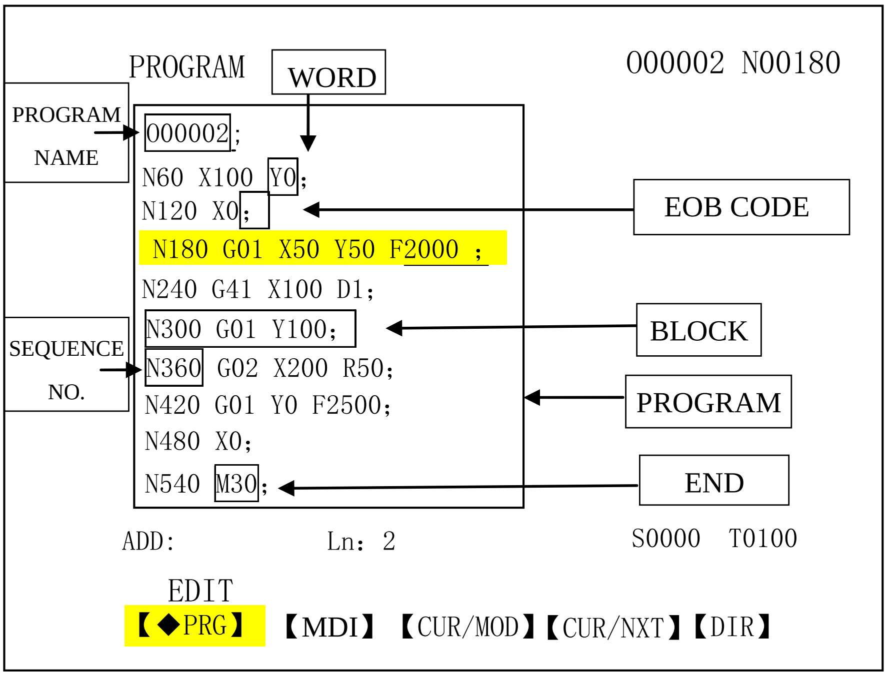


Fig. 1-1 Program configuration

The assembly of commands to complete machining is called program. After a program is input to CNC system, commands such as linear/circular movement of tool, spindle rotation/stop can be performed. The program should be written in accordance with the actual move sequence of a machine tool. Program configuration is shown in Fig. 1-1.

1.2.1 Program Name

This system is able to store several different programs. A program name consisting of the address O followed by four-digit number is assigned to each program at the beginning to identify them. Shown in Fig. 1-2.

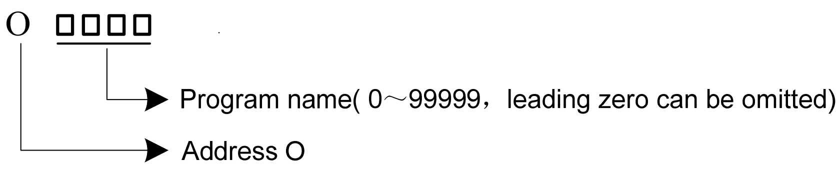


Fig. 1-2 Block configuration

1.2.2 Sequence Number and Block

A program consists of several commands. One command unit is called a block (see Fig. 1-1). One block is separated from another with “; ” as the end of block code. (See Fig. 1-1)

At the head of a block, a sequence number consisting of address N followed by six-digit numbers can be placed (see Fig. 1-1). The leading zero can be omitted. Sequence number can be specified in a random order, and any number can be skipped. Sequence number may be specified for all blocks or only for important blocks of a program. In general, however, it is convenient to assign sequence numbers in ascending order in phase with the machining steps. (For example, when a new tool is used by tool replacement and machining proceeds to a new surface with table indexing.)

1.2.3 Word

Word is an essential for a block. A word consists of an address followed by a number some digit long. (The plus sign (+) or minus sign (-) may be prefixed to a number.)

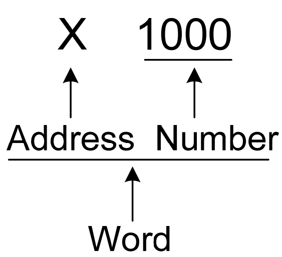


Fig. 1-3 Word configuration

For an address, one of the letters (A to Z) is used. An address defines the meaning of a number that follows the address. Table1-1 indicates the usable address and their meanings.

The same address may have different meanings, depending on the preparatory function specification.

Table 1-1

Address	Ranges	Function and Meaning
O	0～99999	Program name
N	0～999999	Sequence number
G	000～999	Preparatory function
X	-999999.9999～999999.9999（mm）	X-coordinate address
	0～9999.9999（s）	Dwell time
Y	-999999.9999～999999.9999（mm）	X-coordinate address
Z	-999999.9999～999999.9999（mm）	X-coordinate address
R	-999999.9999～999999.9999（mm）	Shift amount of circular radius/angle
	-999999.9999～999999.9999（mm）	R surface of canned cycle
I	-999999.9999～999999.9999（mm）	X vector between arc center and starting point
J	-999999.9999～999999.9999（mm）	Y vector between arc center and starting point
K	-999999.9999～999999.9999（mm）	Z vector between arc center and starting point
F	0.1～1000000（mm/min）	Feedrate per minute
	0.001～10000(mm/r)	Feedrate per revolution
S	0～50000（r/min）	Specifying spindle speed
	00～06	Multi-gear spindle output
T	0～999	Tool function
M	00～999	Miscellaneous function output, program executed flow, subprogram call
P	0～9999（s）	Dwell time
	1～99999	Call subprogram number
Q	-999999.999～999999.999（mm）	Cutting depth or offset amount for low hole in canned cycle
H	00～256	Length offset number
D	00～256	Radius offset number

Please note that Table 1-1 shows the restriction only for CNC device, the restrictions for machine tool are not included. Reading this manual as well as the one provided by machine tool builder before programming enables better understanding to the restriction.

1.3 General Program Structure

A program contains main program and subprogram. Usually, the CNC system performs according to main program, unless there is a subprogram call in the main program. The main program will be executed again after a returning command is performed. The sequence is shown in Fig. 1-4.

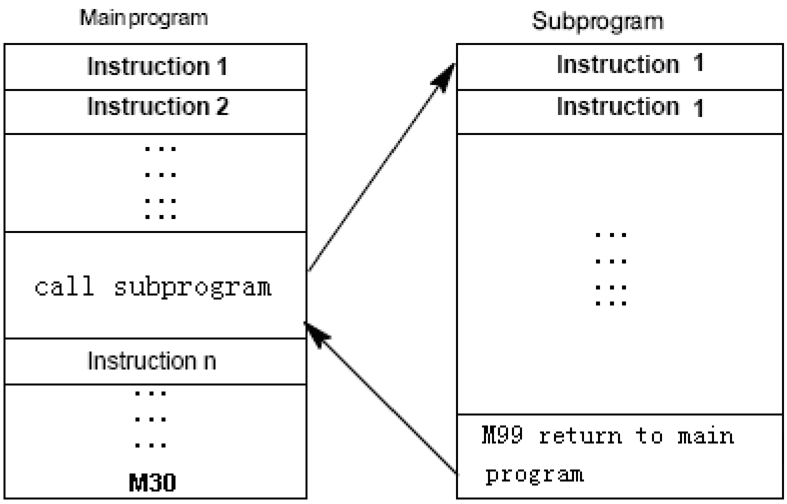


Fig. 1-4 Program run sequence

The structure of a main program is consistent with that of the subprogram.

If a program contains a fixed sequence and frequently repeated pattern, such a sequence or pattern can be stored as subprogram in memory to simplify the program. A subprogram can be called in auto mode by command M98. A called subprogram can also call another subprogram. The subprogram calls can be nested up to four levels (shown in Fig. 1-5). The last block of the main program should be the return command M99 which enables the next subprogram to be executed. The program can be repeated when M99 is executed at the end of main program.

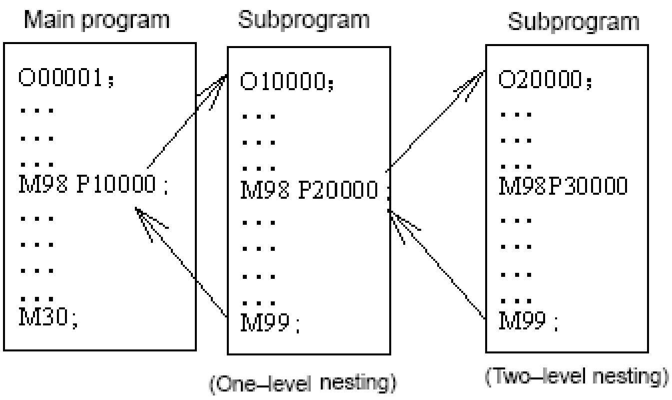


Fig. 1-5 Two-level nesting subprogram

A single call command can repetitively and continually call a subprogram up to 999 times.

1.3.1 Subprogram Writing and Call

1.3.1.1 Subprogram Writing

Write a subprogram as following format:

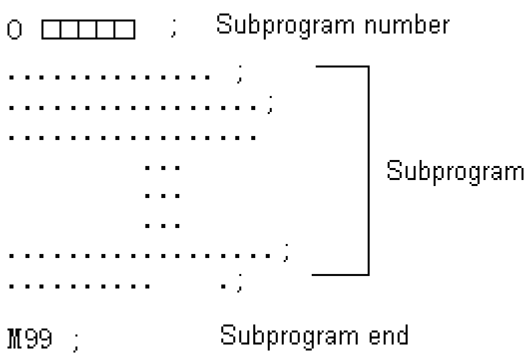
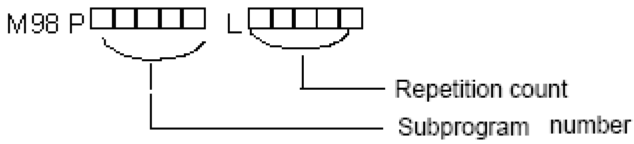


Fig. 1-6

At the beginning of a subprogram, the address O and subprogram number is placed. The end of the subprogram is command M99 (writing format is shown as above).

A subprogram is called by a call command whose format is shown as follows:



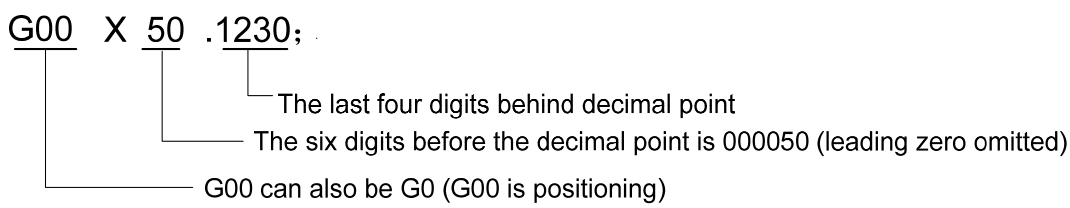
- If the repetition number is omitted, it is assumed to be 1.
  (e.g.) M98 P51002 ; （indicates that subprogram number 1002 is called continually 5 times）
- M98 P__ should not coexist with move command in the same block.
- The sequence of subprogram call in a subprogram is the same with that in main program.

Note: CNC enters the alarm state, if a subprogram number specified by address P can not be found.

1.3.2 Program Inputting Format

Words that constitute a block should be input with following format. When the format is variable, the word quantity in a block and the letter quantity in a word can be changed, it is convenient for programming.

E.g. with following command, the tool can be positioned to 50.123mm along X axis:



Note: If two commands are assigned by one address in the same block, the later command is valid in principle. No alarm will occur.

e.g.:
G00 G01 X100. Y200.;
G01 is valid, G00 is invalid.

1） G code is valid in the last command of the same block.

2） If there are R, I and K codes in the same arc command, R code is valid regardless of the sequence.

1.3.3 Program End

A Program starts from the program name and ends with command M02, M30 or M99. M02 and M30 enables the system enter into a reset state at the end of a program; the program can be repeated with command M99; if M99 is executed at the end of a subprogram, system returns to the program that call the subprogram. By using parameter N0:1803#5 and N0:1803#4 respectively, M30 and M02 determine whether the system returns to the beginning of the program or not.

Warning!

If the optional block skip switch on the machine operation panel is ON, the block with "/" will be skipped, e.g., command /M02; , /M30; , or /M99; do not indicate the program end.

1.3.4 Optional Block Skip （/）

When a slash followed by a number n(n=1~9) is specified at the head of a block, and optional block skip switch n on the machine operator panel is set to on, the information contained in the block for which /n corresponding to switch number n is specified is ignored in DNC operation or memory operation. When the optional block skip switch n is set to off, the information contained in the block specified by /n is valid. This means the operator can decide whether to skip blocks contain /n or not. Number 1 of /1 can be omitted. However, when more than two optional block skip switches are used in one block, number 1 of /1 cannot be omitted.

Example)	(incorrect)	(correct)
	//3 G00X10.0;	/1/3 G00X10.0;

When a program is loaded into memory, this function is ignored. The blocks containing /n are also stored into memory regardless of how the optional block skip is set. Programs held in memory can be output regardless of how the optional block skip is set.

The optional block skip is valid even when sequence number is being searched. Different machine tool has different amount of optional block skip switches (1-9), refer to the manual from machine tool builder for specific details.

Note:

1. The position of the slash

The slash (/) should be at the head of a block. Otherwise, information between the slash and

EOB code is ignored.

2. Disabling of optional block skip switch

When a block is read into buffer from memory or tape, the optional block skip operation is processed. After blocks read into a buffer, the already read blocks are not ignored even if the optional block skip switch is set to on.

3. TV and TH check

When the optional block skip switch is set to on, the TH and TV check is performed for the skipped blocks in the same way as when the optional block skip switch is off.

2 PROGRAMMING FUNDAMENTALS

2.1 Controlled Axes

Table 2-1

Item	GSK25i
Number of basic controlled axes	5 axes（X,Y,Z,4TH,5TH）
Simultaneously controlled axes（in total）	6 axes at most

2.2 Axis Name

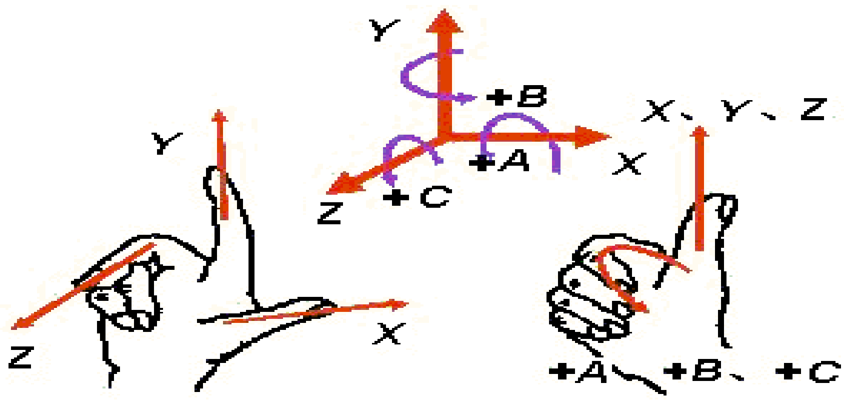
The names of 5 basic axes are always X,Y,Z, 4TH,5TH. Parameter No. 9101 sets the number of controlled axes and NO.1020 assigns name for each.

2.3 Coordinate system

2.3.1 Machine Coordinate System

The point that is specific to a machine and serves as the reference of the machine is referred to as the machine zero point. A machine tool builder sets a machine zero point for each machine. A coordinate system with a machine zero point set as its origin is referred to as a machine coordinate system. A machine coordinate system is set by performing manual reference position return after power-on. A machine coordinate system, once set, remains unchanged until the power is turned off, the system is restart or emergency stop is employed.

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



2.3.2 Reference Point

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

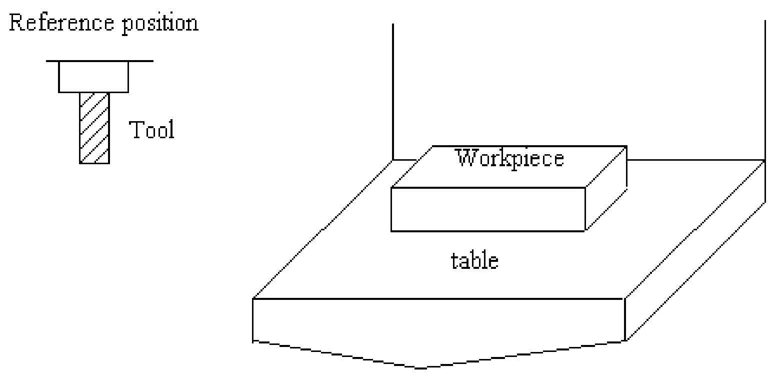


Fig.2-1

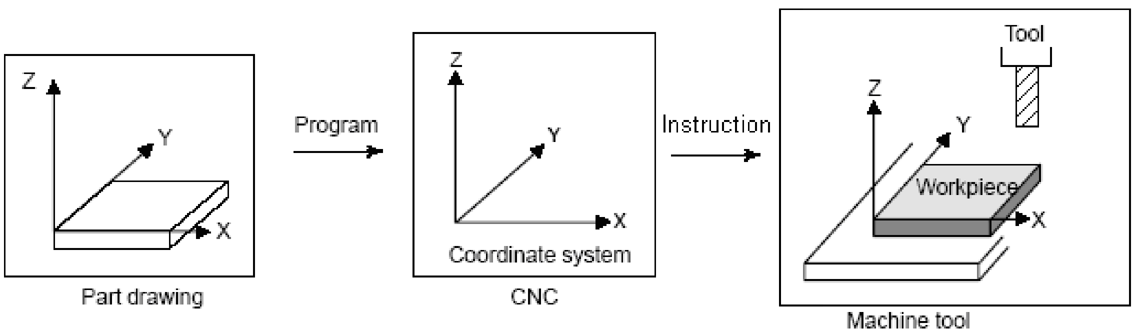
There are two methods to traverse the tool to reference point:

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

2. Auto reference point return

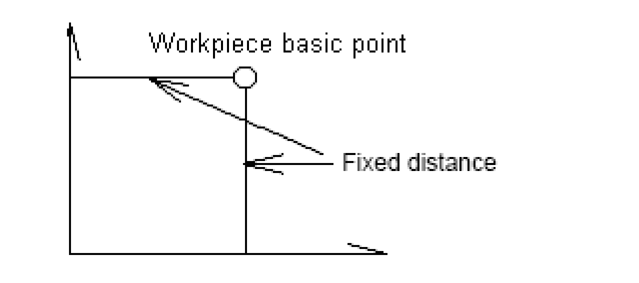
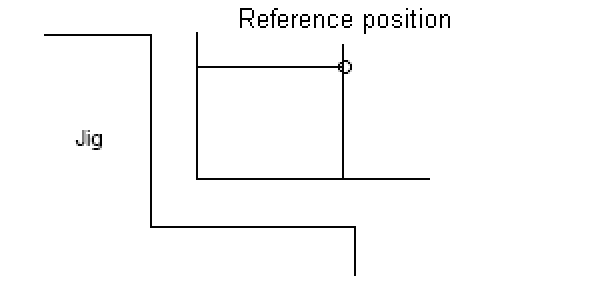
2.3.3 Workpiece Coordinate System

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



The tool machines workpiece into desired shape on the drawing according to program, so it is necessary to set relationship between machine coordinate system and workpiece coordinate system.

The method to determine the relationship between these two coordinate systems is called alignment. It can be done by different methods according to part shape or workpiece quantity.

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

Workpiece coordinate system can be set by one program and can be altered by moving its origin. There are two methods to set the workpiece coordinate system:

1. By G92, see 3.2.11 for details.

2. By G54 to G59, see 3.2.8 for details.

2.3.4 Maximum Stroke

Maximum stroke= least command increment×99999999

Table 2-2 Maximum strokes

Increment system	Maximum stroke
Metric machine system	±999999.9999mm ±999999.9999degree
Inch machine system	±99999.9999inch ±999999.9999degree

Note:

1. A command exceeding the maximum stroke cannot be specified.

2. The actual stroke depends on the machine tool.

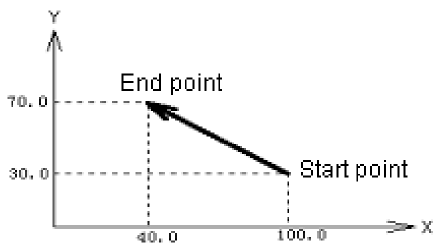


Fig.2-3

2.3.5 Absolute and Incremental Programming

There are two ways to command travels of the tool: the absolute command and the incremental command. In the absolute command, coordinate value of the end position is programmed; in the incremental command, move distance of the position itself is programmed.

Incremental value command is a method based on the move distance. Regardless of the coordinate, it just needs the move direction and distance of end position relative to the start position.

G90 and G91 are used to instruct absolute and incremental command.

In Fig. 2-3, moving from the start position to end position involves the following two commands (G90 and G91) respectively:

G90 G0 X40 Y70;

or G91 G0 X－60 Y40 ;

Either of two methods produces the same motion, and is available for operator to select.

Explanation:

➢ G90 and G91 are the modal value of the same group, i.e. G90 mode is defaulted before G91 is specified; G91 is valid till G90 is specified.

System parameter

Parameter N0:1801#3 determines whether G90 (when parameter is 0) or G91 (when parameter is 1) is employed as default mode.

2.4 Modal and Non-modal

Modal means that the number followed an address is valid till it is reset. Another function of modal is that after a word being set, it is not necessary to re-input the word when the same function is used.

➢ For example:

G0 X100 Y100; （positioning to X100 Y100）

X20 Y30; （positioning to X20 Y30, G0 is modal and can be omitted.）

G1 X50 Y50 F300（linear interpolation to X50 Y50, at a feedrate of 300mm/min G0→G1）

X100; （linear interpolation to X100 Y50, at a feedrate of 300mm/min, G1,Y50 and F300 are all modal and can be omitted.）

Initial mode is the default mode after power-on. See Table 3-1 for details.

➢ For example:

➢ O00001

➢ X100 Y100; （positioning to X100 Y100, G0 is initial mode）

➢ G1 X0 Y0 F100;（linear interpolation to X0 Y0, at a feedrate of 100mm/min, G98 is initial mode）

Non-mode means that the numbers after an address is valid in only in the current block and should be re-specified in next block. As G command of group 00 shown in table 3-1.

Table 2-3 describes the modal and non-modal of commands.

Table 2-3 modal and non-modal of commands

Modal	Modal G function	G commands are being executed till they are invalidated by another G commands.
	Modal M function	M commands is being executed till they are invalidated by another M commands.
Non-modal	Non-modal G function	Only valid in specified blocks and to be cancelled at the end of a program
	Non-modal M function	Only valid in the current block

2.5 Decimal Point Programming

Numerical value can be entered with a decimal point. A decimal point can be used when entering a distance, time, or speed. Decimal points can be specified with the following addresses:
X, Y, Z, A, B, C, I, J, K, R, P, Q, and F

Explanation:

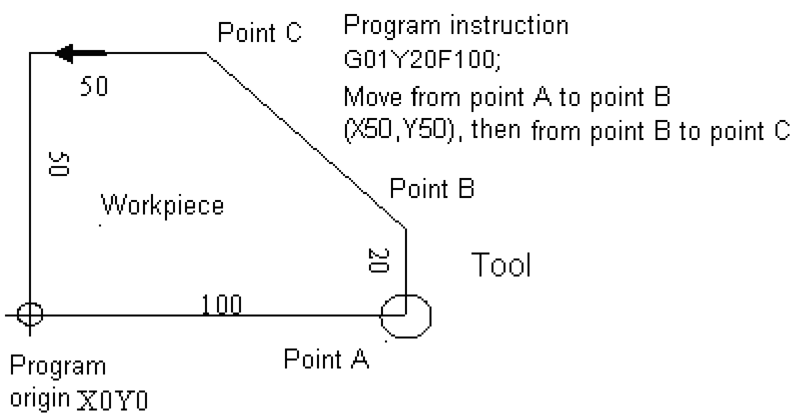
1. Parameter **N0:1800#5** determines the employment of decimal point programming. When **N0:1800#5=1**, the unit of programming value is mm, inch or degree; when **N0:1800#5=0**, the unit is the least movement unit, determining by parameter **N0:1000#1**.
2. Fractions less than the least input increment are truncated.

For example:
X9.87654; when the least input increment is 0.001mm, truncated to X 9.876.
when the least input increment is 0.0001mm, processed as X 9.8765.

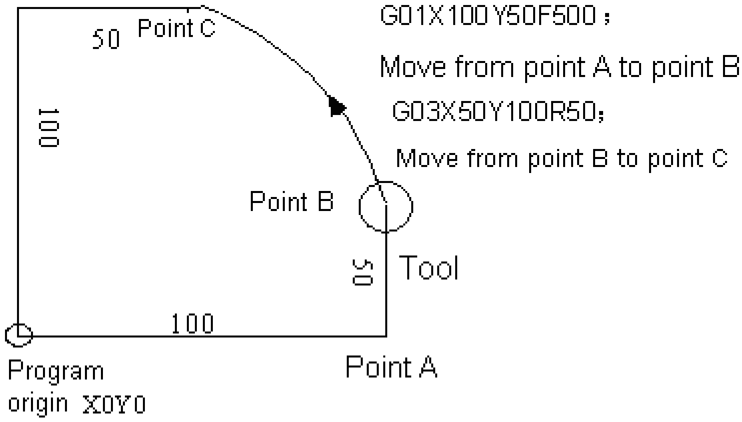
2.6 Basic Functions

2.6.1 Tool Movement along Workpiece Parts Figure—Interpolation

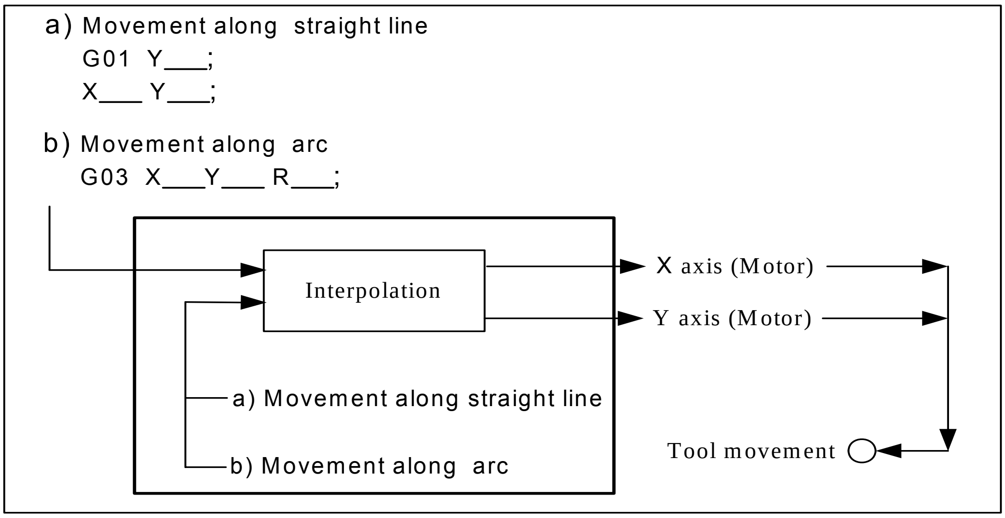
1）The tool moves along straight lines



2） The tool moves along arcs

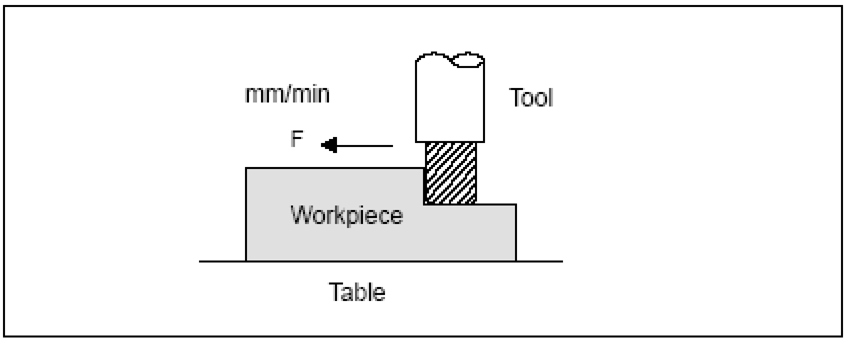


The function of moving the tool along straight lines and arcs is called the interpolation. Symbols of the programmed commands G01, G02…are called the preparatory function and specify the type of interpolation conducted in the control unit.



Note: Some machines move tables instead of tools but this manual assumes that tools are moved against workpiece. Refer to the actual move direction to avoid danger and damages.

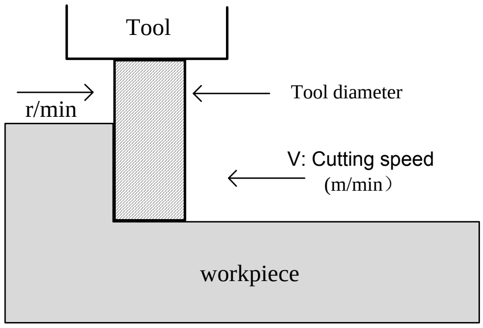
2.6.2 Feed—Feed Function



The function of specifying a feedrate is called feed function.

Feed is to move the tool with a specified rate. The feedrate is indicated by numeric command. For example, command F200 means the tool infeeds at a speed of 200mm/min.

2.6.3 Cutting Speed, Spindle Speed Function

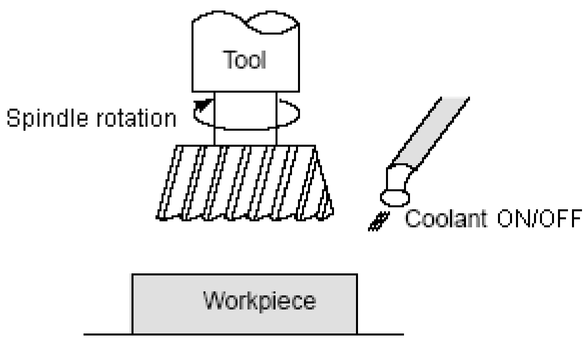


The speed of the tool with respect to the workpiece when the workpiece is cut is called the cutting speed. For CNC, it can be specified by the spindle speed RPM（r/min）.

For example, when a workpiece is machined with a tool 100mm in diameter at a cutting speed of 80m/min, the spindle speed is about 250r/min, which is obtained from N=1000V/πD. The command is S250.

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

2.6.4 Command for Machine Operations—Miscellaneous Function



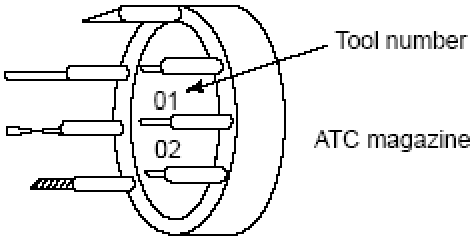
When machining is actually started, it's necessary to rotate the spindle, and feed coolant accordingly. Thus, the on-off switch for spindle motor and coolant valve should be controlled.

The function of specifying the on-off operations of the machine or program through NC system is called the miscellaneous function, which is specified by M mode.

For example, when M03 is specified, the spindle rotates clockwise at the specified speed. (Clockwise means operator views over the spindle along the negative direction of Z axis.)

2.6.5 Selection of Tool Used for Various Machining—Tool

When drilling, tapping, boring, milling or the like, is performed, it is necessary to select a suitable tool. When a number is assigned to each tool and the number is specified in the program. The corresponding tool is selected.

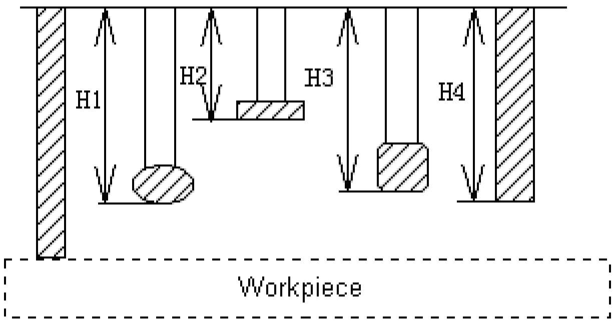


For example, when No. 01 is assigned to a drilling tool

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

2.6.6 Tool Figure and Tool Motion by Program

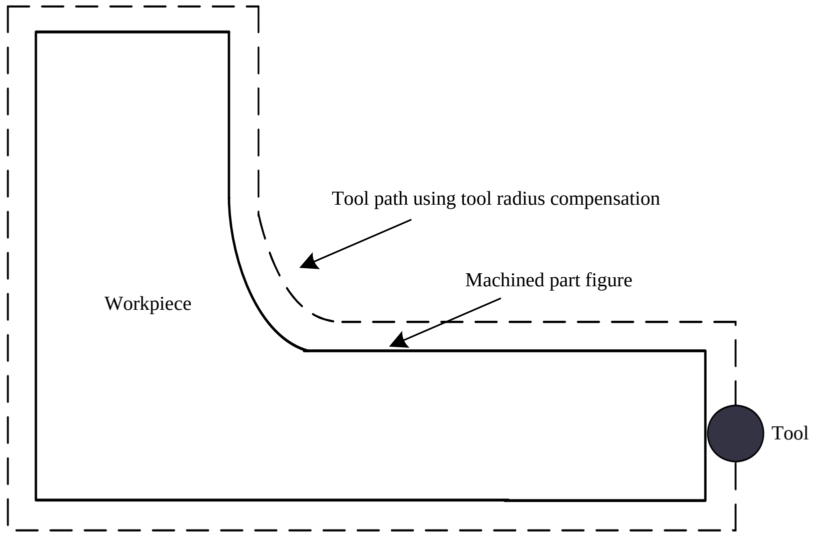
2.6.6.1 Tool Length Compensation



Usually, several tools are used for machining one workpiece. When a command is executed, such as G0Z0, the distance from tools and to workpiece may vary due to different tool lengths. However, it is very troublesome and error-prone to alter the program frequently.

Therefore, the lengths of tools are measured in advance, and their differences from that of the standard tool (usually is the first tool) are input into CNC. In this way, machining can be done without altering the program when tool is changed. The distance from tool end to the workpiece remains unchanged after Z axis positioning (such as G0Z0) is executed. This function is called tool length compensation function.

2.6.6.2 Cutter Compensation Function

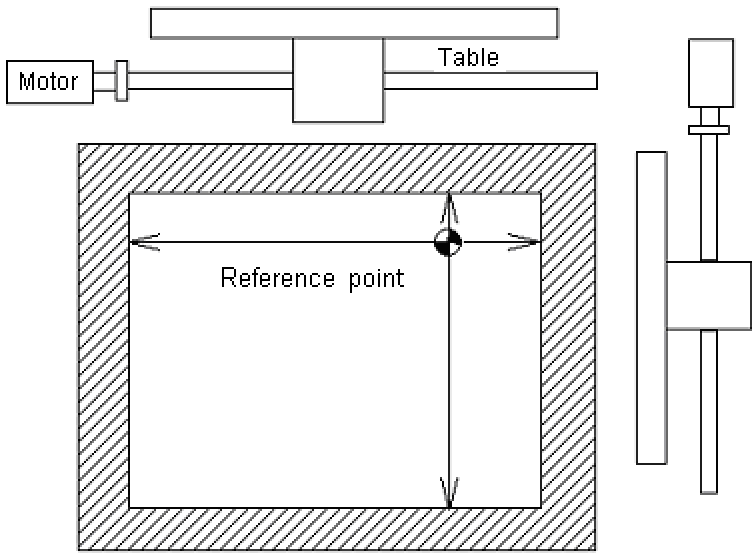


Because a tool has a radius, a workpiece will be overcut a cutter radius if the programmed path is followed. To simplify programming, the program can be run with a cutter radius deviated around the machined part figure. The path of intersection between lines and arcs is processed by system automatically.

Cutter diameters should be stored in the compensation list in advance, so that the cutter path may vary with different cutter compensation values. This function is called cutter compensation function.

2.6.6.3 Tool Movement Range—Stroke

A safe movement range can be set by parameters. Exceeding of the range leads to motion stop of all axes and an alarm will be issued in that case. This function is called stroke check, usually called soft restriction.



3 PREPARATORY FUNCTION G CODES

3.1 Types of G Codes

The number following address G determines the meaning of the command for the concerned block. G codes are divided into the following two types.

Table 3-1 types of G codes

Type	meaning
Non-modal G code	The G code is effective only in the block in which is specified
Modal G code	The G code is effective until another G code of the same group is specified.

(Example) G01 and G00 are modal G codes in group 01.

G01 X __ ;

Z ______ ; G01 is effective

X ______ ; G01 is effective

G00 Z____; G00 is effective

Table 3-2 G codes List

<table>
<tr><th>G code</th><th>Group</th><th colspan="2">Commands format</th><th>Functions</th></tr>
<tr><td>*G00</td><td rowspan="4">01</td><td colspan="2">G00 X_Y_Z_</td><td>Positioning (rapid traverse)</td></tr>
<tr><td>G01</td><td colspan="2">G01 X_Y_Z_F_</td><td>Linear interpolation (cutting feed)</td></tr>
<tr><td>G02</td><td colspan="2" rowspan="2">G02
G03 X_Y_ R_
I_J_ F_;</td><td>Circular interpolation (CW)</td></tr>
<tr><td>G03</td><td>Circular interpolation (CCW)</td></tr>
<tr><td>G04</td><td>00</td><td colspan="2">G04 P_ or G04 X_</td><td>Dwell, Exact stop</td></tr>
<tr><td>G10</td><td>00</td><td colspan="2">G10L_; N_P_R_</td><td>Programmable data input</td></tr>
<tr><td>*G11</td><td>00</td><td colspan="2">G11</td><td>Programmable data input mode cancel</td></tr>
<tr><td>*G15</td><td rowspan="2">17</td><td colspan="2">G15</td><td>Polar coordinate command cancel</td></tr>
<tr><td>G16</td><td colspan="2">G16</td><td>Polar coordinate command</td></tr>
<tr><td>*G17
G18
G19</td><td>02</td><td colspan="2">write followed other words and used in circular interpolation and cutter radius compensation</td><td>XY plane selection
ZX plane selection
YZ plane selection</td></tr>
<tr><td>G20</td><td rowspan="2">06</td><td colspan="2" rowspan="2">Input at the beginning of a block and before the coordinate system is set. Specified by an independent block.</td><td>Input in inch</td></tr>
<tr><td>*G21</td><td>Input in metric</td></tr>
<tr><td>G27</td><td rowspan="6">00</td><td>G27</td><td rowspan="5">X_Y_Z_</td><td>Reference point return check</td></tr>
<tr><td>G28</td><td>G28</td><td>Return to reference point</td></tr>
<tr><td>G29</td><td>G29</td><td>Return from the reference point</td></tr>
<tr><td>G30</td><td>G30Pn</td><td>2nd, 3rd,and 4th reference point return</td></tr>
<tr><td>G31</td><td>G31</td><td>Skip function</td></tr>
<tr><td>G39</td><td>G39</td><td>I_J_; I_J_; J_K_or G39</td><td>Corner offset circular interpolation</td></tr>
</table>

*G40	07	G17	G40 G41 G42	X_Y_	Cutter compensation cancel
G41		G18		X_Z_	Cutter compensation left
G42		G19		Y_Z_	Cutter compensation right
G43	08	G43		Z_	Tool length compensation + direction
G44		G44			Tool length compensation - direction
*G49		G49			Tool length compensation cancel
*G50	11	G51			Scaling cancel
G51		G51 X_ Y_ Z_ P_			Scaling
G53	00	Write in a program			Machine coordinate system selection
*G54	14	Write together with others, usually it is placed at the beginning of a program.			Workpiece coordinate system 1
G55					Workpiece coordinate system 2
G56					Workpiece coordinate system 3
G57					Workpiece coordinate system 4
G58					Workpiece coordinate system 5
G59					Workpiece coordinate system 6
G60	00	G60 X_ Y_ Z_ F_			Single direction positioning
G61	15	G61			Exact stop mode
G62		G62			Automatic corner override
G63		G63			Tapping mode
*G64		G64			Cutting mode
G65	00	G65 H_P# i Q# j R# k			Macro program command
G68	16	G68 X_ Y_ R_			Coordinate rotation
*G69		G69			Coordinate rotation cancel
G73	09	G73 X_Y_Z_R_Q_F_;			Peck drilling cycle
G74		G74 X_Y_Z_R_P_F_;			Counter tapping cycle
G76		G76 X_Y_Z_R_P_F_K_;			Fine boring cycle
*G80		Write together with others			Canned cycle cancel
G81		G81 X_Y_Z_R_F_;			Drilling cycle (spot drilling cycle)
G82		G82 X_Y_Z_R_P_F_;			Drilling cycle (stepped hole boring cycle)
G83		G83 X_Y_Z_R_Q_F;			Peck drilling cycle
G84		G84 X_Y_Z_R_P_F_;			Tapping cycle
G85		G85 X_Y_Z_R_F_;			Boring cycle
G86		G86 X_Y_Z_R_F_;			Drilling cycle
G87		G87 X_Y_Z_R_Q_P_F_;			Counter boring cycle
G88		G88 X_Y_Z_R_P_F_;			Boring cycle
G89		G89 X_Y_Z_R_P_F_;			Boring cycle
*G90	03	Write together with others			Absolute programming
G91					Incremental programming
G92	00	G92 X_Y_Z_			Workpiece coordinate system preset
*G94	05	G94			Feed per minute
G95		G95			Feed per rotation
G96	13	G96S_			Constant surface speed control

*G97		G97S_	Constant surface speed control cancel
*G98	10	Write together with others	Return to initial plane in canned cycle
G99			Return to R point in canned cycle
G110	09	X_ Y_ R_ Z_ I_ L_ W_ Q_ V_ D_ F_ K_	Circular groove inner rough milling (CCW)
G111		X_ Y_ R_ Z_ I_ L_ W_ Q_ V_ D_ F_ K_	Circular grove inner rough milling (CW)
G112		X_Y_ R_ Z_ I_ J_ D_ F_ K_	Circular inner finish milling cycle (CCW)
G113		X_Y_ R_ Z_ I_ J_ D_ F_ K_	Circular inner finish milling cycle (CW)
G114		X_Y_R_Z_I_J_L_W_Q_V_D_F_K_	Circular outer rough milling cycle (CCW)
G115		X_Y_R_Z_I_J_L_W_Q_V_D_F_K_	Circular outer rough milling cycle (CW)
G116		X_Y_ R_ Z_ I_ J_D_F_ K_	Circular outer finish milling cycle (CCW)
G117		X_Y_ R_ Z_ I_ J_D_F_ K_	Circular outer finish milling cycle (CW)
G130		X_Y_Z_R_I_J_L_W_Q_V_U_D_F_ K_	Rectangular groove rough milling (CCW)
G131		X_Y_Z_R_I_J_L_W_Q_V_U_D_F_ K_	Rectangular groove rough milling (CW)
G132		X_ Y_ R_ Z_ I_ J_ D_ L_ U_ F_ K_	Rectangular groove inner finish milling cycle (CCW)
G133		X_ Y_ R_ Z_ I_ J_ D_ L_ U_ F_ K_	Rectangular groove inner finish milling cycle (CW)
G134		X_Y_Z_R_I_J_L_W_Q_V_E_U_D_ F_K	Rectangular groove outer rough milling (CCW)
G135		X_Y_Z_R_I_J_L_W_Q_V_E_U_D_ F_K	Rectangular groove outer rough milling (CW)
G136		X_Y_R_Z_I_J_D_L_U_F_K_	Rectangular outer finish milling cycle (CCW)
G137		X_Y_R_Z_I_J_D_L_U_F_K_	Rectangular outer finish milling cycle (CW)
G120	00	X_Y_I_J_K_	Bolt hole circle (Canned Cycle)
G121		X_Y_I_J_K_	Line at angle (Canned Cycle)
G122		X_Y_I_J_P_K_	Arc (Canned Cycle)
G123		X_Y_I_P_J_K_	Grid (Canned Cycle)
G124		X_Y_R_Z_I_J_P_K_F_	Rectangular drilling (CW)
G125		X_Y_R_Z_I_J_P_K_F_	Rectangular drilling (CCW)
G126		X_Y_Z_I_J_L_F_	Round trip milling
G127		X_Y_Z_I_J_L_F_	Single trip milling

Note:

1. The G codes with mark * are the default G codes at power-on state.

2. G codes in 00 group are non-modal G codes except for G10 and G11.

3. If a G code not presented in G code list is used, or a G code has no corresponding function is specified, an alarm is output.

4. Multiple G codes can be specified in the same block if each G code belongs to a different group. If multiple G codes that belong to the same group are specified in the same block, only the last G code specified is valid.

5. If a G code belonging to group 01 is specified in a canned cycle, the canned cycle is cancelled and G80 is set. However, the G codes in group 01 are not affected by a G code specifying a canned cycle.

6. G codes are indicated by group according to their types.

3.2 Simple G Code

3.2.1 Positioning (G00)

Function : The G00 command moves a tool to the position in the workpiece system specified with an absolute or an incremental command at a rapid traverse rate.

Format: G00 IP_

IP_:For absolute command, the coordinate of an end position, and for an incremental command, the distance the tool moves.

Either of the following tool paths can be selected according to **N0:1200#1** (see Fig. 3-1)

1. Linear interpolation positioning: the tool path is the same as in linear interpolation (G01). The tool is positioned within the shortest possible time at a speed that is not more than the rapid traverse rate for each axis.
2. Non-linear interpolation positioning: the tool is positioned with the rapid traverse rate for each axis separately. The tool path is normally not straight.

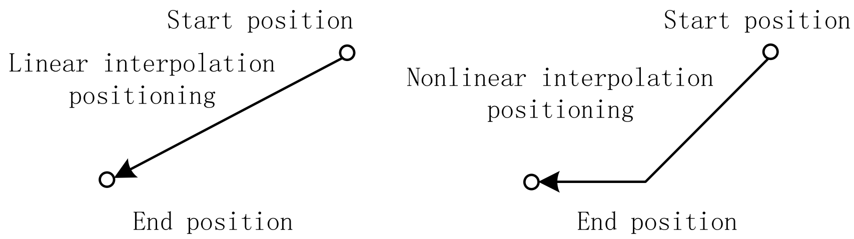


Fig. 3-1

Explanation:

1. G00 rapid traverse rate is set by parameter P1126, and the current tool move mode is changed into G00 mode. By changing parameter P1801#0, the default mode after power-on can be set as G00 (parameter value is 0) or G01 (parameter value is 0).

2. The tool does not move until a positioning parameter is specified. The system only changes tool move mode for G00.

3. G00 is identical with G0.

Restriction:

1. The rapid traverse rate cannot be specified in the address F. If a feedrate is specified in G0 command, it is used as the cutting feedrate that followed. For example:

   G0 X0 Y10 F800; Feeding at a rate set by system parameter

   G1 X20 Y50; at the rate set by F800

The following keys on the operation panel are used to adjust rapid feedrate, see Fig 3-2, involving such overrides as F0, 25, 50, 100%; The feedrate corresponding to F0 is set by parameter P1231, and it applies to all axes.

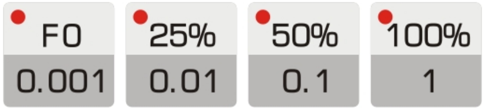


Fig.3-2 Rapid feedrate keys

3.2.2 Linear Interpolation G01

Function: Tool moves linearly to a specified position at the feedrate set by F.

Format : G01 IP_ F_

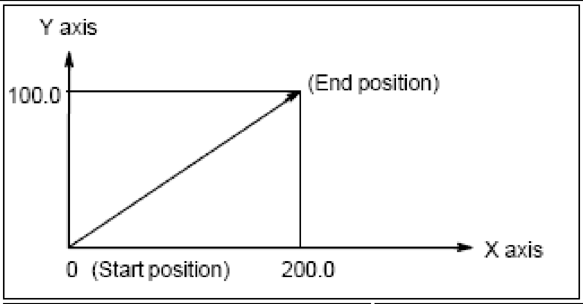
IP_:For absolute command, the coordinate of an end position, and for an incremental command, the distance the tool moves.

F_:Speed of tool feed (feedrate)

Explanation :

The feedrate should be specified in F and it is effective until a new value is specified. The feedrate commanded by the F code is measured along the linear interpolation path. If the F code is not commanded, the feedrate is regarded as zero.

Example (see Fig. 3-3)



G01 X200 Y100 F200 ;

Note: the feedrate of each axis direction is as follows:

G01 Xα Yβ ZγFf ;

Feedrate of X axis direction	Feedrate of Y axis direction	Feedrate of Z axis direction
$F_X = \frac{\alpha}{L} \times f$	$F_Y = \frac{\beta}{L} \times f$	$F_Z = \frac{\gamma}{L} \times f$

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

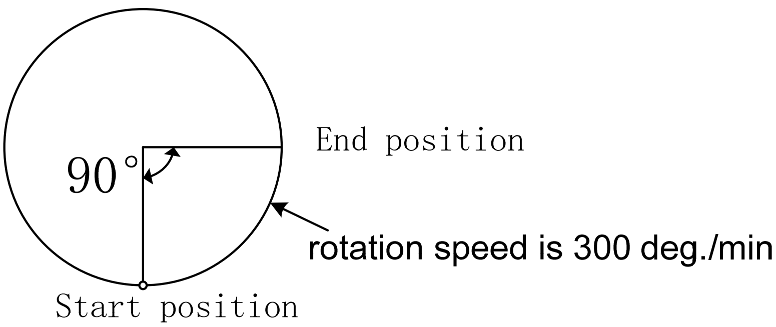
Fig. 3-3

Note:

1. The ceiling limits of cutting feedrate F for each axis can be set by parameter P1125. If the actual cutting feedrate (feedrate after override is used) exceeds the ceiling limit, the later will be adopted as feedrate (Unit mm/min). The ceiling limit of multi-axes resultant cutting feedrate can be set by parameter P1124. If the actual cutting feedrate (feedrate after override is used) exceeds the ceiling limit, the later will be adopted as feedrate (Unit mm/min).

2. The tool does not move when a position parameter followed G01 is not specified, and the current tool move mode is changed into G00 mode. By changing parameter P1801#0, the default mode after power-on can be set as G00 (parameter value is 0) or G01 (parameter value is 0).

3. When the linear interpolation (rotation axes A,B or C) involves over 4-axes, the unit of cutting feedrate changes from degree to inch (or mm), and the cutting feedrate in Cartesian coordinate system is set to be equal to the feedrate specified by F code. The feedrate of rotation axes is calculated by the formula in Fig. 3-3, the unit changed into deg./min.



Example: G91 G01 B90.0 F300;

Example: G91 G01 X20.0 B40.0 F300.0;

When the unit of cutting feedrate of B axis changed from degree to mm or inch, the calculation formula of processing time is as follows:

$$\frac{\sqrt{20^2+40^2}}{300}=0.014907 \quad \text{(min)}$$

The feedrate of B axis is:

$$\frac{40}{0.14907}=268.3 \quad \text{(deg/ min)}$$

3.2.3 Circular Interpolation (Helical Interpolation) G02/G03

3.2.3.1 Circular Interpolation G02/G03

Format: The command below will move a tool along a circular arc.

Arc in the X—Y plane

G17 { G02 / G03 } X——Y—— { R—— / I——J—— } F——;

Arc in the Z——X plane

G18 { G02 / G03 } X——Z—— { R—— / I——K—— } F——;

Arc in the Y——Z plane

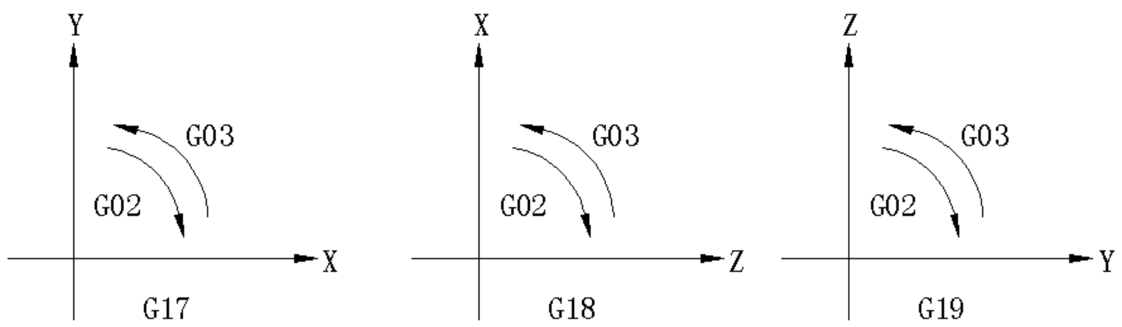
G19 { G02 / G03 } Y——Z—— { R—— / J——K—— } F——;

	Item		Command	Description
1	Plane selection		G17	Arc on plane XY
			G18	Arc on plane ZX
			G19	Arc on plane YZ
2	Rotation direction		G02	CW
			G03	CCW
3	End point	G90 mode	2 axes of X, Y, Z axes	End point of workpiece coordinate system
		G91 mode	2 axes of X, Y, Z axes	Distance from start point to end point
4	Distance from start point to end point		2 axes of I, J, K	Distance from start point to end point
	Arc radius		R	Arc radius

As an initial code, G17 is effective after power-on.

Explanation:

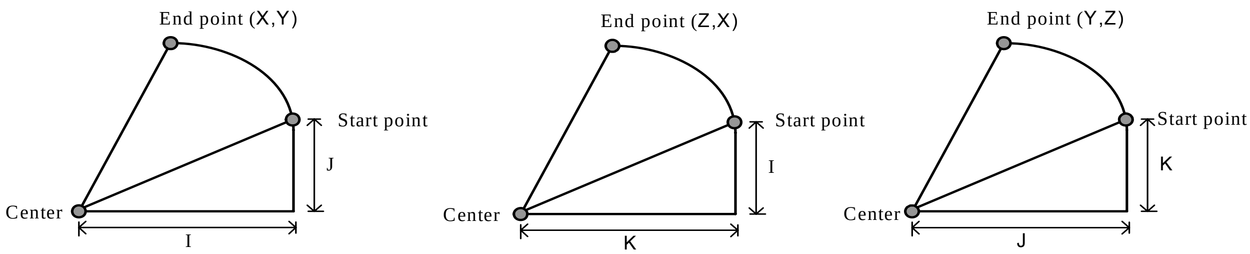
"Clockwise" （G02） and "counterclockwise"（G03）on the XY plane (ZPXP plane or YPZP plane) are defined when the XY plane is viewed in the positive-to negative direction of ZP axis (ZP axis or XP axis respectively) in the Cartesian coordinate system. See the figure below.



The end point of an arc is specified by address ZP, YP or ZP, and is expressed as an absolute or incremental value according to G90 or G91. For the incremental value, the distance of the end point which is viewed from the start point of the arc is specified.

The arc center is specified by address I,J and K for the XP, YP, and ZP axes, respectively. The numerical value following I,J, or K, however, is a vector component in which the arc center is seen from the start point, and is always specified as an incremental value irrespective of G90 and G91, as shown below.

I,J and K must be signed according to the direction (positive or negative).



I0, J0 and K0 can be omitted. When XP, YP AND ZP are omitted (the end point is the same as the start point) and the center is specified with I,J and K, a 360°arc (circle) is specified.

G02 I_; command for a circle.

If the difference between the radius at the start point and that at the end point exceeds the permitted value in a parameter P1810, and alarm occurs.

The distance between an arc and the center of a circle that contains the arc can be specified using the radius, R of the circle instead of I, J and K. In this case, one arc is less than 180°, and the other is more than 180° are considered. When an arc exceeding 180° is commanded, the radius must be specified with a negative value. If XP, YP and ZP are all omitted, if the end point is located at the same position as the start point and when R is used, an arc of 0° is programmed.

G02 R; （the cutter does not move）

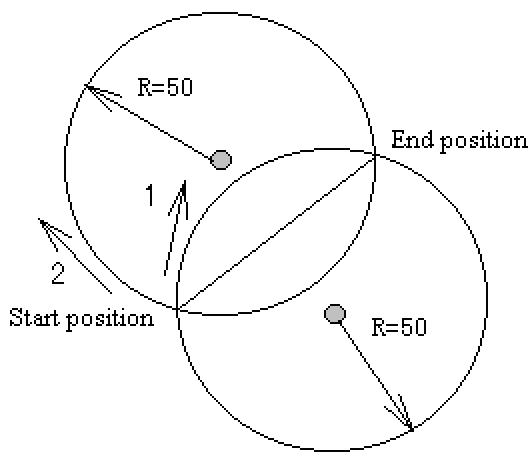
Example:

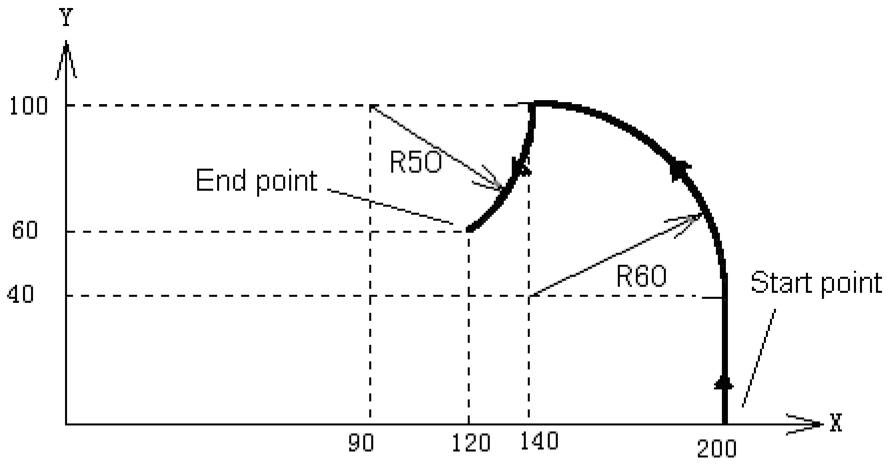
1． For arc less than 180°

G02 X6.0 Y2.0 R5.0;

2． For arc more than 180°

G02 X6.0 Y2.0 R-5.0;





a）Absolute programming

（I） G92 X200.0 Y40.0 Z0;

（II） G90 G03 X140.0 Y100.0 I-60.0 F300.0;

（III） G02 X120.0 Y60.0 I-50.0;

（IV） G92 X200.0 Y40.0 Z0;

（V） G90 G03 X140.0 Y100.0 R60.0 F300;

（VI） G01 X120.0 Y60.0 R50.0;

b）Incremental programming

（I） G91 G03 X-60.0 Y60.0 I-60.0 F300;

（VII） G02 X-20.0 Y-40.0 I-50.0;

（II） G91 G03 X-60.0 Y60.0 R60.0 F300;

（VIII） G02 X-20.0 Y-40.0 R50.0;

The feedrate in circular interpolation is equal to the feedrate specified by the F code, and the feedrate along the arc (the tangential feedrate of the arc) is controlled to be the specified feedrate is±2% or less. However, this feedrate is measured along the arc after the cutter compensation is applied.

If I,J and R addresses are specified simultaneously, the arc specified by address R takes precedence and the other are ignored.

When an arc having a center angle approaching 180° is specified, the calculated center coordinates may contain an error. In such a case, specify the center of the arc with I, J and K.

3.2.3.2 Helical Interpolation G02/G03

Format:G02/G03

Arc of XY plane

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

Arc of ZX plane

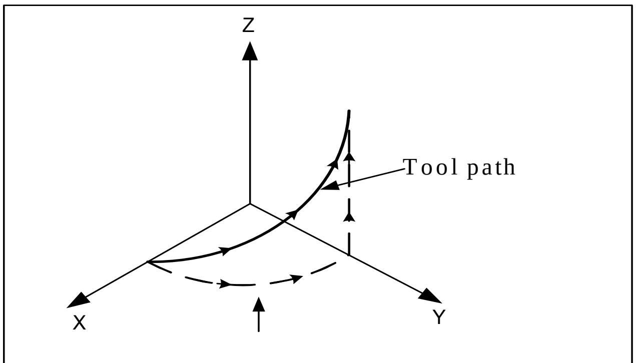
$$G18\begin{Bmatrix}G02\\G03\end{Bmatrix}\ X_p\_\ Y_p\_\ Z_p\_\ \begin{Bmatrix}I\_\ K\_\\R\_\end{Bmatrix}\ F\_$$

Arc of YZ plane

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

Function: Helical interpolation which moved helically is enabled by specifying up to two other axes which move synchronously with the circular interpolation by circular commands.

Explanations:



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

The command method is to simply or secondary add a move command axis which is not circular interpolation axes. An F command specifies a feedrate along arc. Therefore, the feedrate of the linear axis is as follows:

$$F\times\frac{\text{length of linear axis}}{\text{length of circular arc}}$$

Determine the feedrate so the linear axis feedrate does not exceed any of the various limit values.

Restrictions: Cutter compensation is applied only for a circular arc

Tool offset and tool length compensation cannot be used in a block in which a helical interpolation is commanded.

3.2.4 Cylindrical Interpolation (G07.1)

The amount of travel of a rotary axis specified by an angle is once converted into a distance of a linear axis along the outer surface so that linear interpolation or circular interpolation can be performed with another axis. After interpolation, such a distance is converted back into the amount of travel of a rotary axis.

Cylindrical interpolation allows the cylinder flank to be developed for programming. So programs such as a cylindrical cam grooving program can be easily created.

Format:

G07.1 IP1: Starts the cylindrical interpolation mode (enables cylindrical interpolation) ⋮ G07.1 IP0: The cylindrical interpolation mode is cancelled. IP: An address for the rotation axis r: The radius of the cylinder Specify G07.1 IPr: and G07.1 IP0: in separate blocks, G107 can be used instead of G07.1.

Explanation:

1) Plane selection (G17, G18, G19)

Use parameter No. 1024 to specify whether the rotary axis is X axis, Y axis, Z axis or an axis parallel to one of these axes. Specify the G code to select a plane for which the rotary axis is the specified linear axis.

For example, when the rotary axis is an axis parallel to the X axis, G17 must specify and Xp-Yp plane, which is a plane defined by the rotary axis and the Y axis or an axis parallel to the Y axis. Only one rotary axis can be set for cylindrical interpolation.

2) Feedrate

A feedrate specified in the cylindrical interpolation mode is a speed on the developed cylindrical surface.

3) Circular interpolation

In the cylindrical interpolation mode, circular interpolation can be performed between the rotary axis and another linear axis. Radius R is used in commands in the same way as circular interpolation. The unit for a radius is not degrees but mm (for metric input) or inch (for inch input)

Example: for circular interpolation between the Z axis and C axis, 5 is to be set (axis parallel to X axis) for the C axis of parameter No.1024, the command is:

```
G18 Z_C_;
G02(G03) Z_C_R_;
```

6 (axis parallel to Y axis) can be specified instead for the C axis of parameter No.1021, the command is:

```
G19 C_Z_;
G02(G03) Z_C_R_;
```

4) Cutter compensation

To execute cutter compensation in cylindrical interpolation mode, an ongoing cutter compensation should be cancelled before entering into cylindrical interpolation mode, then a cutter compensation can be started and terminated within the cylindrical interpolation mode.

5) Cylindrical interpolation accuracy

In the cylindrical interpolation mode, the amount of travel of a rotary axis specified by an angle is internally converted to a distance of a linear axis on the outer surface so that linear interpolation or circular interpolation can be performed with another axis. After interpolation, such a distance is converted back to an angle. For this conversion, the amount of travel is rounded to a least input increment. Therefore, when the radius of a cylinder is small, the actual amount of travel can differ from a specified amount of travel. Note, however, that such an error is not accumulative. If manual operation is performed in the cylindrical interpolation mode with manual absolute on, an error can occur for the reason described above.

The actual amount of travel= $\left[\frac{\text{MOTION } REV}{2\times 2\pi R}\times\left[\text{Specified value}\times\frac{2\times 2\pi R}{\text{MOTION } REV}\right]\right]$

MOTION REV: The amount of travel per rotation of the rotary axis (360°).

Limitations:

1) Specify arc radius in cylindrical interpolation mode

Address I, J or K cannot be used to specify the arc radius in cylindrical interpolation mode.

2) Circular interpolation and cutter compensation

If the cylindrical interpolation mode is started when the cutter compensation is already applied, even circular interpolation cannot be performed correctly in such case.

3) Positioning

In cylindrical interpolation mode, positioning cannot be specified (including cycles that generate rapid traverse, such as G28, G53, G73, G74, G76, G89~G89). Cylindrical interpolation mode should be cancelled before positioning. Cylindrical interpolation (G07.0) cannot be executed in positioning mode (G00).

4) Coordinate system setting

In cylindrical interpolation mode, workpiece coordinate system (G92, G54~G59) and local coordinate system (G52) cannot be specified.

5) Cylindrical interpolation mode setting

In the cylindrical interpolation mode, the cylindrical interpolation mode cannot be reset. The cylindrical interpolation mode should be cancelled before resetting the cylindrical interpolation.

6) Tool offset

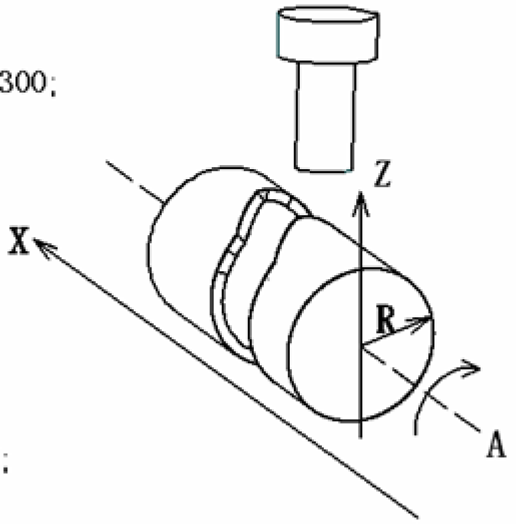
Tool offset should be specified prior to cylindrical interpolation mode setting. Besides, it cannot be changed in cylindrical interpolation mode.

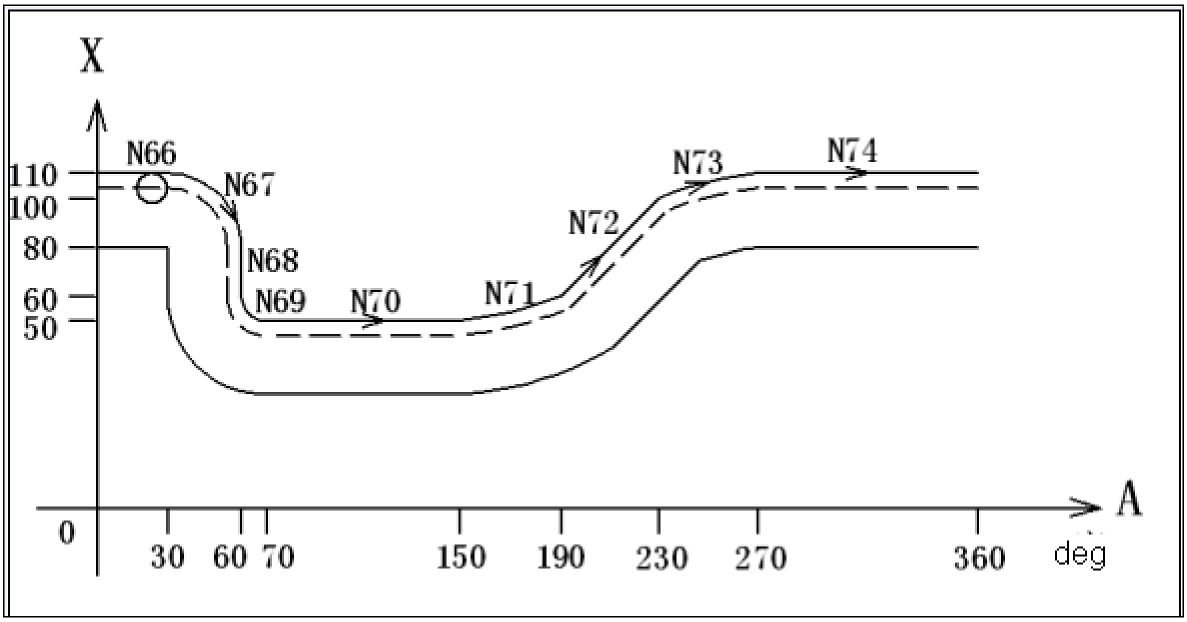
7) Index table indexing function

Cylindrical interpolation cannot be specified when index table indexing function is being used.

Example of a Cylindrical Interpolation Program

```
N61 G91 G28 Z0;
N62 G17 G90 A0 Z150;
N63 G1 Z57.299 X110 F300;
N64 G07.1 A57299;
N65 G41 D1;
N66 A30;
N67 G02 X80 A60 R30;
N68 G01 X60;
N69 G03 X50 A70 R10;
N70 G01 A150;
N71 G03 X60 A190 R75;
N72 G01 X100 A230;
N73 G02 X120 A270 R75;
N74 G1 A360;
N75 G0 Z150;
N76 G40;
N77 G07.1 A0;
N78 M30;
```



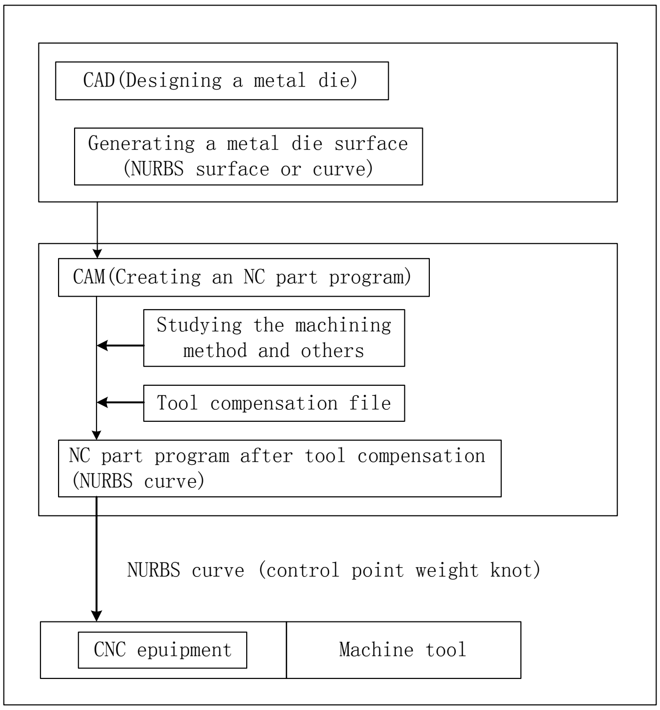


3.2.5 NURBS Interpolation

In the CAD drawings for mould designs such as car and plane, NURBS (Non Uniform Rational B-Spline) is widely used as a method to describe sculptured surfaces and curves of the metal dies. The NURBS interpolation can directly specify the expression of NURBS curves to CNC device. This eliminates the need for approximating the NURBS curve with minute line segments, which brings the following advantages:

1. Eliminate the errors due to linear approximation of designed NURBS curves.
2. Shorten the part programs
3. Avoid the “break” between blocks during the execution of small blocks at high-speed.
4. There is no need to perform high-speed transfer from the main unit to the CNC.

Based on the NURBS expression output by CAD, the NURBS interpolation function creates NURBS curves after compensating for the tool holder length, tool diameter and other tool elements. The NURBS curve is programmed by using 3 parameters: control point, weight, knot.



NC part program for mould machining based on NURBS curve command

NURBS interpolation should be specified in high-precision contouring control mode (between G05 P10000~G05 P0). The NURBS interpolation is executed while the acceleration/deceleration of all axes is controlled within the permitted range to prevent collision.

Format:

G05 P10000; (High precision contour control mode ON)

…

```
G06.2 [P_] K_X_Y_Z_ [R][F];
           K_X_Y_Z_ [R];
           K_X_Y_Z_ [R];
           K_X_Y_Z_ [R];
           ...
           K_;
```

G01...

...

G05 P0; High precision contour control mode OFF

G06.2: NURBS interpolation mode

P_: The rank of NURBS curve

X_Y_Z_: Control point

R_: Weight

K_: Knot

F_: Feedrate

Note:

1. NURBS interpolation mode

NURBS interpolation mode is selected when G06.2 is programmed. G06.2 is the modal G code in 01 group. Therefore, specifying the G codes in 01 group other than G06.2 (such as G00, G01, G02, G03) can end the NURBS interpolation. NURBS interpolation should be completed before the high-precision contouring control mode is OFF.

2. NURBS rank

A rank of NURBS can be specified by address P. the rank setting, if any, must be specified in the first block. If the rank setting is omitted, a rank of four (degree of three) is assumed for NURBS. The valid data range for P is 2-4. the P value have the following meanings:

P2 : NURBS having a rank of two (degree of one)

P3 : NURBS having a rank of three (degree of two)

P4 : NURBS having a rank of four (degree of three) (default value)

The rank referred here is the “k” in the definition expression of NURBS curve described latter. For example, the rank having a rank of four has a degree of three.

3. Weight

The weight of a control point programmed in a single block can be defined. When the weight setting is omitted, the weight value is assumed to be 1.0.

4. Knot

The number of specified knots equals the number of control points plus the rank value. In the blocks specifying the first to last control points, each control point and a knot are specified in an identical block. After these blocks, as many blocks (including only a knot) as the rank value are specified. The NURBS curve programmed for NURBS interpolation must start from the first control point and end at the last control point. The first k knots (where k is the rank) must have the same values as the last k knots (multiple knots). If the absolute coordinate of the start point of NURBS interpolation do not match the position of the first control point, P/S alarm is issued. (To specify incremental values, G06.2 X0 Y0 Z0 K_ must be programmed).

5. NUBRS curve

Describe every variable in following formats:

k : rank

Pi : control point

Wi : weight

Xi : knot (Xi ≤ Xi+1)

Knot vector [X0, X1,..., Xm] (m = n+ k)

t : spline parameter

Spline basis function N based on de Boor-Cox recursive formula can be expressed as follows:

$$N_{i,1}(t) = \begin{cases} 1(x_i < t < 1x_{i+1}) \\ 0(t < x_i, x_{i+1} < t) \end{cases}$$

$$N_{i,k}(t) = \frac{(t\text{-}x_i)N_{i,k\text{-}1}(t)}{x_{i+k\text{-}1} - x_i} + \frac{(x_{i+k}\text{-}t)N_{i+1,k\text{-}1}(t)}{x_{i+k} - x_{i+1}}$$

The NURBS curve P (t) of interpolation can be expressed as follows:

$$P(t)=\frac{\sum_{i=0}^{m}N_{i,k}(t)w_i p_i}{\sum_{i=0}^{m}N_{i,k}(t)w_i}$$

$$(x_0 \le t \le x_m)$$

6. Reset

Resetting in the process of NURBS interpolation results in the clear state, meanwhile the NURBS interpolation mode is cancelled.

Limitations

1. Controlled axes

Up to 3 axes can perform NURBS interpolation. All the axes that perform NURBS interpolation should be specified in the first block (G06.2 block). When there is no command in the first block, the axes should be specified in the second block, otherwise, program error occurs and an alarm is issued.

2. Commands in NURBS interpolation mode

In NURBS interpolation mode, the G codes, feedrate, MSTB codes and other interpolation mode cannot be specified.

3. Manual intervention

When manual intervention is performed in manual absolute mode, P/S alarm is generated.

4. Cutter compensation

It cannot be used together with the cutter compensation. Please cancel the cutter compensation before specifying NURBS interpolation.

5. Control point

As the first control point (coordinate value of G06.2 block) specifies the start point of NURBS curve, it should be identical with the end point of the previous block, otherwise, a program error alarm will be generated.

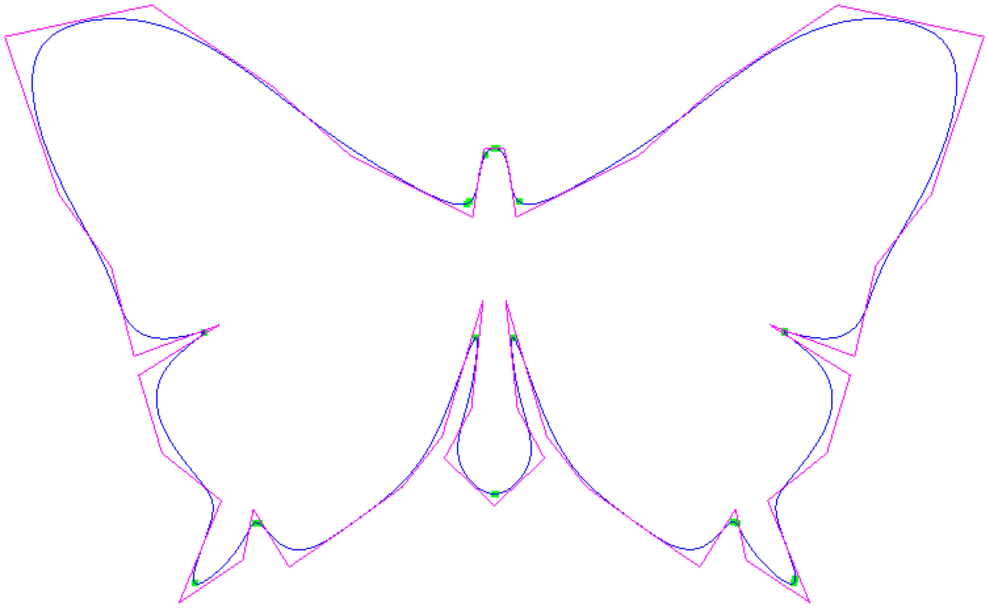
Example

<NURBS interpolation program>

```
G54G40G17G49G90G21;
G91G28Z0.0;
G0G90X0.0Y0.0;
X-33.22Y-15.695S0M03;
G43Z10.H00;
Z-21.F5000;
G0X54.493Y52.139Z0.000;
G6.2P4K0.000000X54.493Y52.139Z0.000R1.000;
K0.000000X55.507Y52.139Z0.000R1.000;
K0.000000X56.082Y49.615Z0.000R1.000;
```

K0.000000X56.780Y44.971Z0.000R1.200;
K0.008286X69.575Y51.358Z0.000R1.000;
K0.014978X77.786Y58.573Z0.000R1.000;
K0.036118X90.526Y67.081Z0.000R1.000;
K0.085467X105.973Y63.801Z0.000R1.000;
K0.129349X100.400Y47.326Z0.000R1.000;
K0.150871X94.567Y39.913Z0.000R1.000;
K0.193075X92.369Y30.485Z0.000R1.000;
K0.227259X83.440Y33.757Z0.000R2.000;
K0.243467X91.892Y28.509Z0.000R1.000;
K0.256080X89.444Y20.393Z0.000R1.000;
K0.269242X83.218Y15.446Z0.000R5.000;
K0.288858X87.621Y4.830Z0.000R3.000;
K0.316987X80.945Y9.267Z0.000R1.000;
K0.331643X79.834Y14.535Z0.000R1.100;
K0.348163X76.074Y8.522Z0.000R1.000;
K0.355261X70.183Y12.550Z0.000R1.000;
K0.364853X64.171Y16.865Z0.000R1.000;
K0.383666X59.993Y22.122Z0.000R1.000;
K0.400499X55.680Y36.359Z0.000R1.000;
K0.426851X56.925Y24.995Z0.000R1.000;
K0.451038X59.765Y19.828Z0.000R1.000;
K0.465994X54.493Y14.940Z0.000R1.000;
K0.489084X49.220Y19.828Z0.000R1.000;
K0.499973X52.060Y24.994Z0.000R1.000;
K0.510862X53.305Y36.359Z0.000R1.000;
K0.533954X48.992Y22.122Z0.000R1.000;
K0.548910X44.814Y16.865Z0.000R1.000;
K0.573096X38.802Y12.551Z0.000R1.000;
K0.599447X32.911Y8.521Z0.000R1.000;
K0.616280X29.152Y14.535Z0.000R1.100;
K0.635094X28.040Y9.267Z0.000R1.000;
K0.644687X21.364Y4.830Z0.000R3.000;
K0.651784X25.768Y15.447Z0.000R5.000;
K0.668304X19.539Y20.391Z0.000R1.000;
K0.682958X17.097Y28.512Z0.000R1.000;
K0.711087X25.537Y33.750Z0.000R2.000;
K0.730703X16.602Y30.496Z0.000R1.000;
K0.743865X14.199Y39.803Z0.000R1.000;
K0.756479X8.668Y47.408Z0.000R1.000;
K0.772923X3.000Y63.794Z0.000R1.000;
K0.806926X18.465Y67.084Z0.000R1.000;
K0.849130X31.197Y58.572Z0.000R1.000;
K0.870652X39.411Y51.358Z0.000R1.000;
K0.914534X52.204Y44.971Z0.000R1.200;
K0.963883X52.904Y49.614Z0.000R1.000;
K0.985023X53.478Y52.139Z0.000R1.000;

```
K0.991714X54.492Y52.139Z0.000R1.000;
K1.000000;
K1.000000;
K1.000000;
K1.000000;
G0Z10.;
G0Z50.;
G91G28Z0.0;
G91G28Y0.0 ;
```



3.2.6 Dwell (G04)

Format:G04 X_ or P_
X_: specify a time (decimal point permitted)
P_: specify a time (decimal point not permitted)

Function: By specifying a dwell, the execution of the next block is delayed by the specified time.

Explanations:

1. As G04 is non-modal command, it is only effective in the current block.
1. With G04, the execution of the next block is delayed by the specified time. In addition, a dwell can be specified to make an exact check in the cutting mode (G64 mode).
3. When neither P or X is specified, exact stop is performed.
4. When P and X is specified simultaneously, P is effective.
5. Alarm No.18 will occur if the value specified by P and X is negative.

Table 3-3 command value range of the dwell time (command by X)

Command value range	Dwell time unit
0.001～99999.999	s

Table 3-4 command value range of the dwell time (command by P)

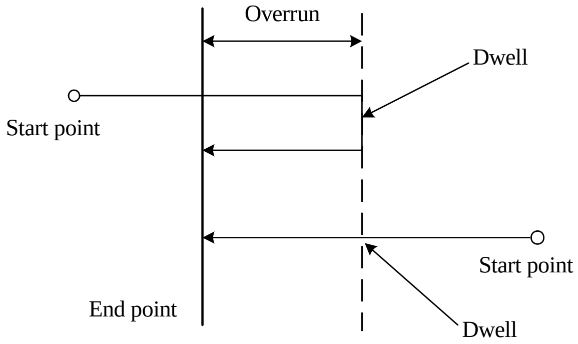
Command value range	Dwell time unit
1～99999999	0.001 s

E.g.: dwell for 3.8s

G04 X3.8 or G04 P3800;

3.2.7 Single Direction Positioning（G60）

Format:G60 X_ Y_ Z_



Function: For accurate positioning without play of the machine (backlash), final positioning from one direction by G60 is available.

Explanations:

Parameters X,Y and Z indicate that, for an absolute command, the coordinate of an end position, and for an incremental command, the distance the tool moves.

In the figure above, the marked overrun can be set by parameter P1880, and the defaulted dwell time is 1s. The positioning direction can be set by the positive or negative value of overrun. See system parameters for details.

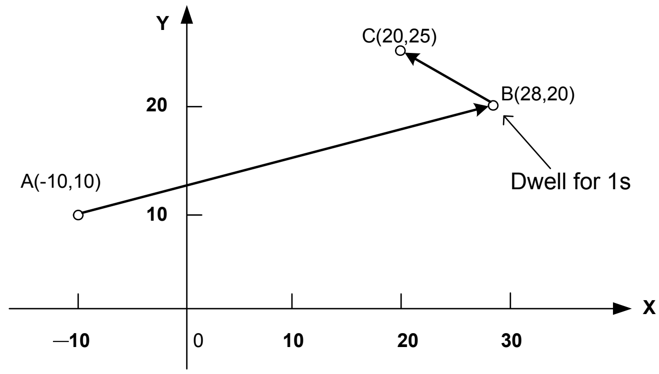
System parameters:

P1880	Overrun on X axis（Unit:mm）
P1880	Overrun on Y axis（Unit:mm）
P1880	Overrun on Z axis（Unit:mm）
P1880	Overrun on 4th axis（Unit:mm）
P1880	Overrun on 5th axis（Unit:mm）

E.g.:

G90 G00 X-10 Y10;

G60 X20 Y25; (1)



In the case that the parameter P1880 is set to（-8, 5, 0, 0, 0）, for statement (1),the tool path is AB→1s dwell→BC.

when non-modal G60 commands are used	when modal G60 command is used
⋮ G90； G60 X0Y0； G60 X100； Single direction positioning G60 Y100； G04 X10； G00 X0Y0； ⋮	⋮ G90G60； Single direction positioning mode X0Y0； X100； Single direction positioning Y100； G04 X10； G00 X0Y0；Single direction positioning mode cancel ⋮

G60 is non-modal code, when parameter No.1870#0（MDL）is set to 1, G60 can be used as mode code in group 01. This setting can eliminate specifying a G60 command for every block. Other specifications are the same as those for a one-shot G60 command. When a one-shot G code is specified in the single direction positioning mode, the one-shot G command is effective like G codes in group 01.

Notes:

1. During canned cycle for drilling, no single direction positioning is performed in Z axis.
2. No single direction positioning is performed in an axis for which not overrun has been set by parameter.
3. When the move distance 0 is commanded, the single direction positioning is not performed.
4. Direction set by parameter is ineffective in case of the mirror image function is used.
5. Single direction positioning does not apply to canned cycle G76 and G87.

3.2.8 Skip Function G31

3.2.8.1 Normal Skip

Format : G31 X_Y_Z_

Function:Linear interpolation can be commanded by specifying axial move following the G31 command, like G01. If and external skip signal is input during the execution of this command, execution of the command is interrupted and the next block is executed. The skip function is used when the end of machining is not programmed but specified with a signal form the machine, for example, in grinding. It is also used for measuring the dimensions of a workpiece.

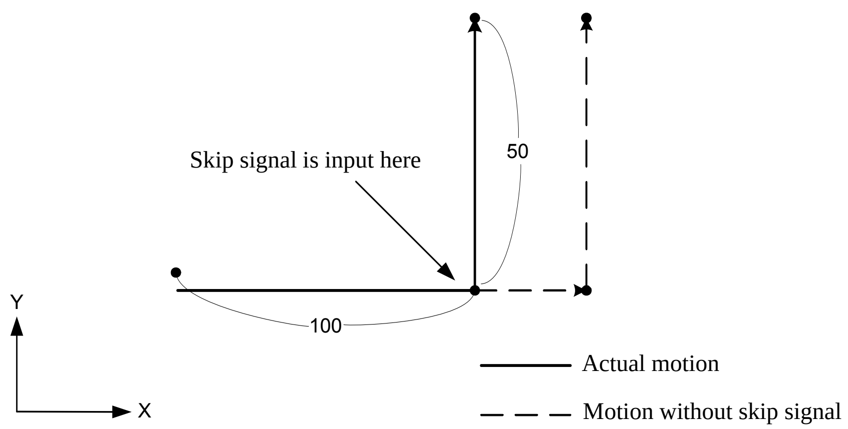
Explanations:

1. As a non-modal code, G31 is effective only in specified blocks.
2. If G31 command is issued while cutter compensation is applied, an alarm is displayed. Cancel the cutter compensation before the command is specified.
3. Disable feedrate override, dry run, and automatic acceleration/deceleration (however, these become available by setting the parameter No.1940#7(SKF) to 1). When the feedrate per minute is specified, allowing for an error in the position of the tool when a skip signal is input. These functions are enabled when the feedrate per rotation is specified.

The motion after skip signal is input depends on the next block (absolute or incremental command).

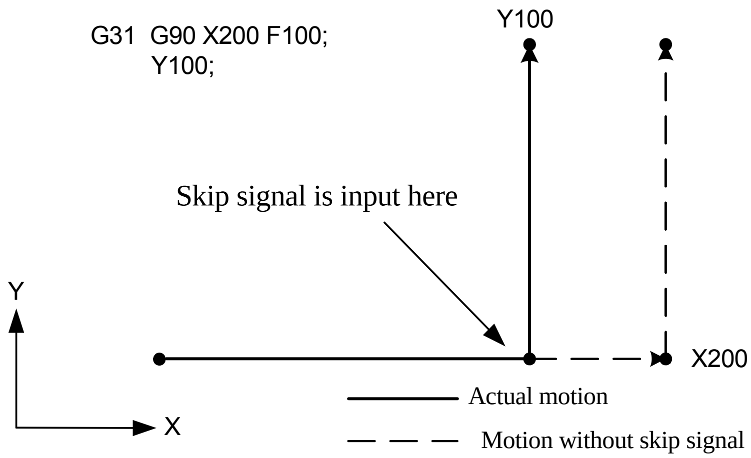
1). The next block is an incremental command.
Incremental movement is performed from the break point

E.g.:G31 G91 X100.0;
Y50.0;



2). The next block is an absolute command for 1 axis
The commanded axis moves to the specified position, the unspecified one stay at the position where skip signal in input.

e.g.:G31 G90 X200.0;
Y100.0;

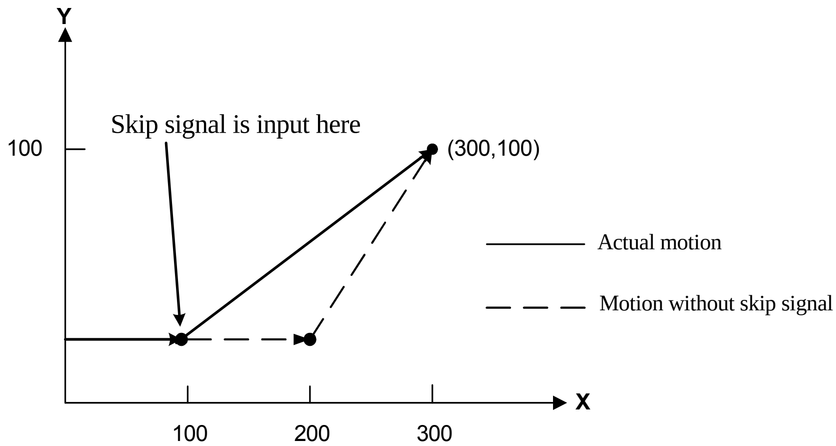


3). The next block is an absolute command for 2 axis

Axes move to the specified position wherever a skip signal is input.

e.g.:G31 G90 X200.0;

X300.0 Y100.0;



Feedrate specified in G31 block can be set with the following two methods:

a） To specified by F code (specified before or in G31 block.)

b）To set by parameter

The coordinate value is stored in the system variables #5061～#5065 of custom macro when the skip signal is turned on, therefore, the skip function can be used in macro program.

#5061………………………coordinate value the 1st axis

#5062………………………coordinate value the 2nd axis

#5063………………………coordinate value the 3rd axis

#5064………………………coordinate value the 4th axis

#5065………………………coordinate value the 5th axis

Skip function can be used when the movement amount is not defined; therefore it applies to the following situations:

a） Feeding in grinding with standard size.

b）Measuring when tool touching the sensor.

Notes:

1. If the feedrate specified by G31 is related to that set by parameter, the relevance is effective even during dry run.

2. If the feedrate specified by G31 is related to that set by parameter, auto-acceleration/deceleration is ineffective, which will improve the measure precision when skip function is applied.

3.2.8.2 High Speed Skip Signal

Format:

G31 X_Y_Z_;

G31:non-modal code（only effective in the specified blocks）

The skip function operates based on a high-speed skip signal (connected directly to the NC; not via the PLC) instead of an ordinary skip signal. In this case, up to eight signals can be input. Delay and error of skip signal input is 0-2 ms at the NC side (not considering those at the PLC side). This high-speed skip signal input function keeps this value to 0.1 mc or less, thus allowing high precision measurement.

For details, refer to relevant manual supplied by machine tool builder.

3.2.9 System Parameter Online Modification (G10)

Function: It can modify parameters and screw-pitch error compensation data. This function is applied for setting screw-pitch error compensation data when attachments is changed, the maximum cutting feedrate or time constant are changed to meet the changing machining condition.

Format:

G10L50; parameter entry mode setting
N_R_; For parameters other than axis type
N_P_R_; For axis type parameters
⋮
⋮
G11; parameter entry mode cancel

Command meaning:

N_ ; Parameter No. (4 digits) or compensation position No. for pitch errors compensation +10, 000（5 digits）

R_; Parameter setting value（leading zero can be omitted）.

P_; Axis No.1~5（Used for entering axis type parameters）

Explanations:

1．Do not use a decimal point in a value set in a parameter（R_）. A decimal point cannot be used in a custom macro variable for （R_）either.

2．Axis No. (P): Specify an axis number from 1 to 5 for an axis type parameter. The control axes are numbered in the order in which they are displayed on the CNC screen.

For example, specified P2 for the control axis which is displayed second.

Warning:

1. Do not fail to perform reference point return manually after changing the pitch error compensation data or backlash compensation data, otherwise, the machine position will deviate from the correct position.
2. The canned cycle mode should be cancelled before inputting parameters, otherwise, malfunction will occur.

Notes:

1．Other NC statements (except for explanatory note) cannot be specified in parameter input mode.

2．G10 block should be commanded alone or else alarm is raised. Do remember to cancel the parameter input mode after executing G10, lest program mistake occurs.

3．The parameter value modified with G10 should be within the range of system parameter, otherwise an alarm will be issued.

4．The canned cycle mode should be cancelled before executing G10, otherwise, alarm will be given.

Examples:

1．Set bit 7（LTM）of parameter P2501

G10L50; Parameter input mode

N2501 R 10000000; LTM setting

G11; Parameter input mode cancel

2．Modify the value of Z axis (3rd axis) and A axis (4th axis) in parameter No.1082 of axis type (set the coordinate value of positive direction boundary in memory stroke check 2).

G10L50; Parameter input mode

N1082 P3 R4500; Modify Z axis

N1082 P4 R120; Modify A axis

G11; Parameter input mode cancel

3.2.10 Workpiece Coordinate System G54～G59

Format:G54～G59

Function:

When the carriages are amounted in different positions on a machine tool, different workpiece coordinate systems are needed. In this case, 6 workpiece coordinate systems set in advanced are available by 6 G codes (G54～G59). Programs are executed in the selected coordinate system whose position depends on the distance from reference point (a fixed point on machine tool) to the coordinate origin (workpiece origin offset value). See Fig. 3-4:

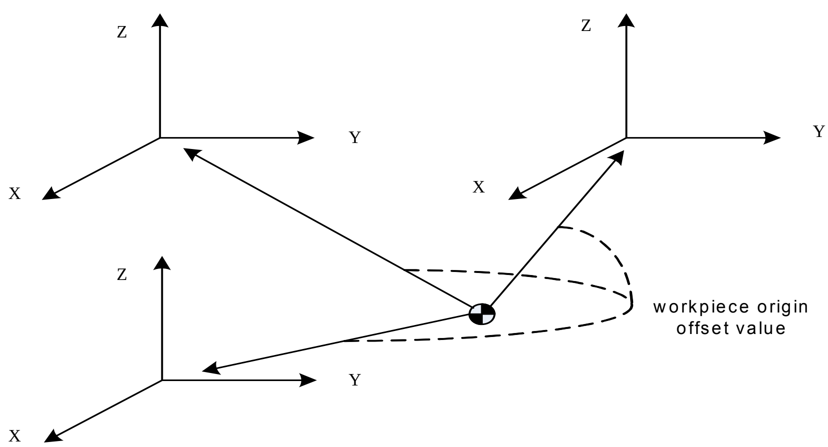


Fig. 3-4

Do not set a coordinate system with G92 command when the said coordinate systems are in use, for G92 will replace the set coordinate system. Usually, G92 and G54～G59 are not used simultaneously.

Note: For coordinate system set by G54～G59, there is no need to set an auto-coordinate system, because when returning to the 1st reference point after power-on, a coordinate system will generate by G54 automatically.

Explanations:

1. No command parameter.
2. Up to 6 workpiece coordinate systems can be set in system, each one can be selected by its corresponding commands (G54~G59).

G54 ---------------- Workpiece coordinate system 1
G55 ---------------- Workpiece coordinate system 2
G56 ---------------- Workpiece coordinate system 3
G57 ---------------- Workpiece coordinate system 4
G58 ---------------- Workpiece coordinate system 5
G59 ---------------- Workpiece coordinate system 6

3. When a different workpiece coordinate system is called, the commanded axis moves to a position in the new workpiece coordinate system; for axis not commanded, it moves to the corresponding position in the new workpiece coordinate system and the actual position of the machine does not change.

For example:

The corresponding machine coordinate for G54 coordinate system origin is（20, 20, 20）.

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

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

Program	Absolute coordinate	Machine coordinate
G0 G54 X50 Y50 Z50	50, 50, 50	70, 70, 70
G55 X100 Y100	100, 100, 30	130, 130, 60
X120 Z80	120, 100, 80	150, 130, 110

4. External workpiece zero point offset value or workpiece zero point offset value can be changed by G10, which is shown as follows:

By command G10 L2 Pp X_Y_Z_

P=0 : External workpiece zero point offset value

P=1 to 6: Workpiece zero offset value corresponds to workpiece coordinate system 1 to 6

X_Y_Z_ : For absolute instruction（G90）, workpiece zero point offset for each axis

For incremental instruction（G91）, value to be added to the set workpiece zero point offset for each axis(the result of addition becomes the new work piece zero point offset).

By G10 command, each workpiece coordinate system can be changed separately.

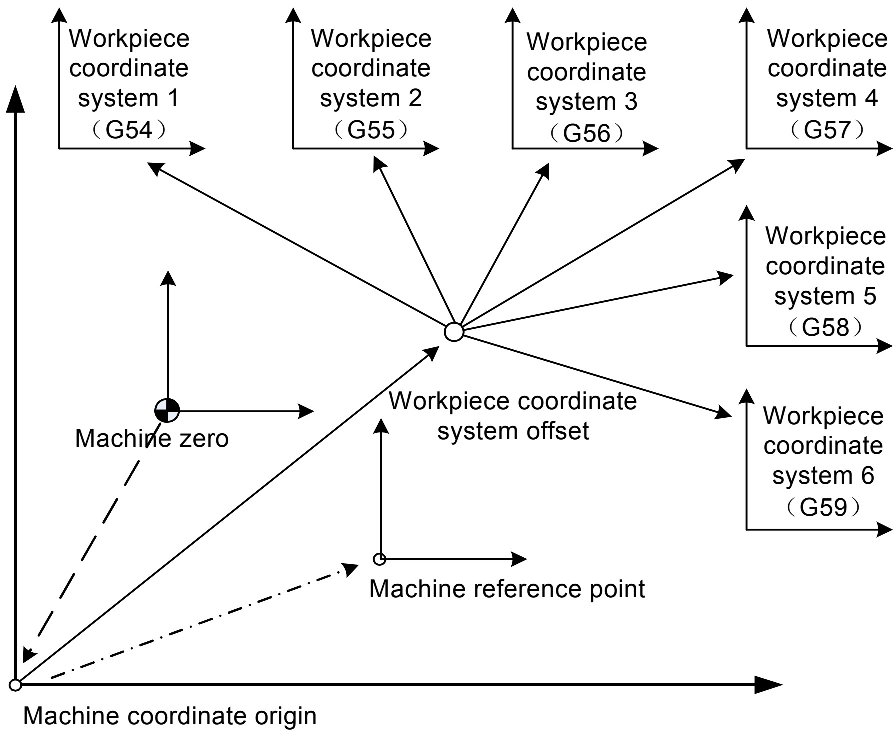


Fig. 3-5

As shown in Fig. 3-5, the machine returns to machine zero by manual zero return function after power-on. The machine coordinate system is set up based on this machine zero, thus machine reference point to be generated and workpiece coordinate system to be defined. The corresponding values of offset data parameter P1040～1046 in workpiece coordinate system indicate the whole offset amount of the 6 workpiece coordinate systems. The 6 workpiece coordinate system origins can be specified by inputting coordinate offset in MDI mode or setting by parameter P1040～1046. These 6 workpiece coordinate systems are set up based on the distances from machine zero to their coordinate system origins.

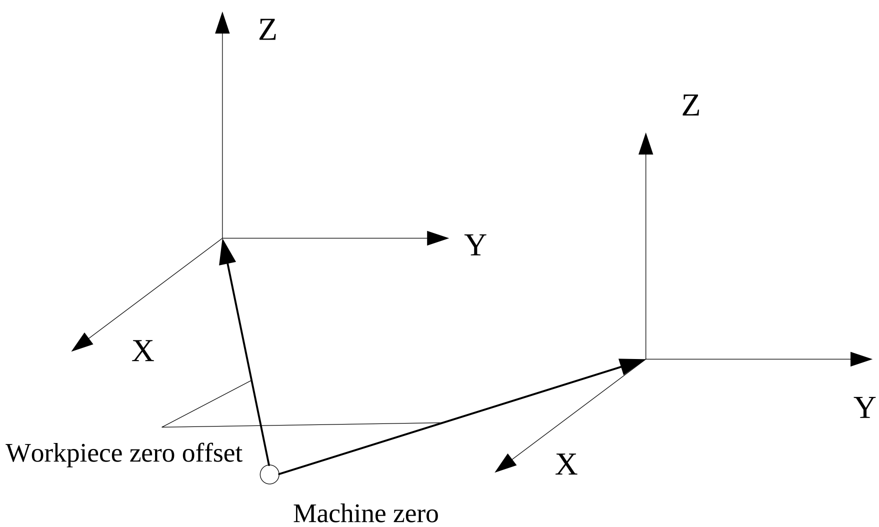


Fig. 3-6

Example:

N10 G55 G90 G00 X100 Y20;

N20 G56 X80.5 Z25.5;

In the example above, when N10 block is being executed, positioning to workpiece coordinate system 2 (X=100, Y=20) is performed. When N20 block is being executed, the absolute coordinate value is changed automatically to the coordinate value （X=80.5, Z=25.5）in workpiece coordinate system 3 at rapid traverse rate.

3.2.11 Optional Angle Chamfering and Corner Rounding

Chamfering and corner rounding blocks can be inserted automatically between the following:

（1）Between linear interpolation and linear interpolation blocks;

（2）Between linear interpolation and circular interpolation blocks;

（3）Between circular interpolation and linear interpolation blocks;

（4）Between circular interpolation and circular interpolation blocks.

Format: L_:chamfering

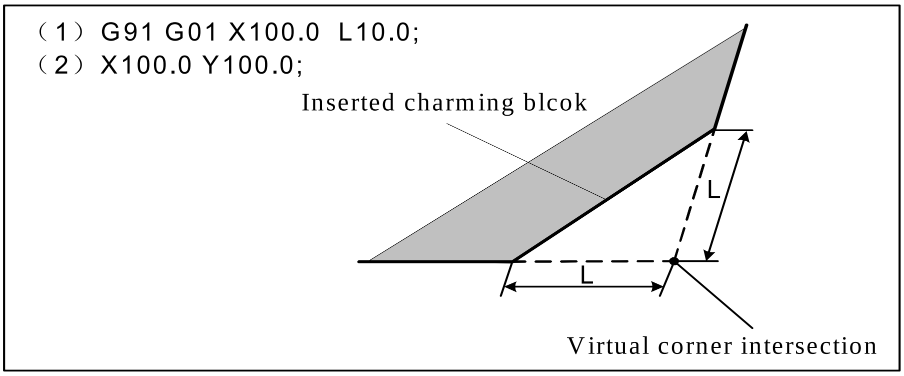
R_:Corner R

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

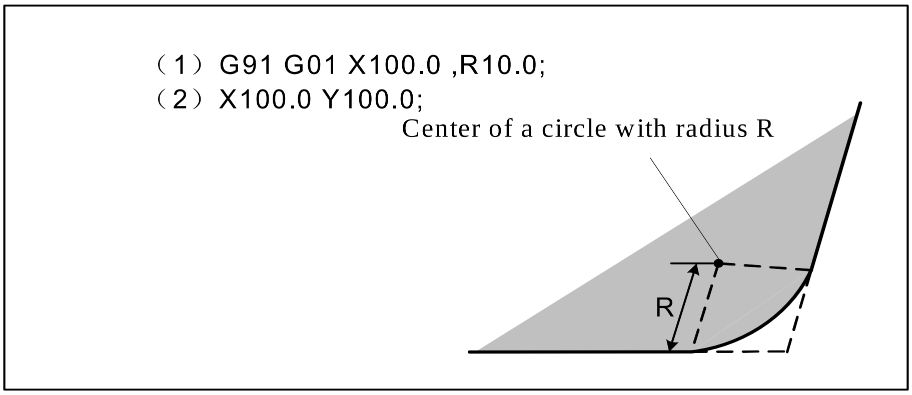
Explanations:

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

2. Chamfering: After L, specify the distance from the virtual corner position to the start and end positions. The virtual corner position is the corner position that would exist if chamfering were not performed, which is shown as below:

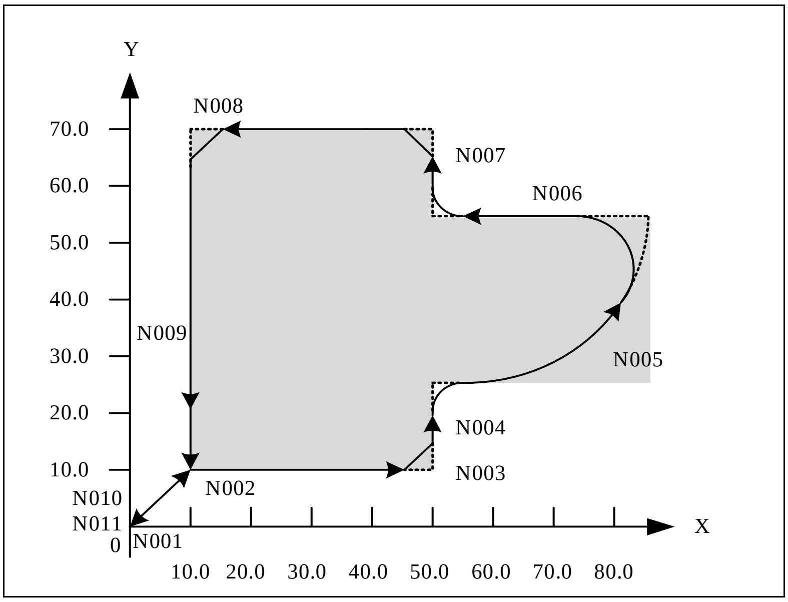


3. Corner R: After R, specify the radius for corner rounding, which is shown as below:



Examples:

N001 G92 G90 X0 Y0;
N002 G00 X10.0 Y10.0;
N003 G01 X50.0 F10.0,L5.0;
N004 Y25.0,R8.0;
N005 G03 X80.0 Y50.0 R30.0,R8.0;
N006 G01 X50.0,R8.0;
N007 Y70.0,L5.0;
N008 X10.0,L5.0;
N009 Y10.0;
N010 G00 X0 Y0;
N011 M0;



Restrictions：

1．Chamfering and corner rounding can be performed only in the plane specified by plane selection (G17,G18 or G19）. These functions can’t be performed for parallel axes.

2. A block specifying chamfering or corner rounding must be followed by a block that specifies a move command using linear interpolation or circular interpolation (G02 or G03). If the next block does not contain these specifications, the alarm is then issued.

3．A chamfering or corner rounding block can be inserted only for move commands which are performed in the same plane. In a block that comes immediately after plane switching, neither chamfering nor corner rounding can be specified.

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

5. In a block that comes immediately after the coordinate system is changed (G92, or G52 to G59) or a return to the reference position (G28 to G30) is specified, neither chamfering nor corner rounding can be specified.

6. Corner rounding can not be specified in a threading block.

7. DNC operation can not be applied to optional-angle chamfering or corner rounding.

8. The following G codes can not be used in a block that specifies chamfering or corner rounding. They can not be used between chamfering and corner rounding blocks, either.

1) G codes of group 00 (except for G04)

2) G68 of group 16

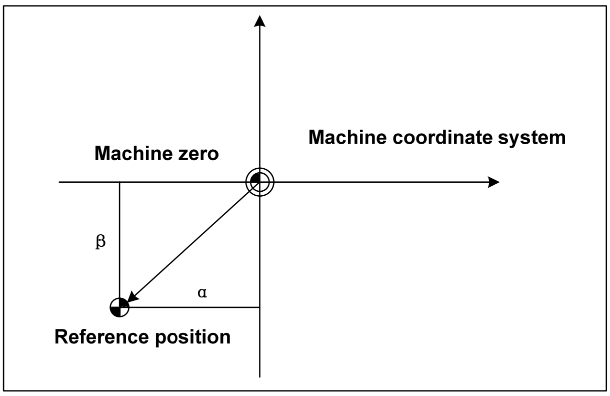
3.2.12 Selecting a Machine Coordinate System (G53)

Format: G53 X_ Y_ Z_

Function: The tool is positioned to corresponding coordinate in the machine coordinate system at a rapid traverse rate.

When the position is specified by a command on a machine coordinate system, the tool moves to the position by rapid traverse. G53, which is used to select a machine coordinate system, is a one-shot G code; that is, it is valid only in the block in which it is specified on a machine coordinate system. Specify an absolute command (G90) for G53. When an incremental command (G91) is specified, the G53 command is ignored. When the tool is to be moved to a machine-specific position such as a tool change position, program the movement in a machine coordinate system based on G53.

When manual reference point Return is performed after power-on, a machine coordinate system is set so that the reference position is at the coordinate values (α, β).



Explanations

1. When the G53 command is specified, the cutter compensation, the tool length offset and the tool offset are cleared.
2. Since the machine coordinate system must be set before the G53 command is specified, at least one manual reference point Return (zero-return should be operated in the manual mode) or automatic reference point Return by the G28 command must be performed after

the power is turned on. This is not necessary when an absolute-position detector is attached.

3.2.13 Floating Coordinate System (G92)

Format: G92 X_ Y_ Z_

Function:

When an absolute command traverse the tool to one position, the coordinate system must be preset by the commands. One coordinate system is set by the commands, the coordinate system origin is set by the offset amount and the origin of the previous coordinate system, which is called as the workpiece coordinate system. Once it is set, the following absolute commands must refer to the coordinate value in the workpiece coordinate system.

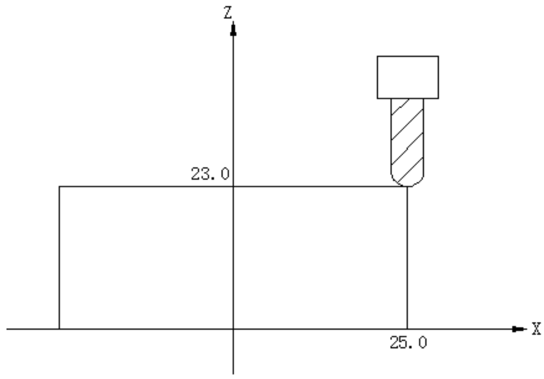


Fig 3-7

G92 X25.0 Z23.0;

At the starting of the block, G92 commands the tool nose coincides with the starting position of the program, which is shown as the above program.

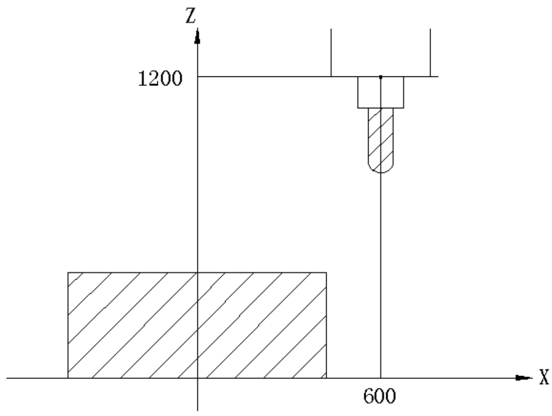


Fig 3-8

G92 X600.0 Z1200.0;

G92 commands the tool position coincides with the starting position of the program at the beginning of the program, which is shown as the above figure, and one absolute command is executed, and the reference position is positioned on the specified point. To position the tool nose on the specified point, the difference from the tool nose to the reference position must be corrected through the tool length compensation.

Explanations:

1. During offset, in the coordinate system set by G92, the tool coordinate value in the specified point doesn't include the offset value.
2. When the workpiece coordinate system is set, the tool position (such as the tool nose) is in the specified coordinate position. If G92 sets the coordinate system during the tool length offset, G92 sets the coordinate system through the coordinate value without the offset. The cutter compensation is deleted by G92 temporarily.
3. After the external workpiece zero position offset value is set, the coordinate system isn't affected by the offset value when G92 sets the coordinate system. For example. When G92X100.0Z80.0 is commanded, the tool current position is specified by the coordinate system as X=100.0, Z=80.0.

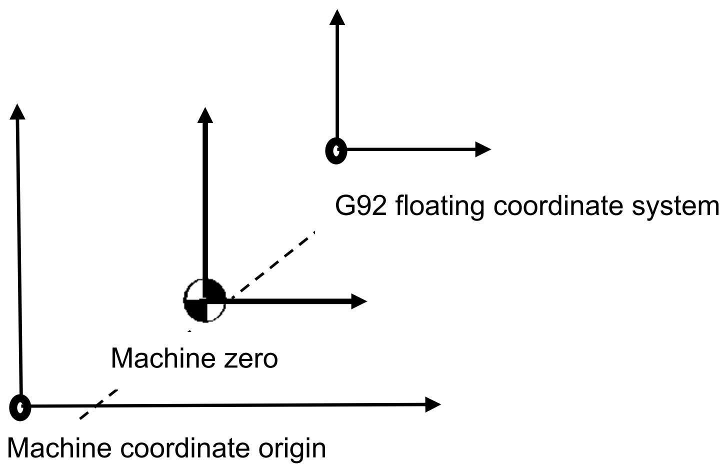


Fig.3-9

The origin corresponding G92 floating coordinate system is the value of the machine coordinate system shown as the above figure, and it is set after the machine is returned to the mechanical zero, and it is irrelevant to the work piece coordinate system.

After G92 setting, they become valid before the following situations:

1) Before system powers off
2) Before calling the workpiece coordinate system
3) Before operating the machine zero return

G92 floating coordinate system is usually for correcting the machining the temporary workpiece, and it gets lost after power-off. Usually, it runs at the starting position of the program or G92 is commanded in MDI mode before the program auto running.

Restriction: After setting the floating coordinate system, the first canned cycle command must be executed in the complete form; otherwise, the tool feeds wrongly.

3.2.14 Local Coordinate System (G52)

When a program is created in a workpiece coordinate system, a sub-coordinate system can be set to facilitate programming. Such a sub-coordinate system is called local coordinate system.

Format

G52 IP_;   set the local coordinate system

¦

G52 IP0;   cancel the local coordinate system

IP_:The origin point of the local coordinate system

Explanation:

A local coordinate system can by set in all the workpiece coordinate systems (G54 to G59) by command G52 IP_; . The origin point of the local system is set at the position specified by IP_ in the workpiece coordinate system. When a local coordinate system is set, the coordinates in the local coordinate system are used in an axis shift command. The local coordinate system can be changed by specifying the G52 command with the origin point of a new local coordinate system in the workpiece coordinate system. To cancel the local coordinate system and specify the coordinate value in the workpiece coordinate system, the origin point of the local coordinate system should be matched with that of the workpiece coordinate system.

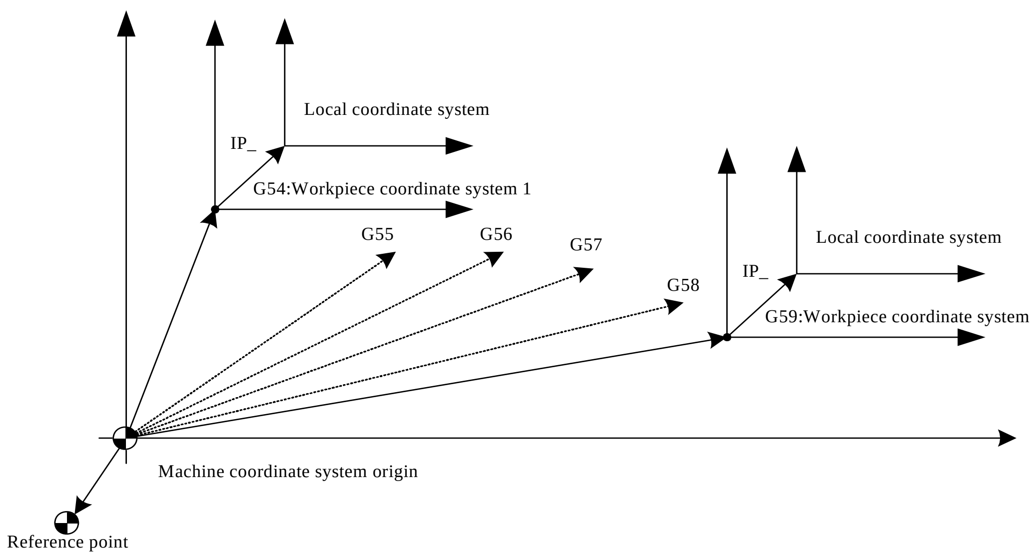


Fig. 3-10 Setting a local coordinate system

Warning:

1. When an axis returns to the reference point by the manual reference point return function. The origin point of the local coordinate system of the axis matches that of the work coordinate system. The same is true when the following command is issued:

G52 α0

α: Axis which returns to the reference point

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

3. Whether the local coordinate system is cancelled at reset depends on the parameter setting. The local coordinate system is cancelled when No.1031#4(G52) or No.1031#3（RLC）is set to 1.

4. If coordinate values are not specified for all axes when a workpiece coordinate system is set with the G92 command, the local coordinate systems of axes for which coordinate values were not specified are not cancelled, but remain unchanged.

3.2.15 Plane Selection G17/G18/G19

Format:G17/G18/G19

Function: Select the planes for circular interpolation, cutter compensation, drilling or boring by G17/G18/G19.

Explanations:

The system defaults G17 plane without commanding parameter after power on. Or, the plane can be set by the bit parameter **N0:1801#1, #2** after power on. The corresponding relation between the command and the plane is shown below:

G17-------------XY plane
G18-------------ZX plane
G19-------------YZ plane

When the axial address isn't specified in G17, G18 or G19 block, it is assumed that the addresses of basic three axes are omitted.
The plane is unchanged in the block in which G17, G18 or G19 is not commanded.

Examples:

G18 X_ Z_; ZX plane
G0 X_ Y_; Plane remains unchanged (ZX plane)

Moreover, the movement command is irrelevant to the plane selection. For example, when G17Z_ is specified, Z moves.

3.2.16 Starting/Canceling Polar Coordinate (G16/G15)

Format: G16/G15

The end position coordinate value can be input in polar cordons (radius and angle). The plus direction of the angle is counterclockwise of the selected plane first axis + direction, and the minus direction is clockwise. Both radius and angle can be commanded in either absolute or increment command (G90, G91）. When G51 is set, the polar coordinate mode can be canceled, and the coordinate value can be input through the Cartesian coordinate.

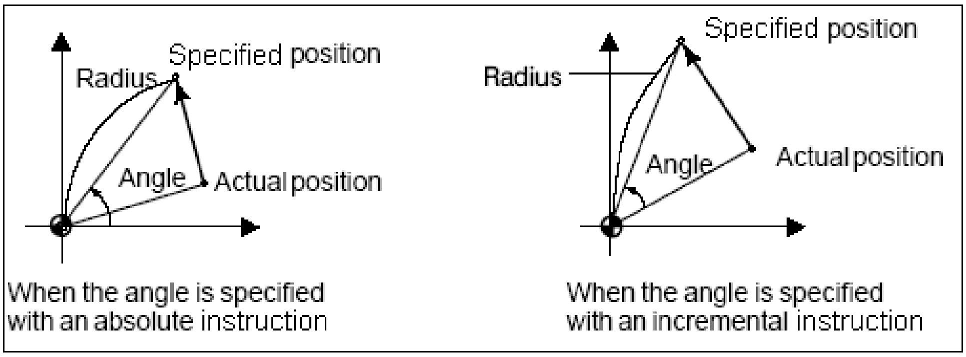
Function format:
G□□G00**G16** Start the polar coordinate command (polar coordinate mode)
G〇〇 IP_; Polar coordinate command
G15; Cancel polar coordinate command (cancel polar coordinate mode)

G□□ Select plane by the polar coordinate commands （G17, G18 or G19）
G00 G90 specifies the zero position of the workpiece coordinate system as the origin of the polar coordinate system, from which a radius is measured.
G91 specifies the current position as the origin of the polar coordinate system, from which a radius is measured.
IP_ Specifying the addresses of axes constituting the plane selected for the polar coordinate system, and their value.
The 1st axis: polar coordinate radius

The 2nd axis: polar coordinate angle

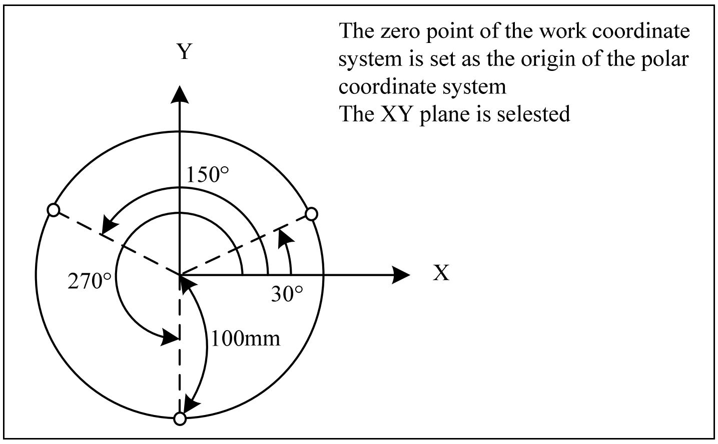
The regulation of the polar coordinates origin:

1. In G90 absolute mode, when G16 commands, the workpiece coordinate system zero position is set as the polar coordinate origin.



2. In G91 increment mode, when G16 commands, the current position is set as the polar coordinates origin.

Such as: Bolt hole circle (The zero position of the work piece coordinate system is taken as the origin of the polar coordinates, select X—Y plane).



- **Specifying angles and a radius with absolute commands**

G17 G90 G16;

Specifying the polar coordinates command and selecting XY plane

setting the zero position of the work piece coordinate system as the origin of the polar coordinate system

G81 X100.0 Y30.0 Z-20.0 R-5.0 F200.0;

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;

Canceling the polar coordinate command

- **Specifying angles with incremental commands and a radius with absolute commands**

G17 G90 G16;
Specifying the polar coordinates command, selecting XY plane
setting the zero position of the work piece coordinate system as the origin of the polar coordinate system
G81 X100.0 Y30.0 Z-20.0 R-5.0 F200.0;
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;
Canceling the polar coordinate command

Moreover, when the polar coordinates are used for programming, pay attention to setting the current coordinate plane. The polar coordinate plane is relative with the current coordinate plane, For example, in G91, if the current coordinate plane is G17, X and Y axial vector of the current tool position is set as the origin. If the current coordinate plane is G18, Z and X vector of the current tool position is taken as the origin.

After G16, the positioning parameter of the first hole cycle command isn't specified, the system sets the current position which the tool is as the defaulted positioning parameter of the hole cycle. At present, the first canned cycle command after the polar coordinates must be complete; otherwise, the tool feeds wrongly.

After G16, except for the hole cycle, the tool traverse command positioning coordinate is connected with the plane selection mode. For example, in mode G18, about the positioning parameter of the hole cycle command, the system sets the 1st axis as X, the 2nd as Y; however, about the positioning parameter of the basic interpolation commands, the system sets the 1st axis as Z, the 2nd as X.

After G15 cancels the polar coordinates and follows with the movement command, default the position, which the current tool is, is the starting position of the movement command.

Restriction:

1．Specifying a radius in the polar coordinate mode

In the polar coordinate mode, specify a radius for circular interpolation or helical cutting (G02, G03) with R.

2．Axes those are not considered part of a polar coordinate command in the polar coordinate mode

Axes specified for the following commands are not considered part of the polar coordinate command:

—Dwell（G04）

—Programmable data input （G10）

—Setting the local coordinate system （G52）

—Switching the workpiece coordinate system（G92）

—Selecting the machine coordinate system （G53）

—Stored stroke check（G22）

—Coordinate system rotation （G68）

—Scaling （G51）

2. Optional-angle chamfering/corner rounding

Neither optional-angle chamfering nor corner rounding can be commanded in polar coordinate mode.

3.2.17 Scaling in the Plane G51/G50

Format:

1）Equivalent scaling of each axis

G51 X_ Y_ Z_ P_ （X_Y_Z_: Absolute command for center coordinate value of scaling, P: Equivalent scaling for each axis）

… Scaled machine block

G50 Cancel the scaling

2）Different scaling of each axis:

G51 X_ Y_Z_ I_ J_ K_（X_Y_Z_:Absolute command for center coordinate value of scaling, I_J_K_: Scaling magnification for X axis, Y axis and Z axis respectively）

… Scaled machining block

G50 Canceling the scaling

The scaling ratio range which can be commanded is shown as below:

0.001 time～99.999 times（P1～P99999）

Function: The programmed figure, of which the specified point is set as the center by G51, can be scaled up or down with the same or different rates of magnification. Moreover, specify G51 in a separate block. After the figure is enlarged or reduced, specify G50 to cancel the scaling mode.

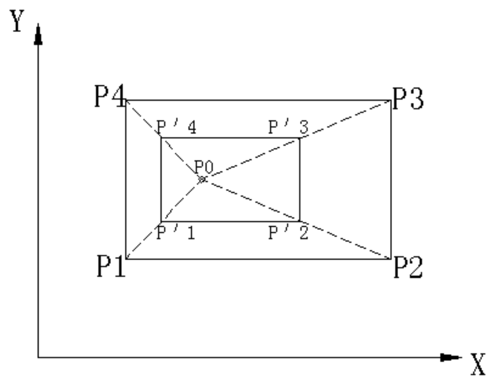


Fig 3-9 Scaling (P1P2P3P4→P'1P'2P'3P'4)

P1～P4: The figure of the machining program

P'1～P'4:The figure after scaling

P0: The scaling center

If P isn’t specified, the scaling ratio can be set by the parameter; if X, Y or Z is omitted, the position commanded by G51 is taken as the scaling center.

The scaling is not applicable to offset values, such as cutter compensation values, tool length offset values and tool position offset values, etc.

Explanations:

1. Scaling center: G51 is with 3 position parameters X_Y_Z_ and they can be selected. The positioning parameter specifies the scaling center of G51. If the positioning parameter isn't specified, the system sets the tool current position as the scaling center. No matter the current positioning mode is the absolute or the increment mode, the scaling center is specified by the absolute positioning mode. Moreover, in polar coordinate mode G16, the parameter commanded by G51 is indicated by the Cartesian coordinate system.

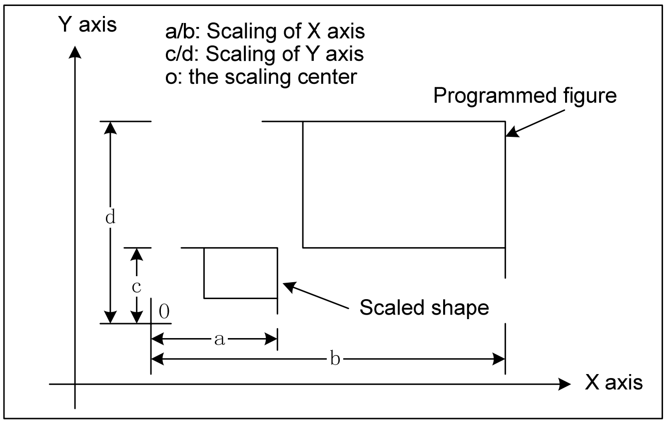


Fig. 3-10

Examples:

G17 G91 G54 G0 X10 Y10;

G51 X40 Y40 P2;

Although in the increment mode, the scaling center is the absolute coordinate（40,40）of G54 coordinate system.

G1 Y90; Parameter Y still uses the increment mode.

2. Scaling: No matter it is G90 or G91 mode, the scaling is still indicated by the absolute mode. The scaling can be specified in the program, and also be set in the parameter. The data parameter **P1862** respectively corresponds to the magnifications of X, Y, Z, 4th and 5th; if there isn't the scaling magnification command, the scaling can be performed by data parameter **P1861**.

   If the parameter P or the parameter values of I, J and K are negative values, the mirror images are used for the corresponding axes.

3. Setting scaling: Whether the single axis scaling is valid is set by bit parameter **N0:1850#3;** whether the scaled mirror image of each axis is valid is set by bit parameter **N0:1850#6** and the scaling magnification unit is set by bit parameter **N0:1850#7.**

4. Scaling cancel: After G50 cancels the scaling and follows with the movement command, it defaults the position, which the tool is, is the starting position of the movement command when the coordinate rotation is canceled.

5. In scaling mode, G27,G28,G29,G30 or commands related to the coordinate system (G54 to G59, G92) must not be specified. When any of these G codes is necessary, specify it after canceling scaling mode.

6. Even if different magnifications are applied to each axis in circular interpolation, the tool will not trace an ellipse.

When different magnifications are applied to axes and a circular interpolation is specified with radius R, it becomes as following figure 3-11 (in the example shown below, a magnification of 2 is applied to the X-component and a magnification of 1 is applied to the Y-component.).

```
G90 G0 X0.0 Y100.0;
G51 X0.0 Y0.0 Z0.0 I200.0 J100.0;
G02 X100.0 Y0.0 R100.0 F500.0;
```

Above commands are equivalent to the following one:

```
G90 G0.0 X0.0 Y100.0 Z0.0;
G02 X200.0 Y0.0 R200.0 F500.0;
```

Scaling of radius R depends on I or J which is larger

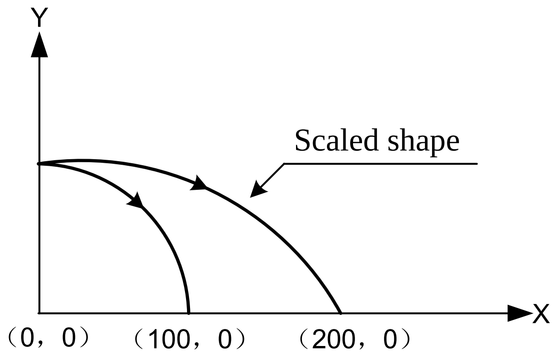


Fig. 3-11 Scaling for circular interpolation 1

When different magnifications are applied to axes and a circular interpolation is specified with I, J and K, it becomes as following figure 3-12 (In the example shown below, a magnification of 2 is applied to the X-component and a magnification of 1 is applied to the Y-component.).

```
G90 G0 X0.0 Y0.0;
G51 X0.0 Y0.0 Z0.0 I200.0 J100.0;
G02 X100.0 Y0.0 I0 J-100.0  F500.0;
```

Above commands are equivalent to the following one:

```
G90 G0.0 X0.0 Y100.0 ;
G02 X200.0 Y0.0 I0 J-100.0 F500.0;
```

In this case, the end position isnot on the radius, a linear section is included

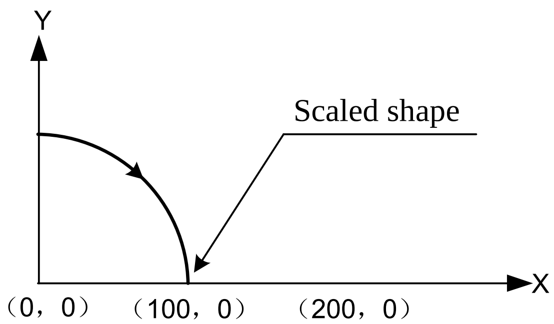


Fig 3-12 Scaling the arc interpolation 2

7. The scaling is invalid for the cutter compensation value, the tool length compensation value and the tool offset value, which is shown as Fig. 3-13.

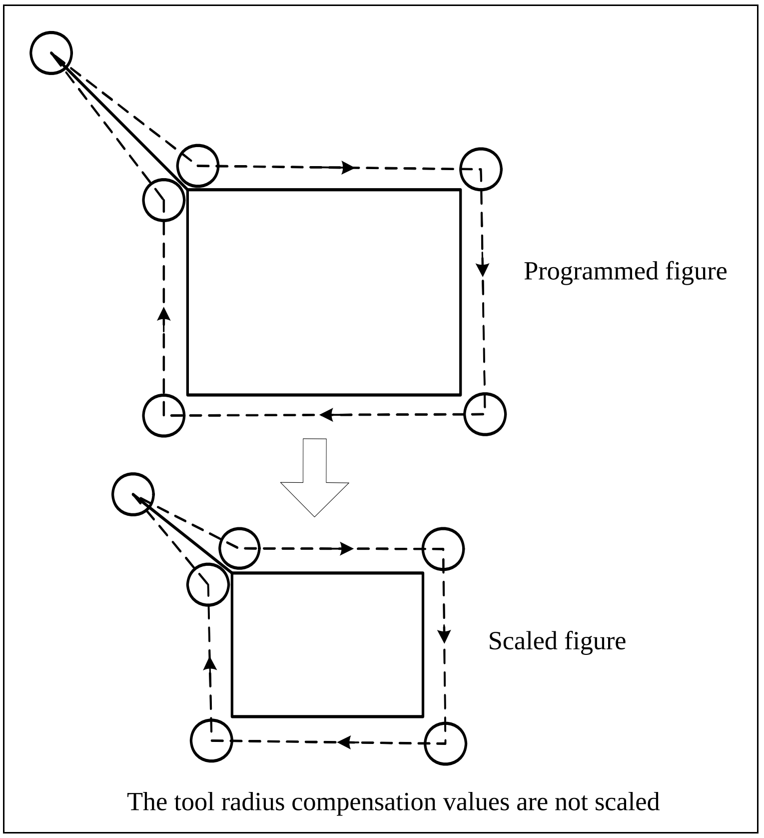
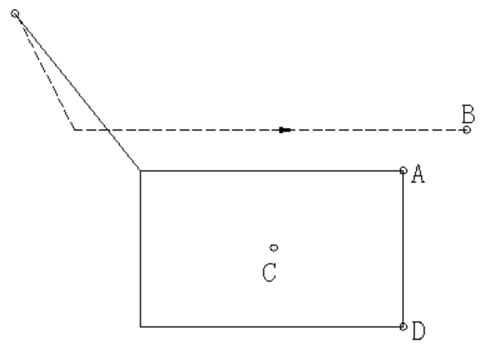


Fig. 3-13 Scaling during cutter compensation

Explanation:

1. The position displays as the coordinate value after scaling.
2. If a parameter setting value is employed as a scaling magnification without specifying P, the setting value at G51 command time is employed as the scaling magnification, and a change of this value, if any, is not effective.
3. Whether each axis scaling function is valid is set by the parameter; in G51 mode, arc radius is set by R command, and the function is always valid without any connection with setting parameter. The scaling function is always invalid for the additional axis.
4. Scaling function is invalid for the manual operation, but it is valid in DNC, auto or MDI mode.
5. Scaling is not applicable to the Z-axis movement in case of the following canned cycle.
   * Cut-in value Q and retraction value d of peck drilling cycle (G83,G73).
   * Shift value of X and Y axes in fine boring cycle（G76）and in back boring cycle (G87).
6. Specify G27, G28, G29, G30 and G92 in G50 mode.
7. If scaling results are rounded by counting fractions of 5 and over as a unit and disregarding the rest, the move amount may became zero. In this case, the block is regarded as a no movement block. Therefore, the tool traverse caused by the tool compensation C may get affected.
8. Reset

   Reset in (a)G51 mode, the original programming coordinate changes into the current coordinate value or the scaled coordinate. Therefore, after resetting, the movement depends on the increment command or the absolute.



Reset in position B, and take position A as B. When the movement command is executed in position D, the following movement depends on the increment command or the absolute one.

*** Increment**

If the movement value from position A to D is the increment, D' becomes the target position on the programming path, and convert position D'into position E. Because only Y axis movement is commanded, the tool traverses to position E_Y.

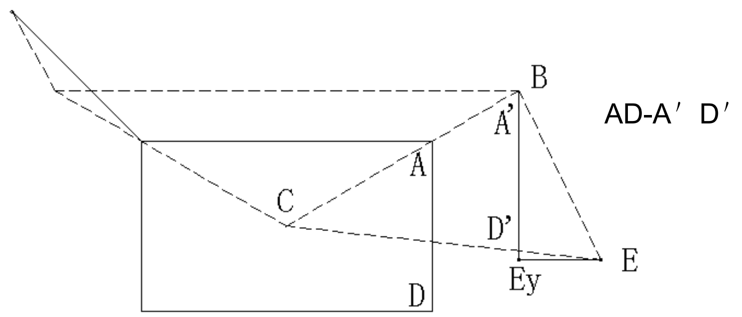


Fig. A

*** Absolute**

If position D is absolute, the tool traverses to position E converted from position D.

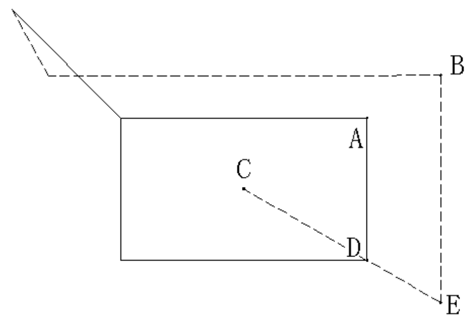


Fig.B

(b) Clear through resetting, which is set by parameter 007 BIT3 CLER.

After G51 mode is converted into G50, if the movement command is the increment, the tool traverses to position D'(refer to figure A); if it is absolute (refer to figure B), the tool traverses to position D.

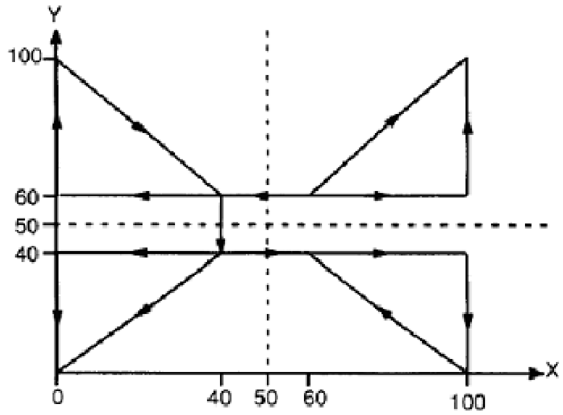
Mirror image program examples:

Main program

G00 G90;
M98 P9000;
G51 X50.0 Y50.0 I1 J-1;
M98 P9000;
G51 X50.0 Y50.0 I-1 J-1;
M98 P9000;
G51 X50.0 Y50.0 I-1 J1;
M98 P9000;
G50;

Subprogram:

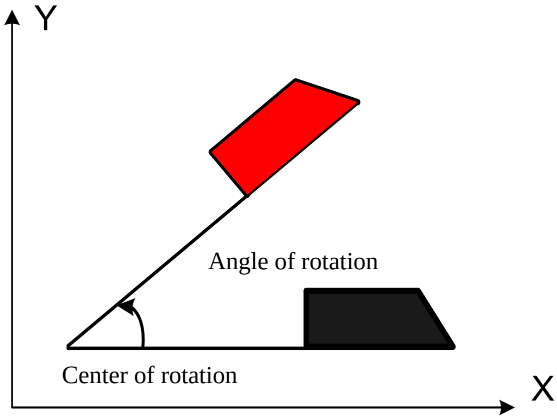
O9000
G00 G90 X60.0 Y60.0;
G01 X100.0 F100;
G01 Y100;
G01 X60.0 Y60.0;
M99;



3.2.18 Coordinate System Rotation G68/G69

A programmed shape can be rotated. When there is a pattern comprising some identical shapes in the positions rotated from a shape, the coordinate rotation function can just program the subprograms of the shape and the subprogram is called after rotation.

Format: G17 G68 X_ Y_ R_
Or G18 G68 X_ Z_ R_
Or G19 G68 Y_ Z_R_
G69



Function: The programmed shape rotates with the origin, which is the specified center position, through G68 , the coordinate system rotation is canceled by G69.

Command meaning:

G17 (G18 or G19): Select the plane and it contains the figure to be rotated.

X_Y_ Absolute command for two of the X_, Y_ and Z_ axes that corresponds to the current plane selected by a command (G17,G18 or G19). The command specifies the coordinates of the center of rotation for the values specified subsequent to G68.

R_ Angular displacement with a positive value indicates CCW rotation. Bit 0 of parameter 1850 selects whether the specified angular displacement is always considered an absolute value or is considered an absolute or incremental value depending on the specified G code (G90 or G91) .

Least input increment: 0.001deg.
Valid data range: -360.000~~360.000

Explanation:

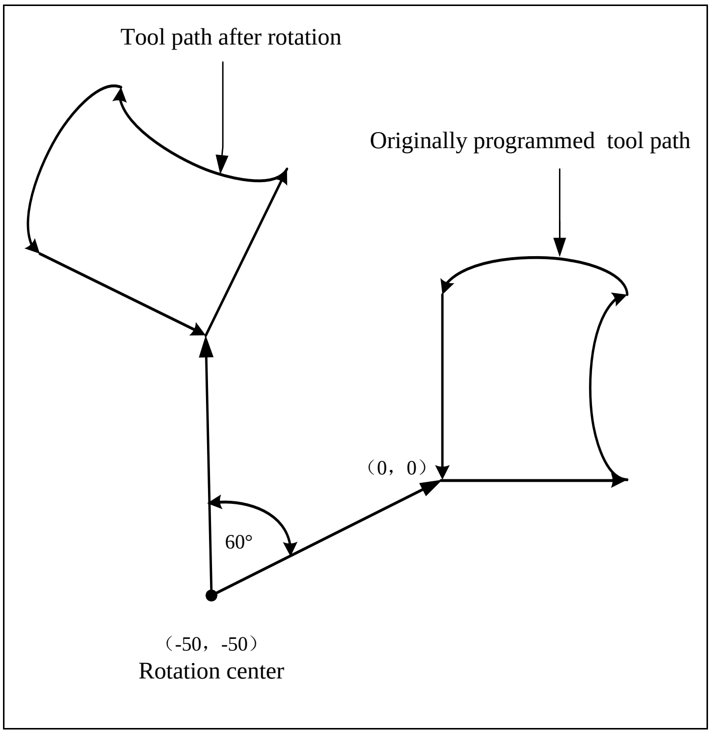
1．G68 is an optional parameter which is held with 2-position parameter. The position parameter specifies the rotation center. If the rotation center isn’t specified, the current tool position is assumed as the rotation center. The position parameter is connected with the current coordinate plane, X and Y are selected in G17; Z and X are selected in G18; Y and Z are selected in G19.
2．No matter the current positioning mode is the absolute or the relative, the rotation center can only be specified by the Cartesian coordinate system absolute positioning mode. G68 is also with one command parameter R, the parameter value is the rotation angle, the positive value indicates CCW rotation. The rotation angle unit is the degree. During the coordinate rotation, if there isn’t the rotation angle command, the rotation angle is set by the data parameter **P1860.**
3．In G91 mode, the rotation angle = the rotation angle of the last time + the angle specified by R in current G68 command.
4．When the system is in the rotation mode, the plane selection can’t be operated; otherwise, the alarm is issued. Please pay attention to it during programming.
5． In the coordinate system rotation mode, G codes related to reference point Return（G27, G28, G29 and G30, etc）and those for changing the coordinate system（G52 to G59 and G92, etc）must not be specified. If any of these G codes is necessary, specify it only after canceling

coordinate system rotation mode.

6. The first move command after the coordinate system rotation cancel command (G69) must be specified with absolute values. If an incremental move command is specified, correct movement will not be performed.
7. Cutter compensation, tool length offset, tool offset, and other compensation operations are executed after the coordinate system is rotated.
8. If a coordinate system rotation command is executed in the scaling mode (G51), the coordinate value of the rotation center will also be scaled, but not the rotation angle (R). When a move command is issued, the scaling is applied firstly and then the coordinates are rotated. A coordinate system rotation command (G68) should not be issued in cutter compensation mode (G41,G42) on scaling mode (G51). The coordinate system rotation command should always be specified prior to setting the cutter compensation mode.

Example 1: Rotation

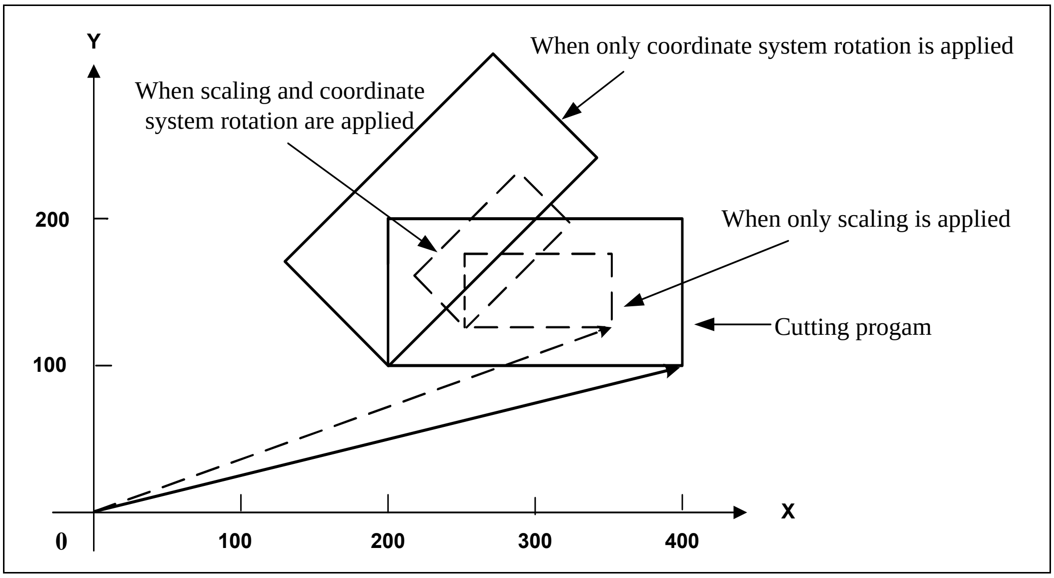
G92 X-50 Y-50 G69 G17;
G68 X-50Y-50 R60;
G90 G01 X0 Y0 F200;
G91 X100;
G02 Y100 R100;
G3 X-100 I-50 J-50;
G01 Y-100;
G69 ;



Example 2: Scaling and coordinate system rotation

G51 X300 Y150 P0.5;
G68 X200 Y100 R45;
G01 G90 X400 Y100;
G91 Y100;
X-200;

Y-100;
X200;
G69 G50;



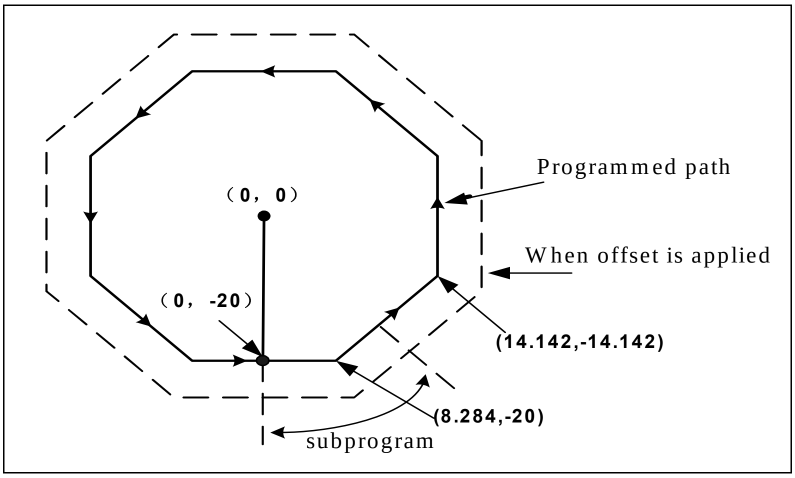
Example 3: Repetitive commands for coordinate system rotation (G68)
It is possible to store one program as a subprogram and recall subprogram by changing the angle.

Basic program (main program)
G92 X0 Y0 Z20 G69 G17;
M3 S1000;
G0Z2 ;
G51 X0 Y0 I1.2 J1.2;
G42 D01; (Setting the tool offset)
M98 P2100(P02100); (Calling the subprogram)
M98 P2200L7; （Calling for seven times）
G40;
G50;
G0 G90 Z20;
X0Y0;
M30;

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

Subprogram 2100
O2100;
G90 G0 X0 Y-20; (Set through the right tool compensation mode)

G01Z－2 F200;
X8.284;
X14.142 Y-14.142;
M99;



3.2.19 Inch/Metric Conversion (G20/G21)

Format: G20: inch input
G21: mm input

Function: Input in the inch system or the metric system.

Explanation:

1. Either inch or metric input can be selected by G code.

System of unit	G codes	Least input unit
inch (the inch system)	G20	0.0001inch
Mm (the metric system)	G21	0.0001mm

The two G codes must be specified in an independent block before setting the workpiece coordinate system at the beginning of the program.

N10 G20;
N20 G92 X—Y—;

The following contents depend on the metric or the inch system;

（1）Feedrate commanded by F code
（2）Positional command
（3）Workpiece zero position offset value
（4）Unit of scale for MPG
（5）Movement distance in incremental feed
（6）Some parameters

When the power is turned on, the G code is the same as that held before the power was turned off.

Positions for attention:

1. Inch and metric input must not be switched during a program.
2. When switching inch input to metric input and vise versa, the tool compensation value must be

preset based on the least input increment.

3. For the first G28 command after switching inch input to metric input or vice verse, operation from the intermediate point is the same as that of manual reference point Return.

4. When the least input increment and the least command increment systems are different, the maximum error is half of the least command increment, and the error isn't accumulated.

5. Switch between metric and inch input, which is set by the bit parameter **N0:0001#2.**

6. G20 and G21 must not be switched during a program.

7. When the mechanical system is different with the program system, the maximum error is half of the least movement unit and the error isn't accumulated.

3.2.20 Adding Workpiece Coordinate Systems（G54.1Pn）

Besides the six workpiece coordinate systems (standard workpiece coordinate system) selectable with G54 to G59, 48 additional workpiece coordinate systems (additional work piece coordinate systems) can be used. Alternatively, up to 300 additional workpiece coordinate systems can be used.

Format: G54.1 Pn;

Pn:Codes specifying the additional work piece coordinate systems.

n :1～48

G54.1 P1 …Additional workpiece coordinate system 1

G54.1 P2 …Additional workpiece coordinate system 2

⋮

G54.1 P48 …Additional workpiece coordinate system 48

The workpiece zero position offset value is set in an additional workpiece coordinate system:

G10L20Pn IP_;

Pn : Codes specifying the workpiece coordinate system for setting the workpiece zero position offset value.

n :1～48

IP_: Axis addresses and a value set as the workpiece zero position offset

Explanations:

1. Selecting the additional workpiece coordinate systems

1) When a P code is specified together with G54.1, the corresponding coordinate system is selected from the additional workpiece coordinate systems（1～48）.

2) Once the workpiece coordinate system is selected, it remains valid till the other workpiece coordinate system is selected.

3) Standard workpiece coordinate system 1 (G54) is selected at power-on.

2. Setting the workpiece zero position offset value in the additional workpiece coordinate systems

Similar as the standard workpiece coordinate system, the following operations can be performed for a workpiece zero position in an additional workpiece coordinate system:

（1） Press 

to display and set a workpiece zero position offset value.

（2） The G10 function enables a workpiece zero position offset value to be set by programming.
（3） A custom macro program allows a workpiece zero position offset value to be handled as a system variable.
（4） The workpiece zero position offset data can be input or output as the external data.
(5) When an absolute workpiece zero position offset value is specified, the specified value becomes a new offset value. When an incremental workpiece zero position offset value is specified, the specified value is added to the current offset value to produce a new offset value.

Limitations:
After G54.1, a P code must be specified. If G54.1 is not followed by a P code in the same block, additional workpiece coordinate system 1（G54.1P1）is assumed. If a value not within the specifiable range is specified in a P code, a P/S alarm（No.030）is issued: Illegal compensation number. In a G54.1 block, P codes other than workpiece offset numbers cannot be specified.

Example) G54.1(G54)P1000 G04;

3.3 Reference Position G Codes

The reference position is the specified one on the machine, and the tool can position on it through the reference point Return command. About the reference position, there are three command modes. For example, by G28, the tool is automatically moved to the reference position via an intermediate point along a specified axis; By G29, the tool is automatically moved from the reference position to a specified point via an intermediate point along a specified axis.

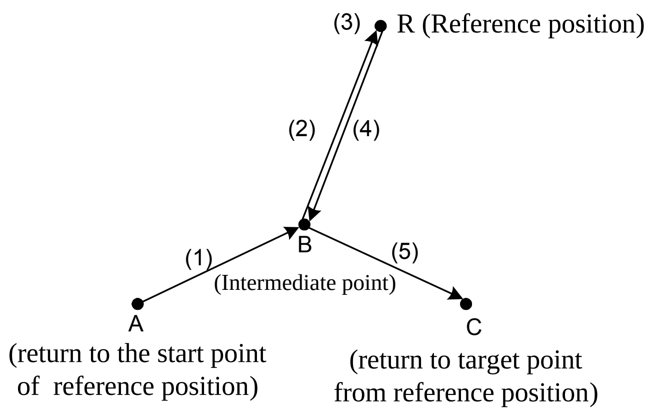


Fig 3-14

3.3.1 Reference Point Return Check G27

Format: G27 X_ Y_ Z_ ;

Function: The reference point Return check is executed by a G27 command, X_ Y_ Z_ specifies the command (absolute value/incremental command) of the reference position.

Explanation:

1. If the tool is on the reference position, the reference point Return indicator corresponding to controllable axis is on. If the tool doesn’t reach the reference position, an alarm occurs. After the reference point Return, the next block is executed continuously if M00 or M01 is not

performed in a block. If each cycle doesn't require the reference point Return, optional block skip function is used.

2. When the machine tool system is an inch system with metric input, the reference point Return indicator is also on even if the programmed position is shifted for 1μfrom the reference position. This is because the least setting increment of the machine tool system is smaller than its least command increment.
3. In the offset mode, the position to be reach by the tool with G27 command is the position obtained by adding the offset value. Therefore, if the position with the offset value added is not the reference position, the indicator is off, but an alarm is displayed instead. Usually, cancel offsets before G27 is commanded.
4. The indicator of the completion of return is off when the machine lock is turned on, even when the tool has automatically returned to the reference position. In this case, it is not checked whether the tool has returned to the reference position even when a G27 command is specified.
5. Before a machine coordinate system is established with the first reference point Return after power-on, the manual and automatic reference point Return federates and automatic rapid traverse rate are same. Even after a machine coordinate system is established upon the completion of reference point Return, the manual reference point Return feedrate conforms to the setting value of the parameter.

3.3.2 Reference Point Return G28

Format: G28 X_ Y_ Z_

Function: When G28 is commanded, the tool returns to the reference position via the intermediate point and the reference position is one specified point on the machine.

Explanations:

The commanded axis can auto position on the reference position, X/Y/Z is the move command and specified by G90/G91 (absolute/incremental command) .

The end position of the command is called as the "intermediate point" and the coordinate value specified by the command is saved in NC, and used by G29 (return from the reference position command).

The movement of G28 block is introduced as below:

Firstly, all controlled axes all rapidly position in the intermediate point and then return to the reference position via the intermediate point. If the machine isn't locked, the reference point Return indicator is on.

Positioning in the intermediate point and the reference position are equivalent to that of G00.

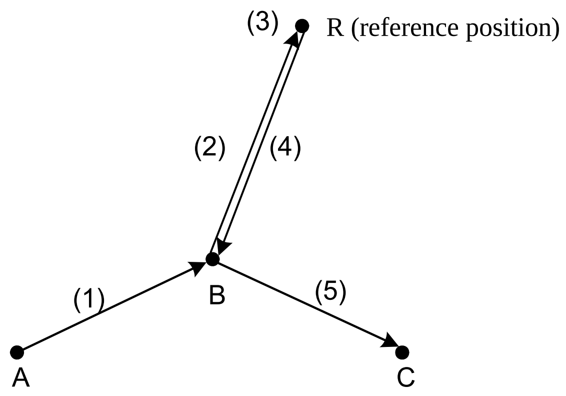


Fig. 3-15

1． The movement of G28 block can be divided into the following (refer to fig 3-15):

(1) Position from the current position to the intermediate point of the commanded axes at the rapid traverse rate (point A→point B).

(2) Position from the intermediate point to the reference position at the rapid traverse rate (point B→point R).

2． G28 is one-shot command, and only valid for the current block.

3． It supports the single axis or the multi-axes and returns to the reference position; during changing the workpiece coordinate, the coordinate of the intermediate point is saved in the system.

Example 1:

N1 G90 G54 X0 Y10;

N2 G28 X40 ; Set the intermediate point in X axis as X40 in G54 workpiece coordinate system, passing point（40,10）to return to the reference position, namely, X axis returns to the reference position independently.

N3 G29 X30 ; From the reference position and passing the point （40, 10）and returns to point (30, 10), X axis returns to the destination position independently.

N4 G01 X20;

N5 G28 Y60 ; The intermediate point (X40, Y60), because there aren't any commands in X axis, X40 commanded by G28 can be replaced. Notice: The intermediate point isn't（20, 60）.

N6 G55; Change the workpiece coordinate, then the intermediate point changes from the point (40, 60) of G54 workpiece coordinate system into point (40, 60) of G55 workpiece coordinate system.

N7 G29 X60 Y20; From the reference position, pass the intermediate point (40, 60) of G55 work piece coordinate system and returns to the point (60, 20).

G28 auto cancels the tool compensation. However, the command is normally used during auto tool changing, namely, after the reference point Return, the tool is changed in the reference position. Therefore, for safety, the cutter compensation and the tool length compensation should be cancelled before executing this command.

Note:

1. The coordinates for the move command and the intermediate point are stored only for the axes for which values are specified in a G28 block. For the other axes, the previously specified coordinates are used. (refer to example 1).

2. When the G28 command is specified and the manual return to the reference position has not been performed after the power has been turned on, the movement from the intermediate point is the same as in manual return to the reference position. In this case, the direction shifted from the intermediate point becomes the reference position direction specified in the parameter.
3. About the rotation axis, G28 is specified, the move direction from the intermediate point to the reference position becomes the direction of reference point Return set by the parameter. And the move amount should be in the range of 360°.

3.3.3 Return from the Reference Position G29

Format: G29 X_ Y_ Z_:
Function: When G29 is commanded, the tool traverses from reference position via the intermediate point specified by the G28 command and returns to the specified point.
Explanations:

The tools traverses from one intermediate point and positions on the specified point with the function and the command always follows one G28 command.

X/Y/Z is the move command and specified by G90/G91 (absolute/incremental command).

In the incremental command, the incremental value corresponding to the intermediate point must be specified.

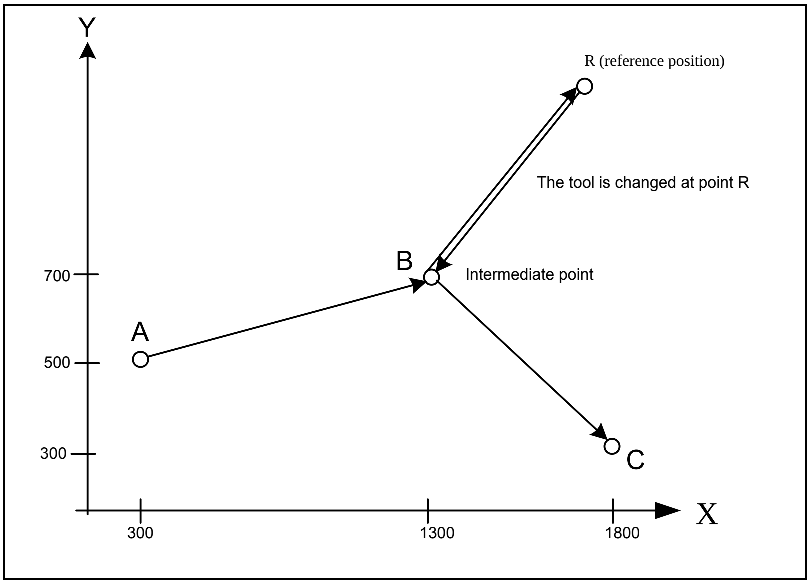
When G29 is commanded, all the commanded axes pass from the intermediate point commanded by G28 in the rapid feedrate and then reach the specified point.

The operation of positioning in the intermediate point and then in the specified point is exactly same as G00 position.

1. The movement of G29 block can be divided into the following steps (refer to figure 3-15):
   (1) Position from the reference position to the intermediate point specified by G28 at the rapid traverse rate (point R→point B).

   (2) Position from the intermediate point to the specified point at the rapid traverse rate (point B→point C).
2. G29 is one-shot information and only valid for the current block. Normally, after completing G28 command, return from the reference position command is executed immediately.
3. In G29 command format, the parameter X, Y and Z can be selected to specify the destination position for return from the reference position, such as point C in figure 3-15, and it can be indicated by the absolute or the incremental command. About the incremental programming, the commanded value specifies the incremental value away from the intermediate point. When some axis isn't specified, it means the relative intermediate point in the axis doesn't have any move amount. After G29, only the command with one axis returns independently, and other axes remains still.

Application examples of G28 and G29:



Example 1:

It is shown as above:
G91:
G28　X1000.0　Y200.0; （From A to B）
M06;
G29　X500.0　Y-400.0; （From B to C）

Example 2:

G90 G0 X10 Y10;

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

G29 X30;　　Return from the reference position (60, 30) via the intermediate point（30, 30）. Pay attention to that in the incremental programming mode, the vector of X axial direction is 60.

3.3.4 2nd, 3rd and 4th Reference Point Return G30

On the machine coordinate system, the four reference positions are set; but in the system without the absolute position detector, the 2nd, 3rd or 4th reference point Return can be executed only after completing auto reference point Return (G28) or manual reference point Return.

The specified axis move toward the 2nd , the 3rd or the 4th reference position by the following commands.

$$G30\left\{\begin{matrix} P2 \\ P3 \\ P4 \end{matrix}\right\} X\_\ Y\_\ Z\_;$$ （P2 can be omitted）

P2: The 2nd reference position

P3: The 3rd reference position

P4: The 4th reference position

The position of the 2nd, the 3rd and the 4th reference position is the coordinate value of each reference position on the mechanical coordinate system, which is preset by the parameter. The function is same as the reference point Return specified by G28 except the tool doesn't return to the 1st reference point. After completing G30 command and when G29 is specified, the tool positions from the intermediate point set by G30 to the specified position by G29, its movement process is same as that of specifying G29 after G28 commands.

Explanation:

1．X_ Y_ Z_: Commands specifying the intermediate position (absolute /incremental command).

2．G30 command setting and restrictions are same as those of G28; about setting the 2nd , 3rd, 4th reference positions, refer to data parameter **P1051～1053.**

3． After the power has been turned on, the manual reference point return or the auto reference point Return（G28）should be operated for one time before executing G30.

3.4 Canned Cycle G Codes

The canned cycle usually uses one block including G codes to replace some blocks for commanding the machining to simplify the programming (The system supports the canned cycle in the three planes and it normally defaults G17 plane.).

List 3-5 Canned cycles

G codes	Drilling（－Z direction）	Operation at the bottom of a hole	Retraction movement (＋Z direction)	Application
G73	Intermittent feed	——	Rapid traverse	High-speed peck drilling cycle
G74	Feed	Spindle CW	Feed	CCW tapping
G76	Feed	Oriented spindle stop	Rapid traverse	Fine boring cycle
G80	——	——	——	Cancel
G81	Feed	——	Rapid traverse	Drilling cycle (spot drilling cycle)
G82	Feed	Dwell	Rapid traverse	Drilling cycle (counter boring cycle)
G83	Intermittent feed	——	Rapid traverse	Peck drilling cycle
G84	Feed	Spindle CCW	Feed	Tapping cycle
G85	Feed	——	Feed	Boring cycle
G86	Feed	Spindle stop	Rapid traverse	Boring cycle
G87	Feed	Spindle CW	Manual operation or rapid traverse	Boring cycle (back boring)
G88	Feed	Dwell→ spindle stops	Manual operation or rapid traverse	Boring cycle
G89	Feed	Dwell	Feed	Boring cycle

The normal process of the canned cycle:

A canned cycle consists of a sequence of six operations (Fig.3-16).

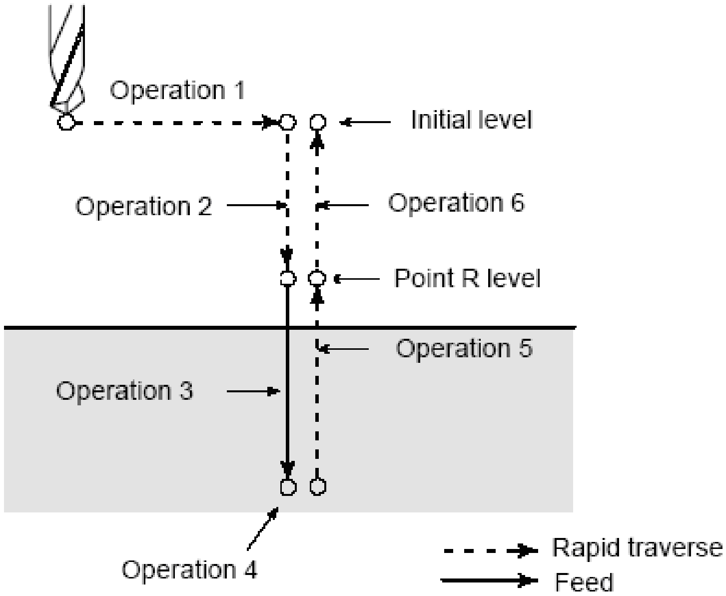


Fig. 3-16

Operation 1: Positioning of X and Y axes (including another axis)
Operation 2: Rapid traverse to point R
Operation 3: Hole machining
Operation 4:Operation at the bottom of a hole
Operation 5: Retraction to point R
Operation 6: Rapid traverse to the initial point

Position on XY plane, and the hole is machined in Z axis direction and one canned cycle operation is set by three modes. They are respectively specified by G codes.

① Data format
- G90 Absolute
- G91 Incremental

② Point plane return
- G98 The plane which the origin is
- G99 The plane which point R is

③ G73, G74, G76, G80, G81 … G89 Refer to list 3-5

Note: The plane, which the origin is, is the absolute value position in Z direction when the canned cycle cancel mode is switched into the canned cycle mode.

The initial level and point R level

The initial level: It means the absolute position in Z axis which the tool is before the canned cycle.

Point R level: It is called as the safe level. In the canned cycle, the rapid feed is switched into the cutting feed, the position in Z axis direction keeps some distance from the work piece surface to avoid the tool hitting the work piece and ensure the enough distance for speeding.

G73/G74 /G76/G81～G89 specify all the data of the canned cycle (the hole position data, hole machine data, repeated times), which forms a block.

The format of the hole machine mode is shown as below:

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

Hole machine data

Hole position data

Hole machine mode

Among them, the basic meaning of the hole position data and the hole machine data is shown as the list 3-6:

List 3-6

Specified content	Name of axis	Remark
Hole machine mode	G	Refer to list 3-5, and pay attention to the above restrictions.
Hole position data	X, Y	Specify the position of a hole through the absolute or incremental value, and the control is same as that of G00 position.
Hole machine data	Z	The incremental value specifies the distance from point R to the bottom of a hole or the absolute value specifies the coordinate value of the hole bottom, which is shown as 3-17(A). The feedrate is the speed specified by F in operation 3; while in operation 5, it can be the rapid feed or the speed specified by F code based on the hole machine mode.
	R	The incremental value specifies the distance from the initial level to point R in figure 3-17(B), or the absolute value specifies the coordinate value of point R. The feedrate is the rapid feed during the operations 2 and 6.
	Q	Specify the cutting amount of each time in G73 and G83, or the translation amount (incremental value) in G76 and G87.
	P	Specify the dwell time at the bottom of a hole. The canned cycle command can be with one parameter P_, the parameter value in P_ can specify the dwell time after the tool reaches Z plane. The unit is ms. The parameter minimum value is set by data parameter P281and the parameter maximum value is set by data parameter P282.

	F	Specify the cutting feedrate.
	K	Specify the repeated times in K_ parameter value, and K is only valid in the specified block. And it can be omitted and one time is assigned by default. The maximum drilling times are 99999. When it is specified as the negative value, and executed as its absolute value; if it is zero, the drilling isn't operated while only mode is changed.

Restrictions:

The canned cycle G command is the mode command, G codes remain valid till the canned cycle is canceled.

G codes, which include G80 and G codes of 01 group, cancels the canned cycle.

Once the machine date are specified in the canned cycle, and they remain valid till the canned cycle is canceled. Therefore, at the beginning of the canned cycle, the required hole machine data all are specified. The following canned cycle just specifies the rewritten data.

Remark: The cutting feed of F command still remains valid even the canned cycle is canceled.

In the single block, generally, the canned cycle uses the machine mode in three steps, position→point R level→initial level.

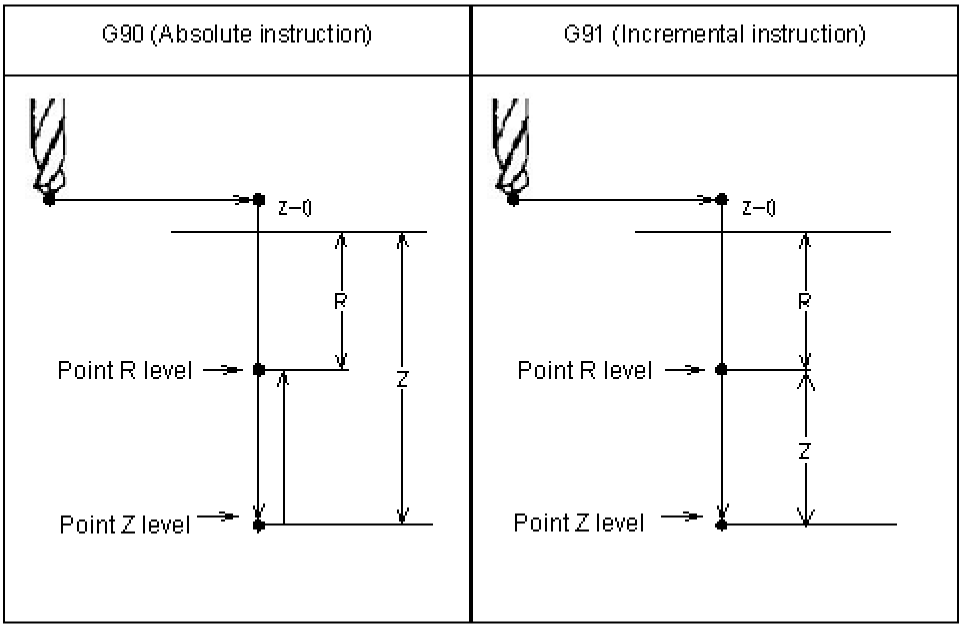
In the canned cycle, if it is reset, the hole machine data, the hole position data all are cleared. The examples of remaining the data valid and clearing the data are shown as the following list:

List 3-7

Order	Specifying the data	Remark
①	G00X-M3;	
②	G81X-Y-Z-R-F-;	At the beginning, specify the required value for Z, R and F.
③	Y-;	The hole machine mode and the hole machine data are same as those specify in hole ②, so G81 and Z-R-F- all can be omitted. The hole position moves to Y, the hole is machined for one time in G81 mode.
④	G82X-P-;	Compared with hole ③ position, the hole just moves in X axis. The hole is machined in G28 mode, Z, R and F specified in ② and P specified in ④ are assumed as the hole machine data.
⑤	G80X- Y-	The hole isn't processed. All the hole machine data are canceled.
⑥	G85X-Z-R-P-;	All the data are canceled in ⑤. Therefore, Z and R should be specified again, F is same as that specified in ②, it can be omitted. P isn't required in the block but just saved.
⑦	X- Z-;	Compared with ⑥, Z value is different in the hole machine, and the hole position just moves in X axis.
⑧	G89X-Y-;	Z specified in ⑦, R specified in ⑥ and F specified in P and ② are assumed as the hole machine data, and the hole is machined in G89 mode.
⑨	G01X-Y-;	Cancel the hole machine mode and the hole machine data.

A. G90/G91 The absolute or incremental value command of the canned cycle G90/G91

Along with the move distance of the drilling axis, the change of G90 and G91 is shown as figure 3-17. (Normally, programming with G90, if it is programmed by G91, Z and R are handled as the negative value,)



（A）　　　　　　　　　　（B）

Fig. 3-17

B. The level return commands in the canned cycle G98/G99

When the tool reaches the hole bottom, the tool can return to point R level or the initial level. Based on the difference between G98 and G99, the tool can return to the initial level or point R level.

Normally, G99 is used for the first drilling operation and G98 for the last. The initial level does not change even when drilling is performed in the G99 mode. The operation of commands G98 and G99 is shown as below.

The system defaults it as G98.

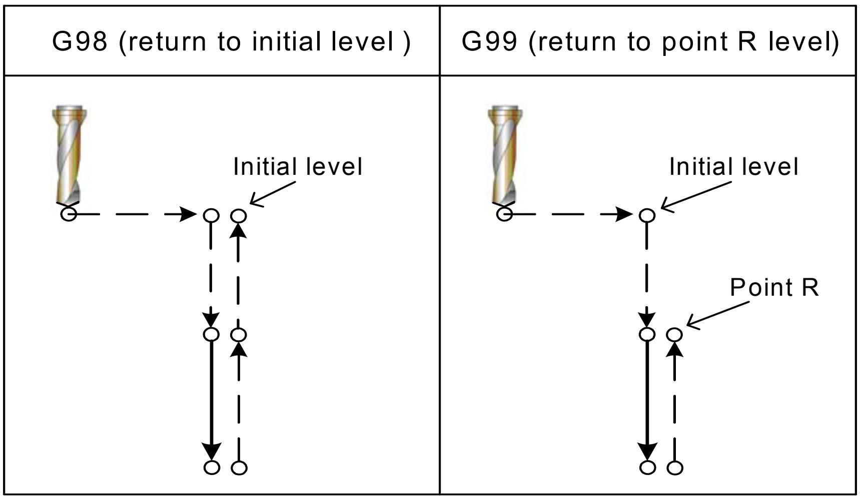
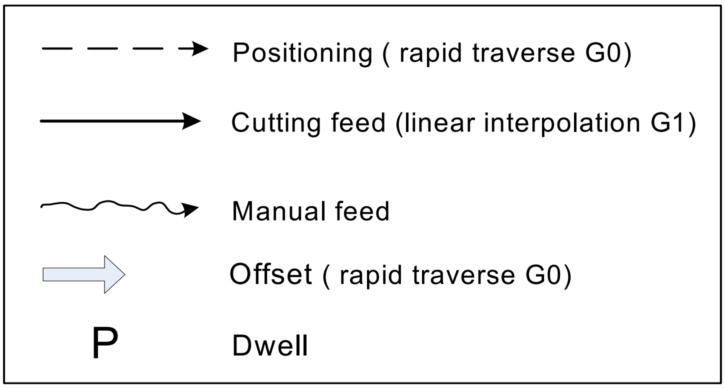


Fig. 3-18

Figures in these explanations of each canned cycle use the following symbols:



Description of each machine mode:

3.4.1 High-Speed Peck Drilling Cycle G73

Format: G73 X_Y_Z_R_Q_F_K_

Function: The cycle performs high-speed peck drilling. It executes intermittent cutting feed to the bottom of a hole while removing chips from the hole. About the operation, refer to Fig. 3-19.

Remark:

X_Y_: Hole position data

Z_: The incremental programming means the distance from point R to the hole bottom; the absolute programming means the absolute coordinate value of the hole bottom.

R_:The incremental programming means the distance from the initial level to point R; the absolute programming means the absolute coordinate value of point R.

Q_:Depth of cut for each cutting feed

F_:Cutting feedrate

K_:Number of repeats

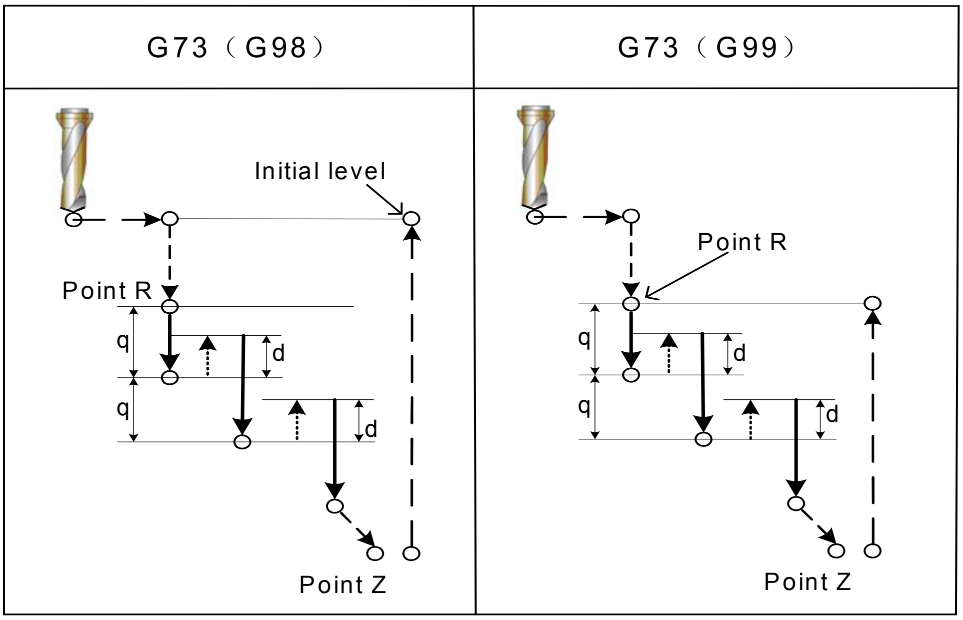


Fig. 3-19

Z and R: When the first hole is drilled, parameters Z and R must be specified correctly and not defaulted.

Q:When the command parameter Q is specified, the intermittent feed is performed shown as the above figure. Then, the system retracts based on the retraction value d set by the data parameter P2010 shown in figure 3-19, the tool intermittently rapidly retracts based on the distance d.

When the G73 command and an M code are specified in the same block, the M code is executed at the time of the first positioning operation. The system then proceeds to the next drilling operation.

When K is used to specify the number of repeats, the M code is executed for the first hole only; for the second and subsequent holes, the M code is not executed.

Note:

1. If the command parameter Q isn't specified, the system alarms: "Fail to find address Q （G73/G83）". If value Q is specified as the negative value, the intermittent feed is performed based on its absolute value.

2. In the canned cycle, the tool length offset (G43, G44 or G49) is specified, the offset is applied during positioning point R.

Restrictions:

Cancel: It's not allowed to specify a G code of group 01 (G00, G01, G02, G03 or G60) in a single block; otherwise, the alarm is issued.

Tool offset: In the canned cycle mode, the tool offset is ignored.

Example 1:

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

Note: In the above example, when the holes of 2～6 are machined, although Q is omitted, the chips are still removed.

3.4.2 Left-Handed Tapping Cycle G74

Format: G74 X_ Y_ Z_ R_ P_ F_

Function: This cycle performs left-handed tapping. In the left-handed tapping cycle, when the bottom of the hole has been reached, the spindle rotates clockwise.

Remark:

X_Y_:Hole position data

Z_:The incremental programming means the distance from point R to the hole bottom, the absolute programming means the absolute coordinate value of the hole bottom.

R_:The incremental programming means the distance from the initial level to point R; the absolute programming means the absolute coordinate value of point R.

P_:Dwell time

F_:Cutting feedrate

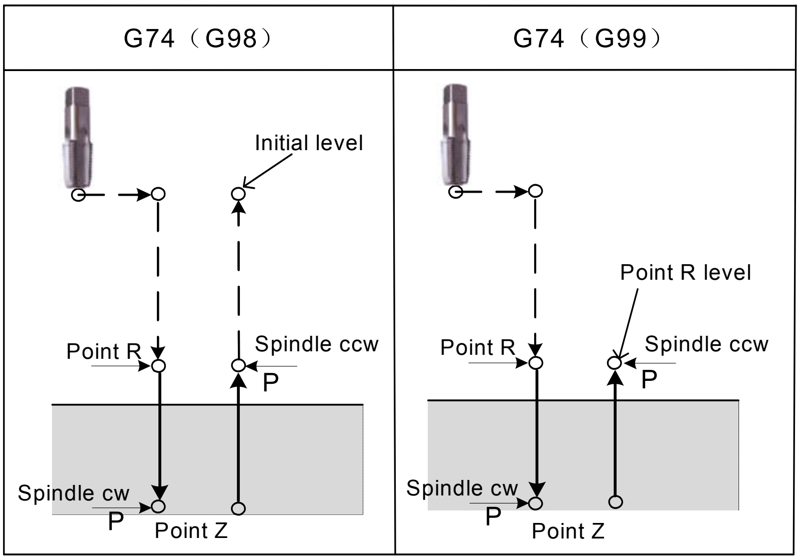


Fig. 3-20

Tapping is performed by turning the spindle counterclockwise. When the bottom of the hole has been reached, the spindle is rotated clockwise for retraction, This creates a reverse thread.

Feedrate overrides are ignored during left-handed tapping. A feed hold doesn't stop the machine unit the return operation is completed.

Before specifying G74, the miscellaneous function (M code) is used to rotate the spindle. If the spindle CCW rotation isn't commanded, the system changes into spindle CCW rotation on point R level based on the current spindle commanded speed.

When the G74 command and an M code are specified in the same block, the M code is executed at the time of the first positioning operation. The system then proceeds to the next drilling operation.

When K is used to specify the number of repeats, the M code is executed for the first hole only; for the second and subsequent holes, the M code is not executed.

In the canned cycle, when a tool length offset (G43, G44 or G49) is specified, the offset is applied during positioning to point R.

Examples:

M4 S100; Cause the spindle to start rotating;

G90 G99 G74 X300. Y-250. Z-150. R-120. P300 F120;

Position, tap hole 1, then return to point R;

Y-550.; Position, tap hole 2, then return to point R;

Y-750.; Position, tap hole 3, then return to point R;

X1000; Position, tap hole 4, then return to point R;

Y-550.; Position, tap hole 5, then return to point R;

G98 Y-750.; Position, tap hole 6, then return to the initial level;

G80;

G28 G91 X0 Y0 Z0 ; Return to the reference position;

M5; Cause the spindle to stop rotating.

M30;

Restrictions:

Cancel: It's not allowed to specify a G code of group 01(G00, G01, G02, G03 or G60) in the same block; otherwise, the alarm is issued.

Tool offset: In the canned cycle mode, the tool offset is ignored.

3.4.3 Fine Boring Cycle G76

Format: G76 X_Y_Z_Q_R_P_F_K_

Function: The fine boring cycle bores a hole precisely. When the bottom of the hole has been reached, the spindle stops, and the tool is moved away from the machined surface of the workpiece and retracted. Therefore, it can avoid the tool withdrawal mark when the tool withdraws and damaging the tool and keep the fineness of the machined surface.

Explanation:

X_Y_:Hole position data

Z_:The incremental programming means the distance from point R to the hole bottom; the absolute programming means the absolute coordinate value of the hole bottom.

R_:The incremental programming means the distance from the initial level to point R; the absolute programming means the absolute coordinate value of point R.

Q_:Offset value of the hole bottom

P_:Dwell time

F_:Cutting feedrate

K_:The times of the fine boring cycle

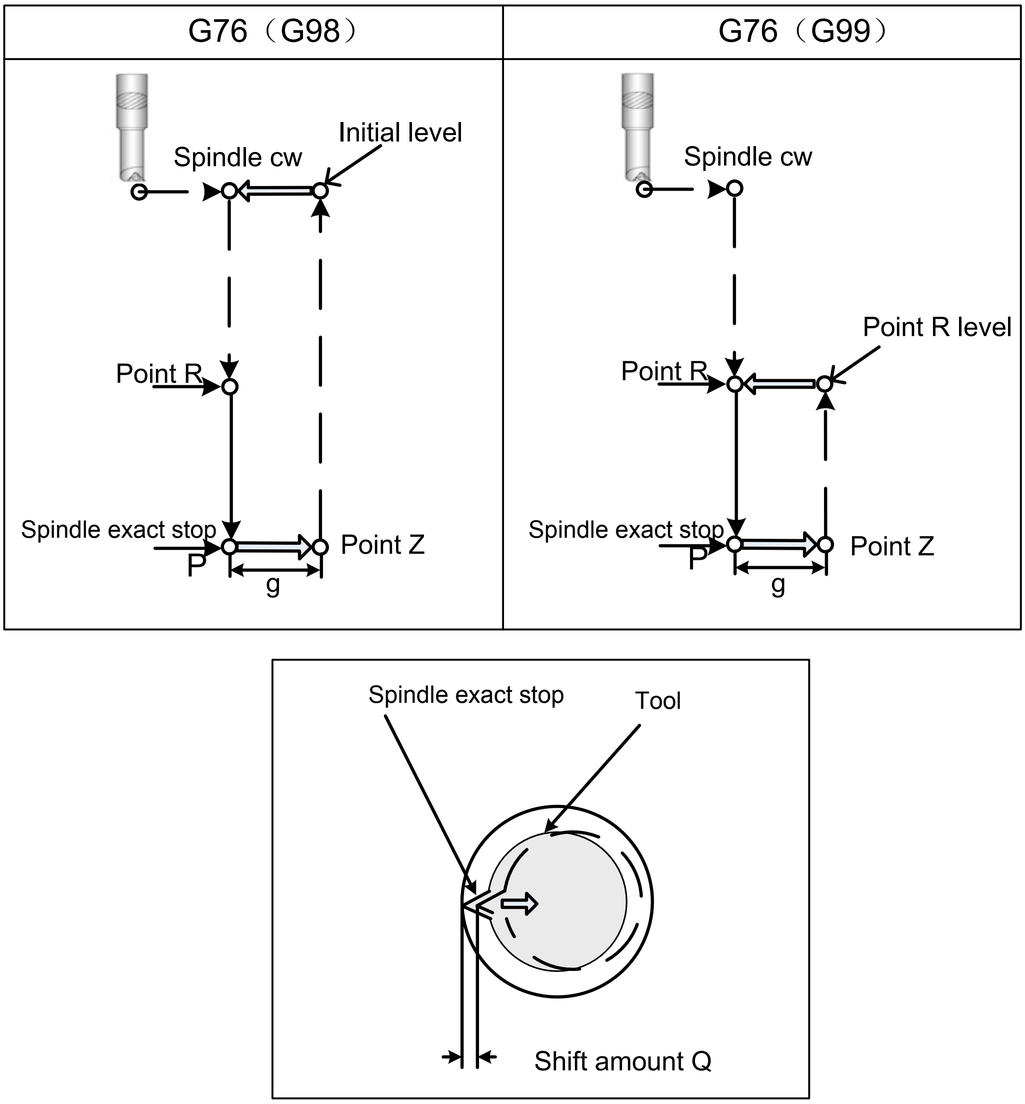


Fig. 3-21

Rapid traverse

Cutting feed

Jog feed ‘d’ is set by the parameter

Tool movement

OSS Oriented spindle stop (the spindle stops in the fixed point)

P Dwell

Note: G76 can only output M codes, which are set by the parameter 009 BIT7（FIX2）, as the output signals of the spindle CCW rotation, CW rotation and the spindle exact stop,

At the bottom of the hole, the spindle stops in the oriented position, and then the tool head is pulled after the offset on the machined surface, so the hole can be processed in high precision and high efficiency without damaging the work piece surface. The address Q which is always the positive number sets the offset value. If Q is specified with a negative value, the sign is ignored. Among ＋X, ＋Y, －X and －Y, one offset direction is preset by the parameter 022 BIT4 or 5（PMXY1 or 2）. Pay attention to that Q is modal value retained within canned cycles and is also used as the depth of cut for G73 and G83.

The tool offset can be set by addresses I and J at the hole bottom. X and Y axes move in the linear interpolation, which is set by parameter 022 BIT6（SIJ）, Q is replaced by the incremental value specified by I and J. Therefore, offset in any direction, the feedrate is same as the speed specified by F code. In the canned cycle mode, I and J are modal values. Commanding I and J can't complete machining the hole, but just specifies I and J again.

Switching the axis: Before the drilling axis can be changed, the canned cycle must be canceled.
Boring : In a block that does not contain X, Y, Z, R or any additional axes, boring isn't executed.

Examples:

M3 S500;	Cause the spindle to start rotating;
G90 G99 G76 X300.Y-250.;	Position, bore hole 1, and then return to point R;
Z-150. R-100.Q5;	Orient at the bottom of a hole, and then shift by 5mm;
P1000 F120;	Stop at the hole bottom for 1s;
Y-550.;	Position, bore hole 2, and then return to point R;
Y-750.;	Position, bore hole 3, and then return to point R;
X1000.;	Position, bore hole 4, and then return to point R;
Y-550.;	Position, bore hole 5, and then return to point R;
G98 Y-750.;	Position, bore hole 6, and then return to the initial level;
G80 G28 G91 X0 Y0 Z0;	Return to the reference position;
M5;	Cause the spindle to stop rotating.

Restrictions:

Cancel: It's not allowed to specify a G code of group 01(G00, G01, G02, G03 or G60) in the same block; otherwise, the alarm is issued.
Tool offset: In the canned cycle mode, the tool offset is ignored.

3.4.4 Canned Cycle Cancel G80

Format: G80
Function: G80 cancels canned cycles.

Explanation:

All canned cycles are canceled to perform normal operation. Points R and Z are cancelled. Other drilling and boring data are also canceled (cleared).

Examples:

M3 S100;	Cause the spindle to start rotating;
G90 G99 G88 X300. Y-250. Z-150. R-120. F120;	
	Position, bore hole 1, and then return to point R;
Y-550.;	Position, bore hole 2, and then return to point R;
Y-750.;	Position, bore hole 3, and then return to point R;
X1000.;	Position, bore hole 4, and then return to point R;
Y-550.;	Position, bore hole 5, and then return to point R;
G98 Y-750.;	Position, bore hole 6, and then return to the initial level;
G80;	
G28 G91 X0 Y0 Z0 ;	Return to the reference position and cancel the canned cycle;
M5;	Cause the spindle to stop rotating.

3.4.5 Drilling Cycle, Spot Drilling (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. The tool is then retracted from the bottom of the hole in rapid traverse.

Explanation:

X_Y_:Hole position data

Z_:The incremental programming means the distance from point R to the hole bottom; the absolute programming means the absolute coordinate value of the hole bottom.

R_:The incremental programming means the distance from the initial level to point R; the absolute programming means the absolute coordinate value of point R.

F_:Cutting feedrate.

K_:Number of repeat (if required).

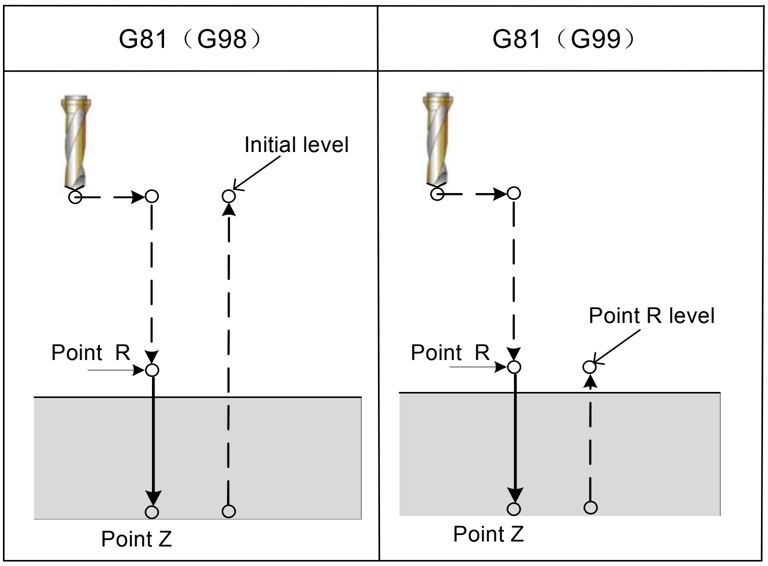


Fig. 3-22

Z & R: When the first drilling hole is executed, the hole bottom parameters Z and R must be specified correctly and not defaulted. If parameters P and Q are specified, the system ignores them.

After positioning along the X- and Y- axes, rapid traverse is performed to point R, and drilling is executed from point R to Z. The tool is then retracted in rapid traverse.

Before specifying G81, the spindle is rotated with a miscellaneous function (M code).

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

When K is used to specify the number of repeats, the M code is performed for the first hole only; for the second and subsequent holes, the M code is not executed.

When a tool length offset (G43, G44 or G49) is specified in the canned cycle, the offset is applied at the time of positioning to point R.

Examples:

M3 S2000; Cause the spindle to start rotating
G90 G99 G81 X300. Y-250. Z-150. R-10. F120;
Position, drill hole 1, and then return to point R;
Y-550.; Position, drill hole 2, and then return to point R;
Y-750.; Position, drill hole 3, and then return to point R;
X1000.; Position, drill hole 4, and then return to point R;
Y-550.; Position, drill hole 5, and then return to point R;
G98 Y-750.; Position, drill hole 6, and then return to the initial level;
G80;
G28 G91 X0 Y0 Z0 ; Return to the reference position;
M5; Cause the spindle to stop rotating.
M30;

Restrictions:

Cancel: It’s not allowed to specify a G code of group 01(G00, G01, G02, G03 or G60) in the same block; otherwise, the alarm is issued.
Tool offset: In the canned cycle mode, the tool offset is ignored.

3.4.6 Drilling Cycle, Counter Boring Cycle G82

Format: G82 X_ Y_ Z_ R_ P_ F_ K_;
Function: The cycle is used for normal drilling. Cutting feed is performed to the bottom of the hole. At the bottom, a dwell is performed, and then the tool is retracted in rapid traverse.

Remark:

X_Y_:Hole position data.

Z_:The incremental programming means the distance from point R to the hole bottom; the absolute programming means the absolute coordinate value of the hole bottom.

R_:The incremental programming means the distance from the initial level to point R; the absolute programming means the absolute coordinate value of point R.

F_:Cutting feedrate

P_:Dwell time

K_:Number of repeats

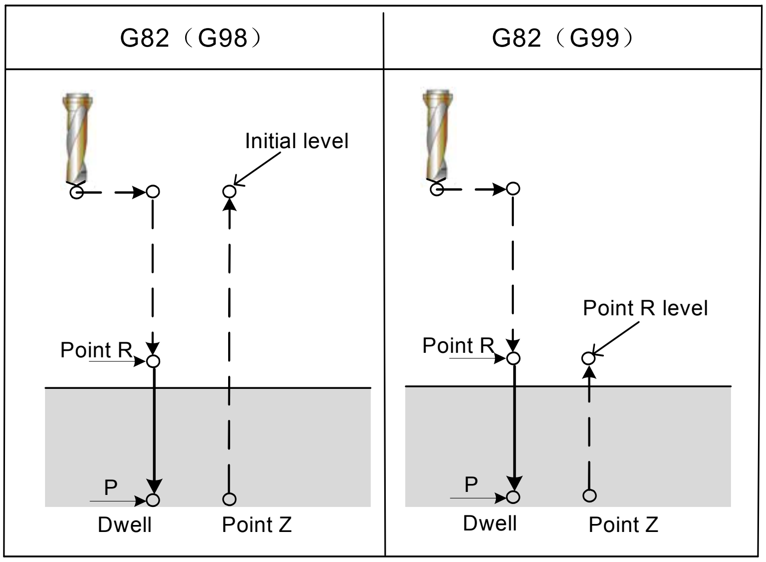


Fig.3-23

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

Before specifying G82, use a miscellaneous function (M code) to rotate the spindle.

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

When K is used to specify the number of repeats, the M code is performed for the first hole only; for the second and subsequent holes, the M code is not executed.

When a tool length offset (G43, G44 or G49) is specified in the canned cycle, the offset is applied at the time of positioning to point R.

Examples:

M3 S2000; Cause the spindle to start rotating;
G90 G99 G82 X300. Y-250. Z-150. R-100. P1000 F120
Position, drill hole 1, and dwell for 1s at the hole bottom, then return to point R;
Y-550; Position, drill hole 2, and dwell for 1s at the hole bottom, then return to point R;
Y-750; Position, drill hole 3, and dwell for 1s at the hole bottom, then return to point R;
X1000.; Position, drill hole 4, and dwell for 1s at the hole bottom, then return to point R;
Y-550; Position, drill hole 5, and dwell for 1s at the hole bottom, then return to point R;
G98 Y-750; Position, drill hole 6, and dwell for 1s at the hole bottom, then return to initial level;
G80; Canned cycle cancel;
G28 G91 X0 Y0 Z0 ; Return to the reference position;
M5; Cause the spindle to stop rotating.
M30;

Restrictions:

Cancel: It's not allowed to specify a G code of group 01(G00, G01, G02, G03 or G60) in the same

block.
Tool offset: In the canned cycle mode, the tool offset is ignored.

3.4.7 Peck Drilling Cycle (G83)

Format: G83 X_ Y_ Z_ R_ Q_ F_ K_
Function: This cycle performs peck drilling. It performs intermittent cutting feed to the bottom of a hole while removing shavings from the hole.

Explanation:

X_Y_:Hole position data.

Z_:The incremental programming means the distance from point R to the hole bottom; the absolute programming means the absolute coordinate value of the hole bottom.

R_:The incremental programming means the distance from the initial level to point R; the absolute programming means the absolute coordinate value of point R.

Q_:Depth of cut for each cutting feed.

F_:Cutting feedrate.

K_:Number of repeats.

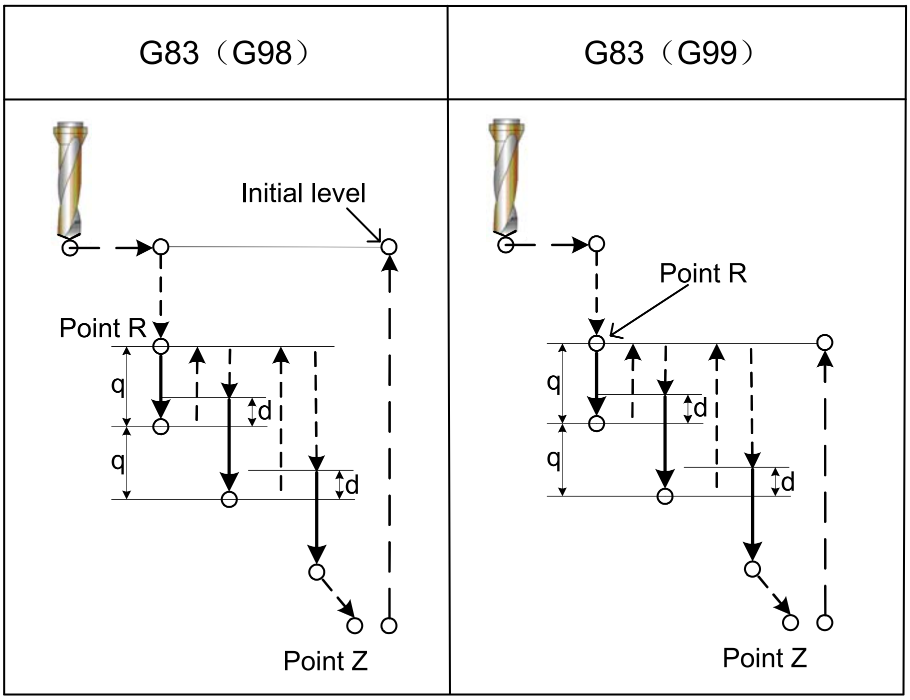


Fig. 3-24

Q:It represents the depth of cut for each cutting feed. It must always be specified as an incremental value. In the second and subsequent cutting feeds, rapid traverse is performed up to a d point just before where the last drilling ended, and cutting feed is performed again and d is set in parameter (**No.P2011)**. The operation is shown as figure **3-24**.

Be sure to specify a positive value in Q. Negative value are ignored and the system proceeds as the positive value.

Specify Q in the block executing the drilling. If it's specified in the block which doesn't execute the drilling, Q can't be saved as the modal data. Before specifying G83, use a miscellaneous function (M code) to rotate the spindle.

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

When K is used to specify the number of repeats, the M code is performed for the first hole only; for the second and subsequent holes, the M code is not executed.

When a tool length offset (G43, G44 or G49) is specified in the canned cycle, the offset is applied at the time of positioning to point R.

Examples:

M3 S2000; Cause the spindle to start rotating;
G90 G99 G83 X300. Y-250. Z-150. R-100. Q15 F120;
Position, drill hole 1, and then return to point R;
Y-550; Position, drill hole 2, and then return to point R;
Y-750; Position, drill hole 3, and then return to point R;
X1000; Position, drill hole 4, and then return to point R;
Y-550; Position, drill hole 5, and then return to point R;
G98 Y-750; Position, drill hole 6, and then return to initial level;
G80;
G28 G91 X0 Y0 Z0 ; Return to the reference position;
M5; Cause the spindle to stop rotating,
M30;

Restrictions:

Cancel: It's not allowed to specify a G code of group 01(G00, G01, G02, G03 or G60) in the same block.

Tool offset: In the canned cycle mode, the tool offset is ignored.

3.4.8 Right-Handed Tapping Cycle G84

Format: G84 X_ Y_ Z_ R_ P_ F_ ;

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

Explanation:

X_Y_:Hole position data.

Z_:The incremental programming means the distance from point R to the hole bottom; the absolute programming means the absolute coordinate value of the hole bottom.

R_:The incremental programming means the distance from the initial level to point R; the absolute programming means the absolute coordinate value of point R.

P_:Dwell time.

F_:Cutting feedrate.

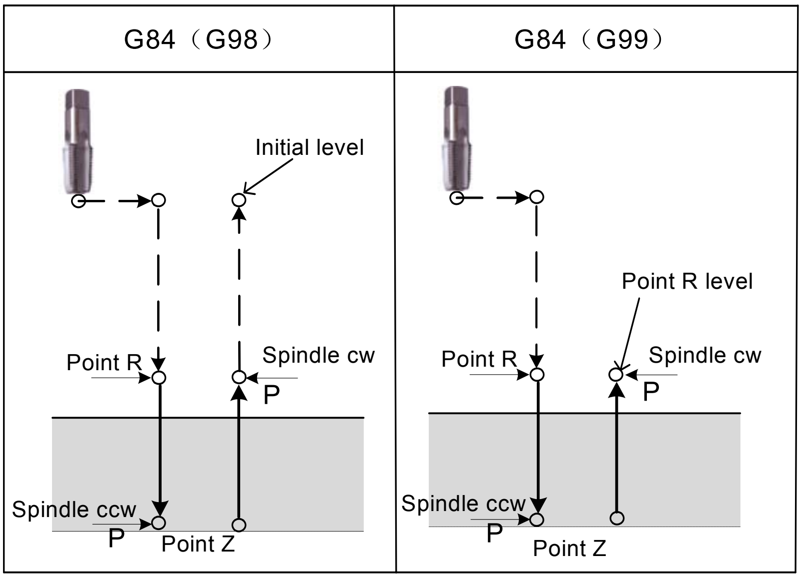


Fig. 3-25

Tapping is performed by rotating the spindle clockwise. When the bottom of the hole has been reached, the spindle is rotated in the reverse direction for retraction. This operation creates threads.

During tapping, feeding overrides are ignored. A feed hold does not stop the machine until the return operation is completed.

Before specifying G84, the spindle rotates with a miscellaneous function (M code). If the spindle CW rotation isn't commanded, the system changes into CW rotation on R level based on the current spindle command speed.

When the G84 command and an M code are specified in the same block, the M code is executed at the time of the first hole positioning operation. The system then proceeds to the next drilling operation.

When K is used to specify the number of repeats, the M code is performed for the first hole only; for the second and subsequent holes, the M code is not executed.

When a tool length offset (G43, G44 or G49) is specified in the canned cycle, the offset is applied at the time of positioning to point R.

In the feeding mode per minute, the relation among the thread lead, the feedrate and the spindle speed is as below:

Feedrate F = Tap pitch **X** Spindle speed S

For example: The bolt hole of M12×1.5 is tapped on the part, the parameter can be selected.

S500=500r/min F=1.5×500=750mm/min

If they are the multiple thread, F value can be obtained after multiplying the number of the thread.

Examples:

M3 S100;	Cause the spindle to start rotating;
G90 G99 G84 X300. Y-250. Z-150. R-120 P300 F120;	
	Position, tap hole 1, and then return to point R;
Y-550.;	Position, tap hole 2, and then return to point R;
Y-750.;	Position, tap hole 3, and then return to point R;
X1000;	Position, tap hole 4, and then return to point R;
Y-550.;	Position, tap hole 5, and then return to point R;
G98 Y-750.;	Position, tap hole 6, and then return to initial level;
G80;	
G28 G91 X0 Y0 Z0 ;	Return to the reference position;
M5;	Cause the spindle to stop rotating;
M30;	

Restrictions:

Cancel: It's not allowed to specify a G code of group 01(G00, G01, G02, G03 or G60) in the same block.
Tool offset: In the canned cycle mode, the tool offset is ignored.

3.4.9 Boring Cycle G85

Format: G85 X_ Y_ Z_ R_ F_ K_ ;
Function: The cycle is used to bore a hole.

Explanation:

X_Y_:Hole position data.

Z_:The incremental programming means the distance from point R to the hole bottom; the absolute programming means the absolute coordinate value of the hole bottom.

R_:The incremental programming means the distance from the initial level to point R; the absolute programming means the absolute coordinate value of point R.

F_:Cutting feedrate.

K_:Number of repeats.

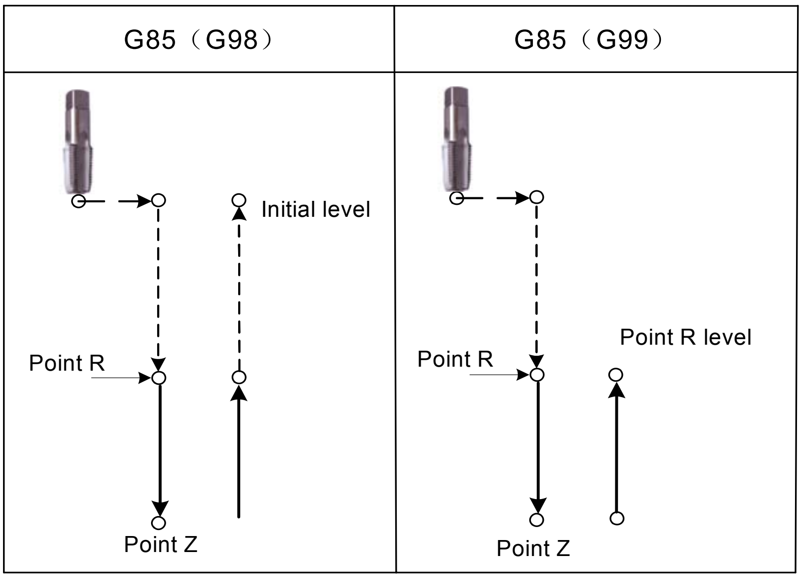


Fig. 3-26

After positioning along the X- and Y- axes, rapid traverse is performed to point R, and boring is then executed from point R to Z. When the bottom of the hole has been reached, cutting feed is performed to return to point R.

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

When the G85 command and an M code are specified in the same block, the M code is executed at the time of the first hole positioning operation. The system then proceeds to the next drilling operation.

When K is used to specify the number of repeats, the M code is performed for the first hole only; for the second and subsequent holes, the M code is not executed.

When a tool length offset (G43, G44 or G49) is specified in the canned cycle, the offset is applied at the time of positioning to point R.

Axis switching: Before the boring axis can be changed, the canned cycle must be canceled.

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

Examples:

```
M3 S100;                                    Cause the spindle to start rotating;
G90 G99 G85 X300. Y-250. Z-150. R-120. F120;
                                            Position, bore hole 1, then return to point R;
Y-550.;                                     Position, bore hole 2, then return to point R;
Y-750.;                                     Position, bore hole 3, then return to point R;
X1000.;                                     Position, bore hole 4, then return to point R;
Y-550.;                                     Position, bore hole 5, then return to point R;
G98 Y-750.;                                 Position, bore hole 6, then return to initial level;
G80;
G28 G91 X0 Y0 Z0 ;                          Return to the reference position;
M5;                                         Cause the spindle to stop rotating;
M30;
```

Restrictions:

Cancel: It's not allowed to specify a G code of group 01(G00, G01, G02, G03 or G60) in the same block.

Tool offset: In the canned cycle mode, the tool offset is ignored.

3.4.10 Boring Cycle G86

Format: G86 X_ Y_ Z_ R_ F_ K_;

Function: This cycle is used to bore a hole.

Explanation:

X_Y_:Hole position data.

Z_:The incremental programming means the distance from point R to the hole bottom; the absolute programming means the absolute value of the hole bottom.

R_:The incremental programming means the distance from the initial level to point R; the absolute programming means the absolute coordinate value of point R.

F_: Cutting feedrate.

K_: Number of repeats.

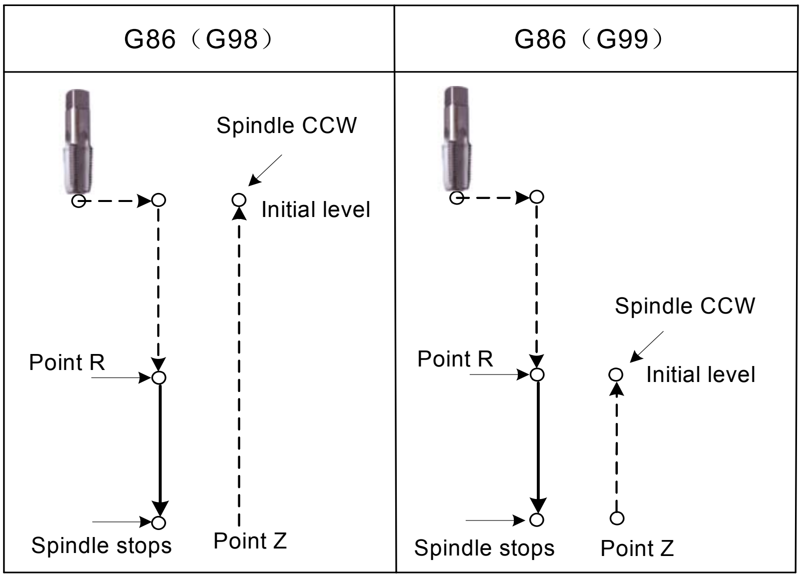


Fig. 3-27

After positioning along the X- and Y- axes, rapid traverse is performed to point R, and boring is then executed from point R to Z. When the spindle is stopped 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.

When the G86 command and an M code are specified in the same block, the M code is executed at the time of the first hole positioning operation. The system then proceeds to the next drilling operation.

When K is used to specify the number of repeats, the M code is performed for the first hole only; for the second and subsequent holes, the M code is not executed.

When a tool length offset (G43, G44 or G49) is specified in the canned cycle, the offset is applied at the time of positioning to point R.

Axis switching: Before the drilling axis can be changed, the canned cycle must be canceled.

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

Examples:

M3 S2000;	Cause the spindle to start rotating;
G90 G99 G86 X300. Y-250. Z-150. R-100. F120;	
	Position, bore hole 1, then return to point R;
Y-550.;	Position, bore hole 2, then return to point R;
Y-750.;	Position, bore hole 3, then return to point R;
X1000.;	Position, bore hole 4, then return to point R;
Y-550.;	Position, bore hole 5, then return to point R;
G98 Y-750.;	Position, bore hole 6, then return to point R;
G80;	
G28 G91 X0 Y0 Z0 ;	Return to the reference position;
M5;	Cause the spindle to stop rotating;
M30;	

Restrictions:

Cancel: It’s not allowed to specify a G code of group 01(G00, G01, G02, G03, G60 or G86) in the same block.

Tool offset: In the canned cycle mode, the tool offset is ignored.

3.4.11 Boring Cycle, Back Boring Cycle (G87)

Format :G87 X_Y_Z_R_Q_ F_ ;

Function: The cycle performs accurate boring.

Explanation:

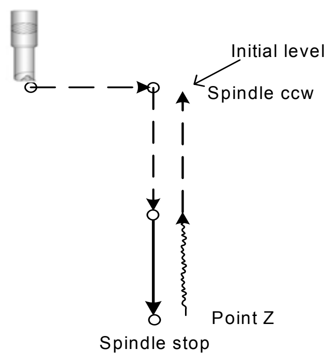
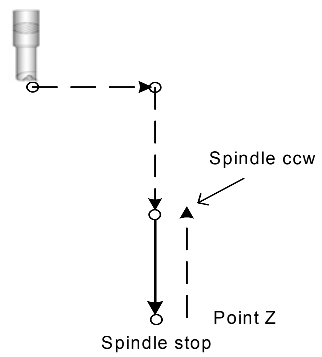
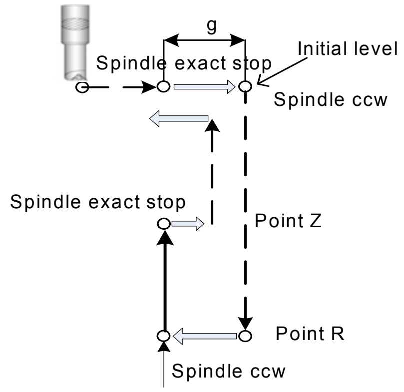
X_Y_:Hole position data.

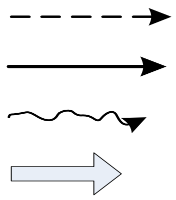
Z_:The incremental programming means the distance from point R to the hole bottom; the absolute programming means the absolute coordinate value of the hole bottom.

R_:The incremental programming means the distance from the initial level to point R; the absolute programming means the absolute coordinate value of point R.

Q_:Offset value at the hole bottom.

F_:Cutting feedrate.

	G87 (G98)	G87 (G99)
Canned cycle Ⅰ		
Canned cycle Ⅱ		No use



Rapid traverse
Cutting feed
Jog feed
Tool movement

Fig. 3-28

Canned cycle I (Boring cycle)

When the tool reaches the hole bottom, the spindle stops, the system accesses the feed hold state. In such case, the tool traverses in the manual mode; When any manual operation is executed, the tool retracts from the middle of the hole for safety.

To restart processing, switch into DNC or auto mode, and press the cycle start key. Through G98 or G99, the tool returns to the position which the origin is or point R is, the spindle CW turns and the next block commands are executed.

Canned cycle Ⅱ (back boring cycle)

After positioning along the X- and Y- axes, the spindle is stopped at the fixed rotation position. The tool is moved in the direction opposite to the tool tip, positioning(rapid traverse) is performed to the bottom of the hole (point R). The tool is then shifted in the direction of the tool tip and the spindle is rotated clockwise.

Boring is performed in the positive direction along the Z-axis until point Z is reached. Even with P command, the operation is continued. At point Z, the spindle is stopped at the fixed rotation position again, the tool is shifted in the direction opposite to the tool tip, then the tool is returned to the initial level. The tool is then shifted in the direction of the tool tip and the spindle is rotated clockwise to proceed to the next block operation. X and Y axes offset values and directions are exactly same as those in G76, and the setting direction is same as that of G76 and G87.

Explanation:

Canned cycle I:

The signals SRV and SSP are the output signals of the spindle CCW turning and stopping, which are set by parameter 009 BIT7（FIX2）.

Canned cycle Ⅱ:

M codes are assumed as the output signals of the spindle CCW turning, stopping and orientation stopping, which are set by parameter 009 BIT7（FIX2）.

Axis switching: Before the boring axis can be changed, the canned cycle must be canceled.

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

Remind:

During programming the back boring, the values of Z and R must be specified. Normally, point Z is above point R; otherwise, the system alarms.

Examples:

M3 S500; Cause the spindle to start rotating;

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

Position, bore hole 1. Orient at the initial level, then shift by 5mm. Stop at point Z for 1s;

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

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

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

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

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

G80 G28 G91 X0 Y0 Z0 ; Return to the reference position;

M5; Cause the spindle to stop rotating.

Restrictions:

Cancel: It’s not allowed to specify a G code of group 01(G00, G01, G02, G03, G60 or G86) in the same block.

Tool offset: In the canned cycle mode, the tool offset is ignored.

3.4.12 Boring Cycle (G88)

Format: G88 X_Y_Z_R_ P_F_ ;

Function: This cycle is used to bore a hole.

Explanation:

X_Y_:Hole position data

Z_:The incremental programming means the distance from point R to the hole bottom; the absolute programming means the absolute coordinate value of the hole bottom.

R_:The incremental programming means the distance from the initial level to point R; the absolute programming means the absolute coordinate value of point R.

P_: Dwell time

F_: Cutting feedrate

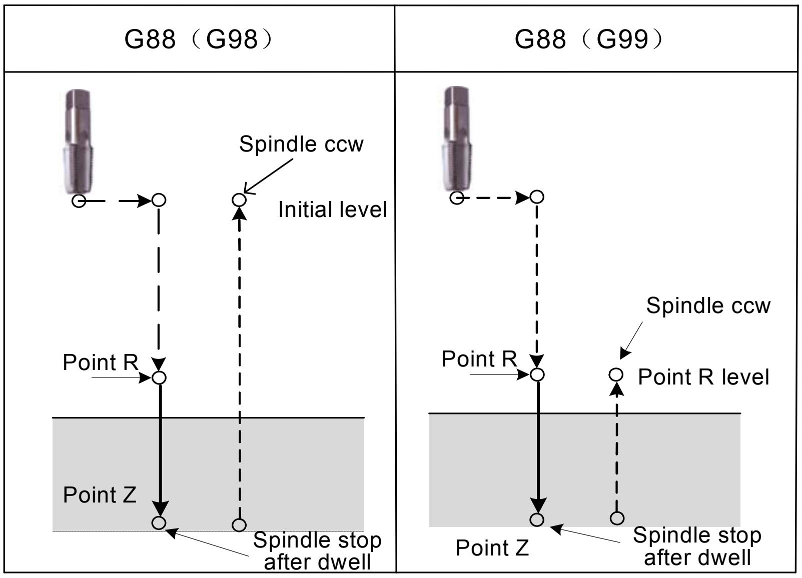


Fig. 3-29

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

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

When the G88 command and an M code are specified in the same block, the M code is executed at the time of the first hole positioning operation. The system then proceeds to the next drilling operation.

When K is used to specify the number of repeats, the M code is performed for the first hole only; for the second and subsequent holes, the M code is not executed.

When a tool length offset (G43, G44 or G49) is specified in the canned cycle, the offset is applied at the time of positioning to point R.

Axis switching: Before the boring axis can be changed, the canned cycle must be canceled.
Boring: In a block that does not contain X,Y,Z, R or any other axes, boring is not performed.

Examples:
M3 S2000; Cause the spindle to start rotating;
G90 G99 G88 X300. Y-250. Z-150. R-100. P1000 F120;
Position, bore hole 1, and then return to point R;
Y-550.; Position, bore hole 2, and then return to point R;
Y-750.; Position, bore hole 3, and then return to point R;
X1000.; Position, bore hole 4, and then return to point R;
Y-550.; Position, bore hole 5, and then return to point R;
G98 Y-750.; Position, bore hole 6, and then return to initial level;
G80 G28 G91 X0 Y0 Z0 ; Return to the reference position;
M5; Cause the spindle to stop rotating.

Restrictions:

Cancel: It's not allowed to specify a G code of group 01(G00, G01, G02, G03, G60 or G86) in the same block.
Tool offset: In the canned cycle mode, the tool offset is ignored.

3.4.13 Boring Cycle (G89)

Format: G89 X_ Y_ Z_ R_ P_ F_ K_;
Function: This cycle is used to bore a hole.

Explanation:

X_Y_:Hole position data.

Z_:The incremental programming means the distance from point R to the hole bottom; the absolute programming means the absolute coordinate value of the hole bottom.

R_:The incremental programming means the distance from the initial level to point R; the absolute programming means the absolute coordinate value of point R.

P_:Dwell time

F_:Cutting feedrate.

K_:Number of repeats.

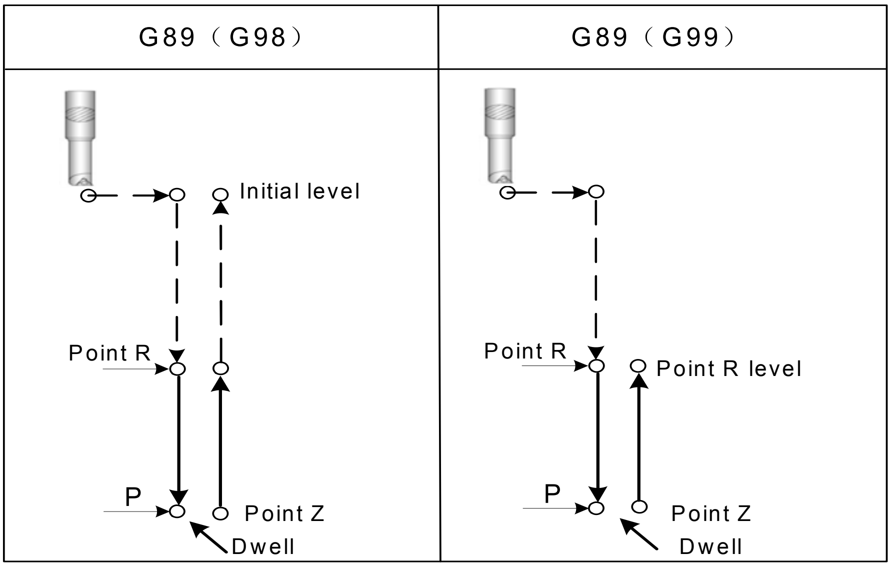


Fig. 3-30

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

Before specifying G89, use a miscellaneous function (M code) to rotate the spindle. When the G89 command and an M code are specified in the same block, the M code is executed at the time of the first hole positioning operation. The system then proceeds to the next drilling operation.

When K is used to specify the number of repeats, the M code is performed for the first hole only; for the second and subsequent holes, the M code is not executed.

When a tool length offset (G43, G44 or G49) is specified in the canned cycle, the offset is applied at the time of positioning to point R.

Axis switching: Before the boring axis can be changed, the canned cycle must be canceled.

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

Examples:

```
M3 S100;                                    Cause the spindle to start rotating;
    G90 G99 G89 X300. Y-250. Z-150. R-120. P1000 F120;
          Position, bore hole 1, return to point R, and then stop at the hole bottom for 1s;
    Y-550.;                      Position, bore hole 2, and return to point R;
    Y-750.;                      Position, bore hole 3, and return to point R;
    X1000.;                      Position, bore hole 4, and return to point R;
    Y-550.;                      Position, bore hole 5, and return to point R;
    G98 Y-750.;                  Position, bore hole 6, and return to initial level;
    G80;
    G28 G91 X0 Y0 Z0 ;                      Return to the reference position;
    M5;                                     Cause the spindle to stop rotating.
    M30;
```

Restrictions:
Cancel: It’s not allowed to specify a G code of group 01(G00, G01, G02, G03 or G60) in the same block.
Tool offset: In the canned cycle mode, the tool offset is ignored.

3.4.14 Left-handed Rigid Tapping Cycle（G74）

Format: G74 X_Y_Z_R_P_F_K_
Function: When the spindle motor is controlled in rigid tapping mode as if it were a servo motor, tapping cycles can be sped up in high precision.

Explanation:

X_Y_: Hole position data.

Z_:The incremental programming means the distance from point R to the hole bottom; the absolute programming means the absolute coordinate value of the hole bottom.

R_:The incremental programming means the distance from the initial level to point R; the absolute programming means the absolute coordinate value of point R.

P_:Dwell time

F_:Cutting feedrate.

K_:Number of repeats.

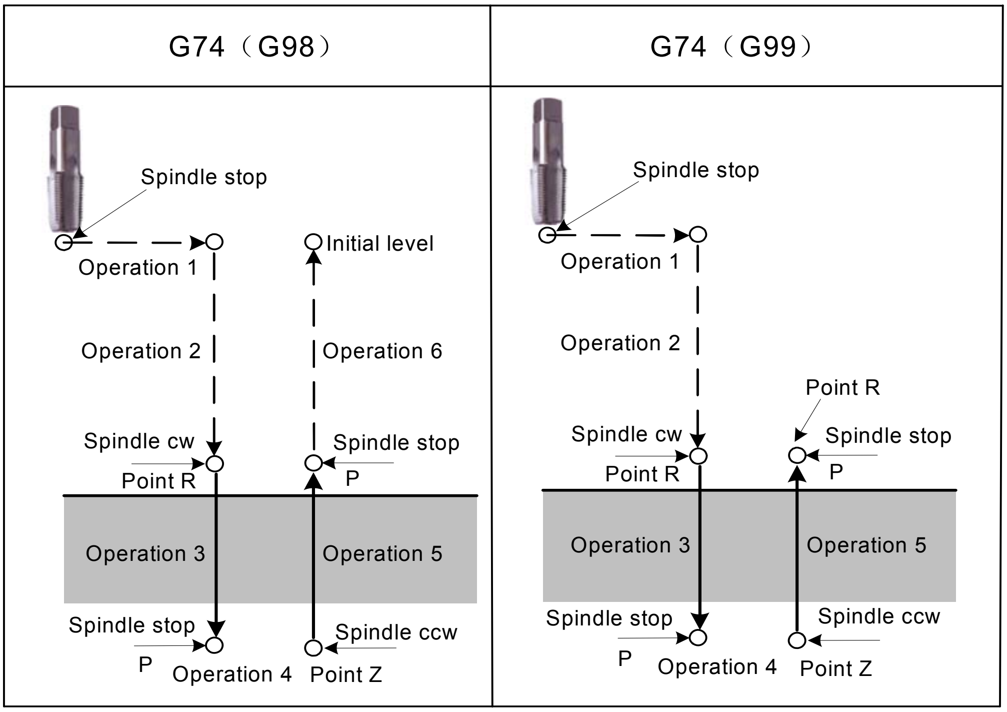


Fig. 3-31

After positioning along the X- and Y- axes, Z axis moves to point R in rapid traverse. Spindle starts CCW turning with G74, and tapping is executed from point R to Z. 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, then the spindle is stopped. Rapid traverse to the initial level is then performed. While tapping 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 command;

(2) Specify M29 S***** in a block which contains a tapping command.

When the G74 command and an M code are specified in the same block, the M code is executed at the time of the first hole positioning operation. The system then proceeds to the next drilling operation.

When K is used to specify the number of repeats, the M code is performed for the first hole only; for the second and subsequent holes, the M code is not executed.

When a tool length offset (G43, G44 or G49) is specified in the canned cycle, the offset is applied at the time of positioning to point R.

Axis switching: Before the tapping axis can be changed, the canned cycle must be canceled. If the tapping axis is changed in the rigid mode, the alarm is issued.

Specify S and an axial movement command between M29 and G84, the system alarms.

The formula of the thread lead: Feedrate/spindle speed

Z axis feedrate= the spindle speed * the thread lead.

Examples:

Spindle speed 1000rpm

Thread lead1.0mm

And then, Z axis feedrate= 1000*1=1000mm/min

```
 G00 X120 Y100;           Positioning
 M29   S1000              Specifying the rigid mode
G74 Z-100 R-20 F1000;     Rigid tapping
```

Restrictions:

F:It alarms if the specified F value exceeds the upper limit value of the cutting feedrate.

S: It alarms if the speed exceeds the maximum speed of the specified gear. The gear speed is set by the parameters **P2140~2142**.

Cancel: It's not allowed to specify a G code of group 01(G00, G01, G02, G03 or G60) in the same block.

Tool offset: The tool radius offset is retracted automatically after the canned cycle mode is performed till the offset is set automatically.

Restarting the program: It is invalid that the program is restarted during the rigid tapping.

3.4.15 Right-handed Rigid Tapping Cycle (G84)

Format: G84 X_Y_Z_R_P_F_K_

Function: When the spindle motor is controlled in rigid tapping mode as if it were a servo motor, tapping cycles can be sped up in high precision. And point R remains unchanged, the tapping starting positions are same, namely, repeatedly execute the tapping command in one position, the thread is in the normal state.

Explanation:

X_Y_:Hole position data.

Z_:The incremental programming means the distance from point R to the hole bottom; the absolute programming means the absolute coordinate value of the hole bottom.

R_:The incremental programming means the distance from the initial level to point R; the absolute programming means the absolute coordinate value of point R.

P_: Dwell time

F_: Cutting feedrate.

K_: Number of repeats.

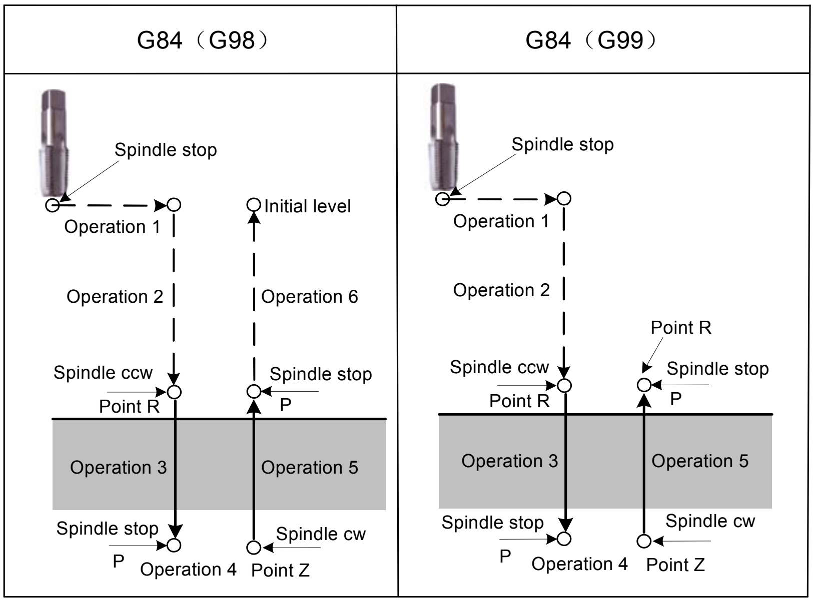


Fig. 3-32

After positioning along the X- and Y- axes, Z axis moves to point R in rapid traverse. Spindle starts CCW turning with G84, and tapping is executed from point R to Z. 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, then the spindle is stopped. Rapid traverse to the initial level is then performed.

While tapping 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 specifying a tapping command;
(2) Specify M29 S***** in a block with a tapping command.
When the G84 command and an M code are specified in the same block, the M code is executed at the time of the first hole positioning operation. The system then proceeds to the next drilling operation.

When K is used to specify the number of repeats, the M code is performed for the first hole only; for the second and subsequent holes, the M code is not executed.

When a tool length offset (G43, G44 or G49) is specified in the canned cycle, the offset is applied at the time of positioning to point R.

Axis switching: Before the tapping axis can be changed, the canned cycle must be canceled. If the tapping axis is changed in the rigid mode, the alarm is issued.

If S and the axis movement command are specified between M29 and G84, the system alarms. If M29 is specified in M29, the system alarms.

In the feeding/min mode, the formula of the thread lead: Feedrate/spindle speed.
Z axis feedrate= the spindle speed * the thread lead

Examples:
Spindle speed: 1000r/min
Thread lead: 1.0mm
Then, Z axis feedrate=1000*1=1000mm/min
G00 X120 Y100; Positioning
M29 S1000 Specifying the rigid mode
G84 Z-100 R-20 F1000; Rigid tapping

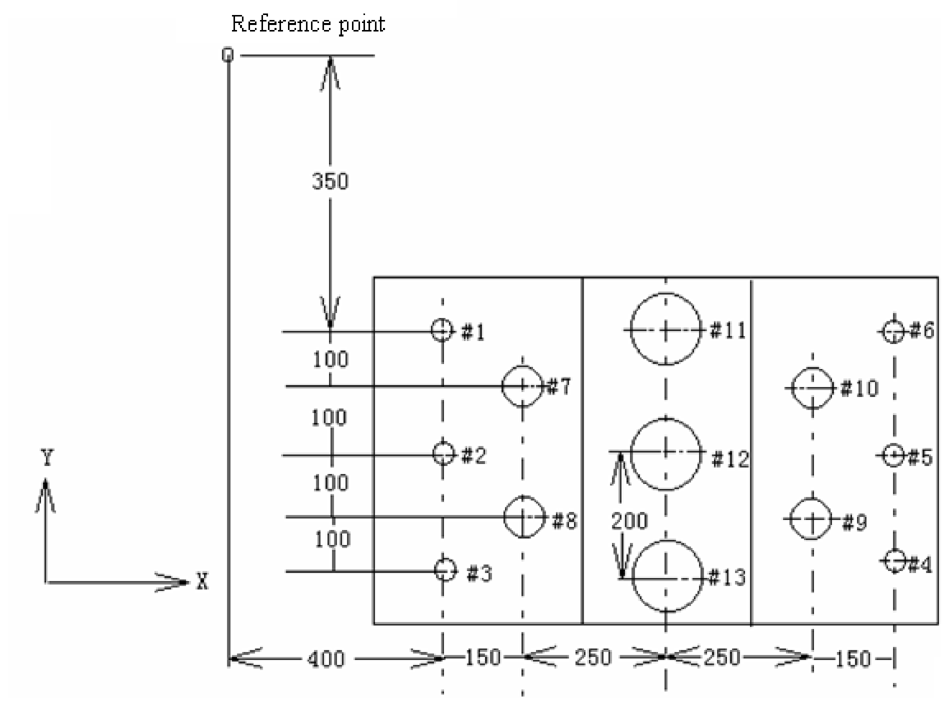
Restrictions:
F: It alarms if the specified F value exceeds the upper limit value of the cutting feedrate.
S: It alarms if the speed exceeds the maximum speed of the specified gear. The speed gear is set by the data parameter **P2140~2142.**

Cancel: G00, G01,G02 or G60 of group 01 can't be specified in one block.

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

Restarting the program: It is invalid that the program is restarted during the rigid tapping.

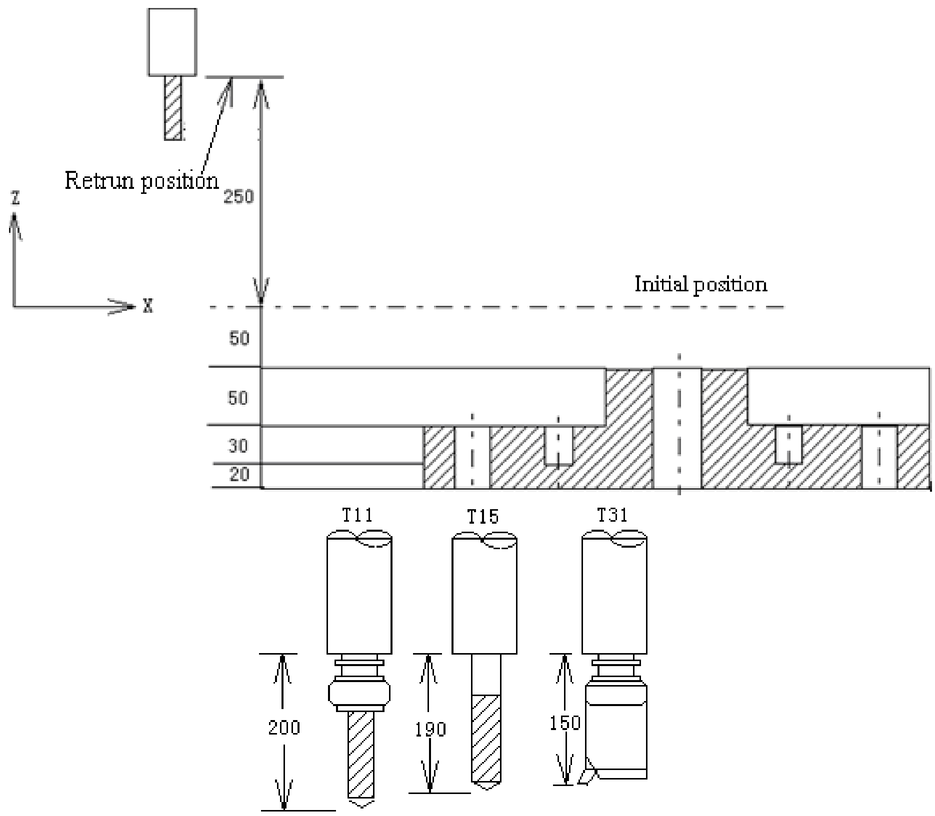
Examples;
Introduce the canned cycle through the tool length compensation.



 # 1～ 6... Drill Φ10 hole

 # 7～10... Drill Φ20 hole

#11～13.. BoreΦ95 hole



The value of offset number 11 is 200, the value of the offset number 15 is 190, the value of offset number 31 is 150, which are set as the offset value. The programs are as below:

N001 G92 X0 Y0 Z0 ;	The coordinate system is set on the reference position.
N002 G90 G00 Z250 T11 M6 ;	The tool is changed.
N003 G43 Z0 H11 ;	Compensate the plane tool length in the initialized position.
N004 S300 M3 ;	The spindle is started.
N005 G99 G81 X400 Y-350 ; Z-153 R-97 F120 ;	Machine #1 hole after positioning.
N006 Y-550 ;	Machine #2 hole after positioning, and return to point R level.
N007 G98 Y-750 ;	Machine #3 hole after positioning, return to the initial level.
N008 G99 X1200 ;	Machine #4 hole after positioning, and return to point R level.
N009 Y-550 ;	Machine #5 hole after positioning, and return to point R level.
N010 G98 Y-350 ;	Machine #6 hole after positioning, and return to the initial level.
N011 G00 X0 Y0 M5 ;	Return to the reference position and the spindle stops.
N012 G49 Z250 T15 M6 ;	Cancel the tool length compensation and the tool is changed.
N013 G43 Z0 H15 ;	Compensate the tool length in the initial level.
N014 S200 M3 ;	The spindle starts.
N015 G99 G82 X550 Y-450 ; Z-130 R-97 P30 F70 ;	Machine #7 hole after positioning, and return to point R level.
N016 G98 Y-650 ;	Machine #8 hole after positioning, and return to the initial level.
N017 G99 X1050 ;	Machine #9 hole after positioning, and return to point R level.
N018 G98 Y-450 ;	Machine #10 hole after positioning, and return to the initial level.
N019 G00 X0 Y0 M5 ;	Return to the reference position and the spindle stops.
N020 G49 Z250 T31 M6 ;	Cancel the tool length compensation and the tool is changed.
N021 G43 Z0 H31 ;	Compensate the tool length on the initial level.
N022 S100 M3 ;	The spindle starts.
N023 G85 G99 X800 Y-350 ; Z-153 R47 F50 ;	Machine #11 hole after positioning, and return to point R level.
N024 G91 Y-200 ; Y-200 ;	Machine #12 and #13 holes after positioning, and return to point R level.
N025 G00 G90 X0 Y0 M5 ;	Return to the reference position, the spindle stops.
N026 G49 Z0 ;	Cancel the tool length compensation.
N027 M30 ;	The program stops.

3.4.16 Rough of the Groove in the Circle (G110/G111)

Format:

G110
G98/G99 **X_ Y_ R_ Z_ I_ L_ W_ Q_ V_ D_ F_ K_**
G111

Command function: Start from the center of the circle, execute the arc interpolation for many time in the helical mode till the round groove of the programmed dimension is processed.

Command introduction:

G110: CCW rough of the groove in the circle

G111: CW rough of the groove in the circle

I:The groove radius in the circle, I should be more than the radius of the current tool;

L:The cut width increment on XY planes is less than the tool diameter but more than 0;

W: Feed in Z axis direction for the first time, the distance below R datum surface should be more than 0. If the feeding exceeds the groove bottom, directly process at the groove bottom.

Q:The feeding increment of each time in Z axis direction;

V:When cut rapidly, the distance from the unprocessed face is more than 0;

D:The span of the tool radius serial number is 0 ~ 256, D0 is defaulted as 0. Based on the specified serial number, the current tool radius value is set.

K:Number of repeats.

Cycle process:

⑴. Rapidly position on XY plane;

⑵ Rapidly cut into point R level;

⑶ Cut down for the distance of W in the cutting speed;

⑷ From the center, helical mill the rectangular section inside out based on L value and L value increases each time.

⑸ Z axis rapidly returns to point R level;

⑹ X and Y axes rapidly positions to the center of a circle;

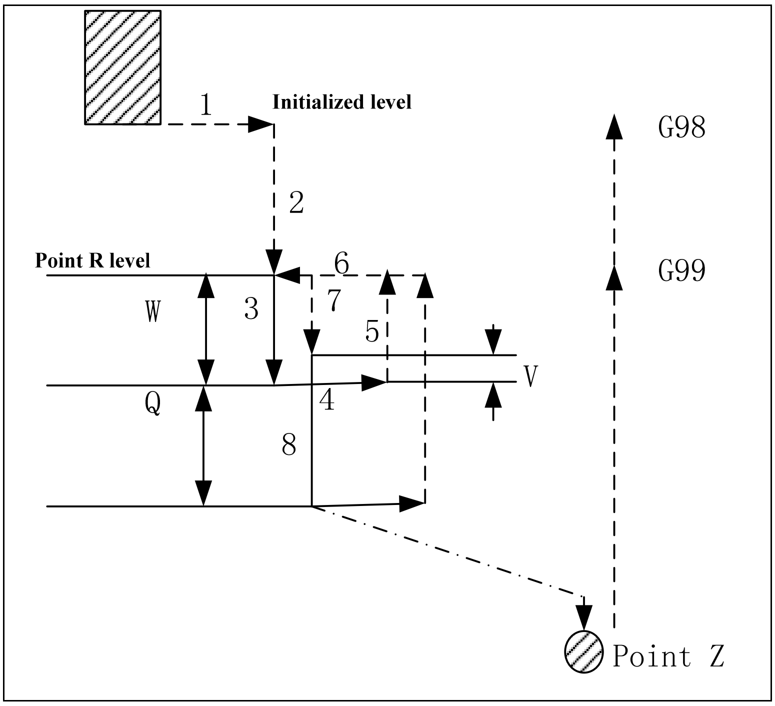
⑺ Z axis rapidly moves downward the position which keeps distance V from the unprocessed face;

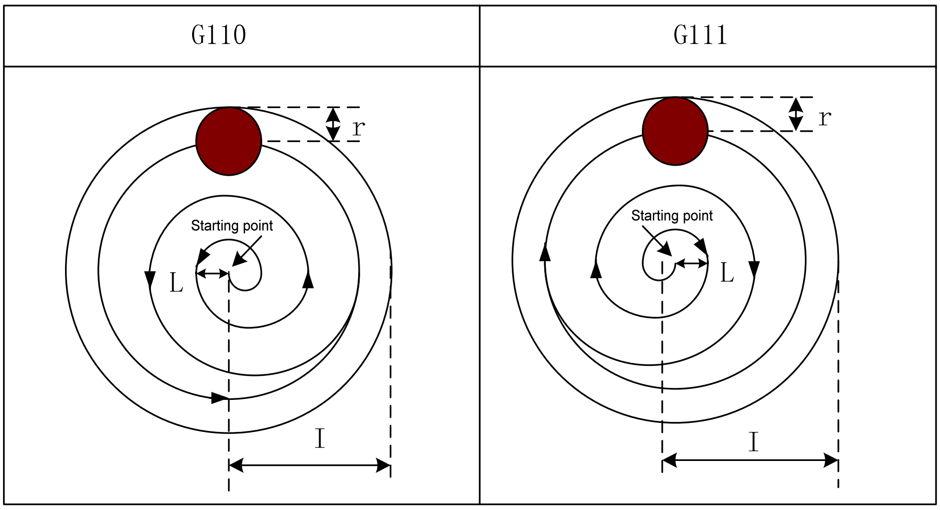
⑻ Z axis cuts down for the depth（Q+V）;

⑼ Cycle the steps of（4）~（8）till process the round surface of the total cutting depth;

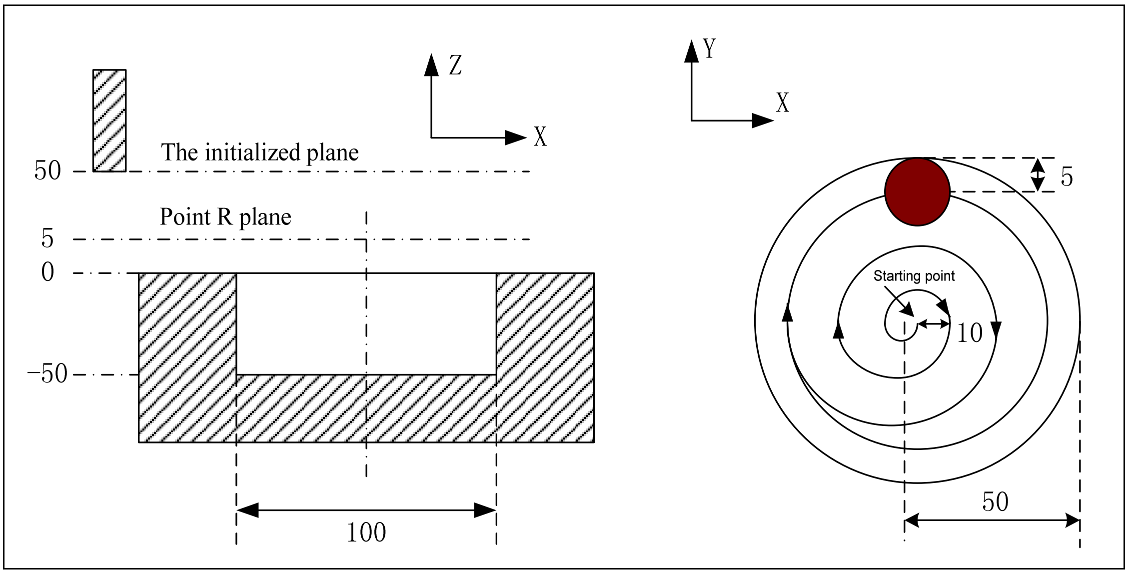
⑽ Base on G98 or G99, return to the initial level or point R level.

Commanded path:





For example: The canned cycle G111 commands rough milling the groove in the circle, which is shown as the following figure:



G90 G00 X50 Y50 Z50; （G00 rapid position）

G99 G111 X25 Y25 R5 Z-50 I50 W20 Q10 L10 V10 F800 D1; （D1=5 Roughing the groove in the circle in cycle）

G80 X50 Y50 Z50; （Cancel the canned cycle, return from point R level）

M30;

3.4.17 Finishing the Whole Circle Cycle(G112/G113)

Command formula:

G112
G98/G99 **X_ Y_ R_ Z_ I_ J_ D_ F_ K_**
G113

Command function: The tool finish mills one whole circle based on the specified radius value I and the direction in the circle, and the tool returns after complete the finish milling.

Command introduction explanation:

Finish milling cycle in the full circle in CCW direction.

Finish milling cycle in the full circle in CW direction.

I:Finish milling the circle radius, the span solution range is 0 mm ~9999.999mm.When it is the negative value, the absolute value is taken.

J:The distance between the finish milling starting position and the finish milling circle center, the span is 0 mm ~9999.999mm. When it is the negative value, the absolute value is taken.

D:The tool radius serial number, the span is 0 ~ 256, D0 is defaulted as 0. Based on the specified serial number, the current tool radius value is taken.

K:Repeated times.

The cycle process:

(1). Rapidly position on X and Y planes;

(2)Rapidly cut down and into point R plane;

(3)Cut into the hole bottom;

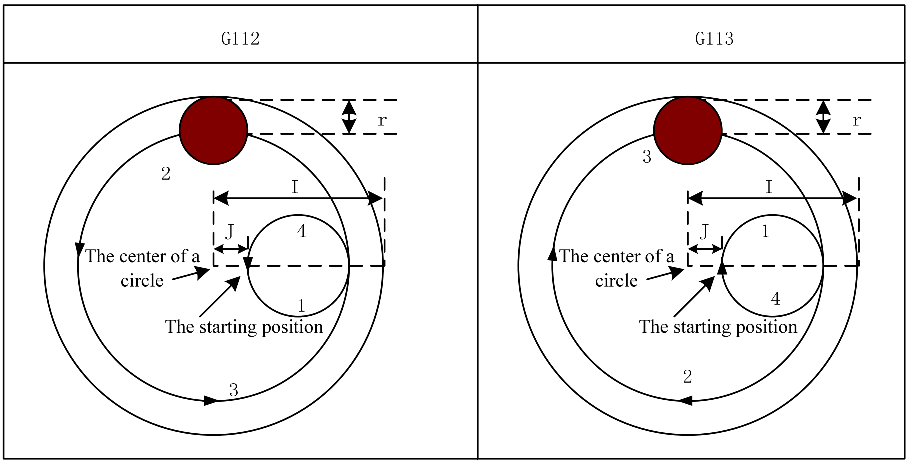
(4)The arc interpolation based on the path of the transition arc 1;

(5)The whole circle interpolation based on the path of the transition arcs 2 and 3.

(6)The arc interpolation and returning the starting position based on the path of the transition arc 4;

⑺Based on G98 or G99, return to the initialized position plane or point R plane.

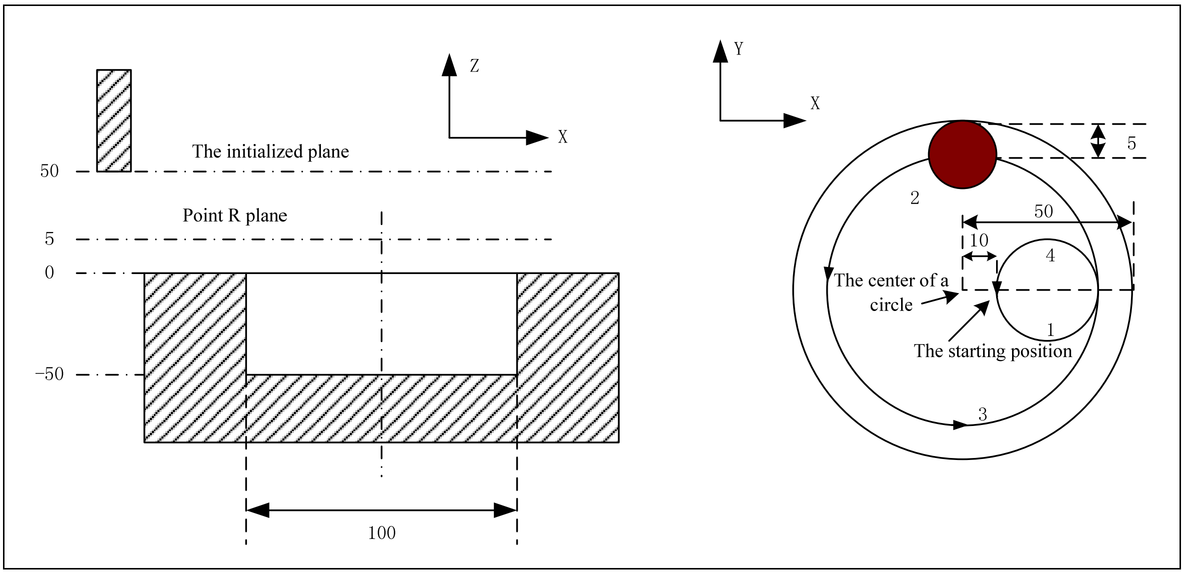
Command path:



Relative introduction:

In the cycle, it is invalid to command Q, P and L, but the values of Q and P are saved as the canned cycle mode values.

For example: The canned cycle G112 commands the finish milling the round groove which has already been rough milled, which is shown as the following figure.



G90 G00 X50 Y50 Z50; （G00 rapid positioning）

G99 G112 X25 Y25 R5 Z-50 I50 J10 F800 D1; （The canned cycle starts, and the tool cuts into the hole bottom for the finish milling in the circle in cycle D1=5）

G80 X50 Y50 Z50; （Cancel the canned cycle and return from point R plane）

M30;

3.4.18 Protruding Roughing Outside of the Circle (G114/G115)

Command formula:

G114
G98/G99 **X_ Y_ R_ Z_ I_ J_ L_ W_ Q_ V_ D_ F_ K_**
G115

Command function: Start from the circle outside, and the arc interpolates in the spiral mode for many times until the work piece outside the circle is processed into the programming dimension.

Command remark:

G114: Protruding rough milling CCW outside of the circle

G115: Protruding rough milling CW outside of the circle

I: Process the circle radius;

J: The radius outside of the circle Radius of external circle

L: The cut width increment of the circle outside external circle should be less than the tool radius and more than 0;

W:; Cut down in Z axis direction for the first time and the distance from R base level should be more than 0;

Q: The cutting depth increment each time in Z axis direction;

V: When the tool rapidly cuts, the distance far away from the unprocessed should be more than 0;

D: The tool radius serial number, the span solution range is 0 ~ 256, D0 defaults as 0. Based on the specified serial number, take the current tool radius value.

K: Repeated times;

Cycle process:

⑴ Rapidly position on the plane of the circle outside;

⑵ Rapidly cut down into the plane of point R;

⑶ Cut down for distance W in the cutting speed;

⑷ From the starting position of the circle outside, helical mill the circles inside from radius J to L based on L value and L value is increased each time.

⑸ Z axis rapidly returns to R base level;

⑹ X and Y axes rapidly position at the starting position of the circle outside external circle;

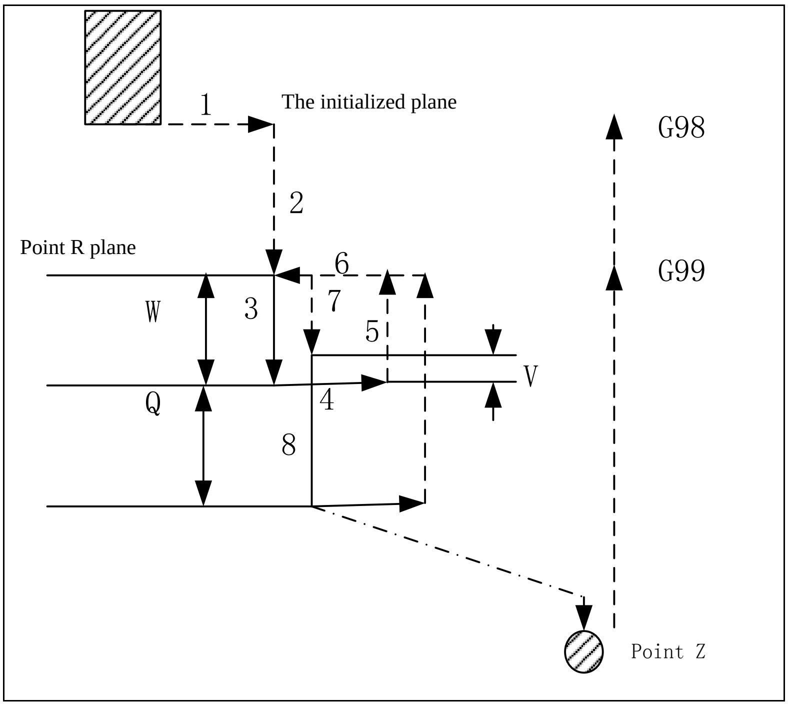
⑺ Z axes rapidly moves downward the position which keeps distance V from the unprocessed the end of machining face;

⑻ Z axis cut down for distance（Q+V）;

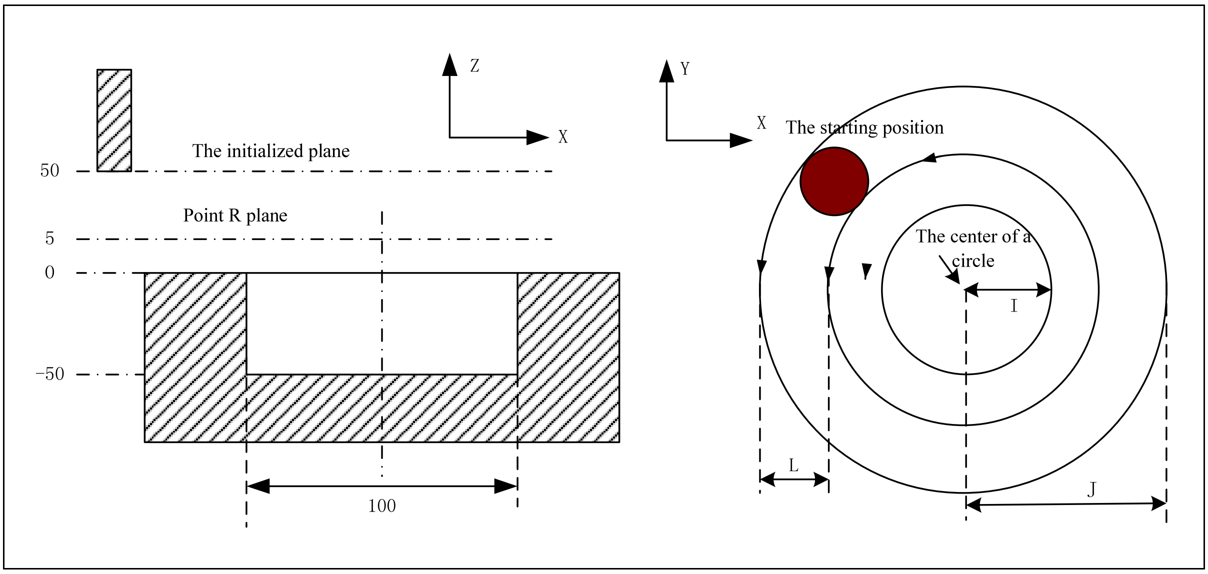
⑼ Cycle the movements （4）~（8） until the circle outside of the total cutting depth is processed;

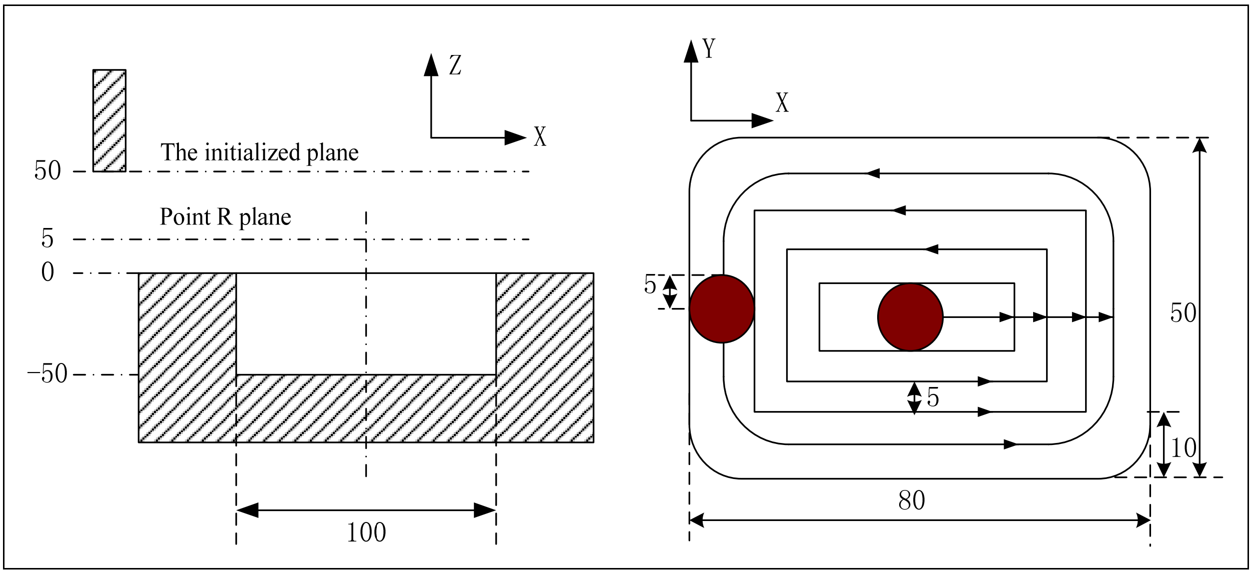
⑽ Based on the difference between G98 or G99, return to the initialized position plane or point R plane.

Command path:



For example: The canned cycle G114 commands rough milling one groove inside the circle, which is shown as the following figure:





G90 G00 X50 Y50 Z50; （G00 rapid position）

G99 G114 X25 Y25 R5 Z-50 I50 J100 W20 Q10 L10 V10 F800 D1; （Protruding rough milling cycle outside of the circle D1=5）

G80 X50 Y50 Z50; （Cancel the canned cycle, return from point R plane）

M30;

3.4.19 Outside of the Circle of External Circle (G116/G117)

Command formula:

G116
G98/G99 **X_ Y_ R_ Z_ I_ J_ D_ F_ K_;**
G117

Command function: The tool finish mills one full circle in the specified radius value and direction outside of the circle, and it returns after the finish milling completes.

Command introduction:

G116: CCW finish milling cycle outside of the circle

G117: CW finish milling cycle outside of the circle

I: The span of the finish milling circle radius is 0 mm ~9999.999mm, and the absolute value is taken when it is the negative value.

J: The span of the distance between the finish milling starting position and the finish milling circle is 0 mm ~9999.999mm, and the absolute value is taken when it is the negative value.

D: The span of the tool radius serial number is 0 ~ 256, D0 defaults as 0. Based on the specified serial number, the current tool radius value is taken.

K: Repeated times.

Cycle processing:

⑴ Rapidly position on planes of X and Y.

⑵ Rapidly cut down into point R plane;

⑶ Cutting feed at the bottom of hole;

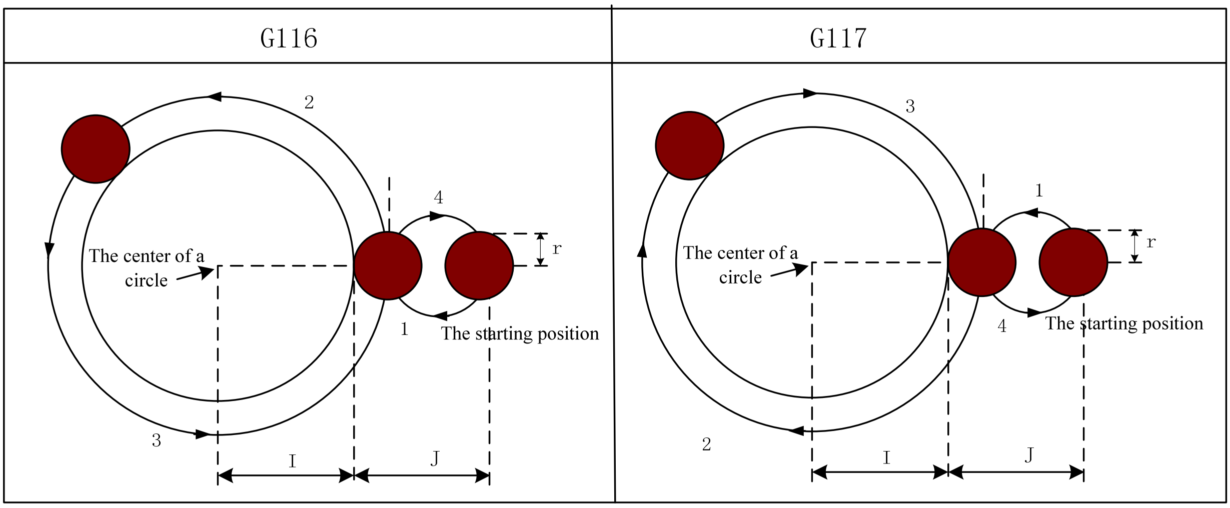
⑷ The arc interpolation takes the transition arc 1 as the path;

(5) The full circle interpolation take the transition arc 2 or 3 as the path;

(6) The arc interpolation takes the transition arc 4 as the path and returns to the starting position;

(7) Based on the specified G98 or G99, return to the initialized position plane or point R plane.

Command path:

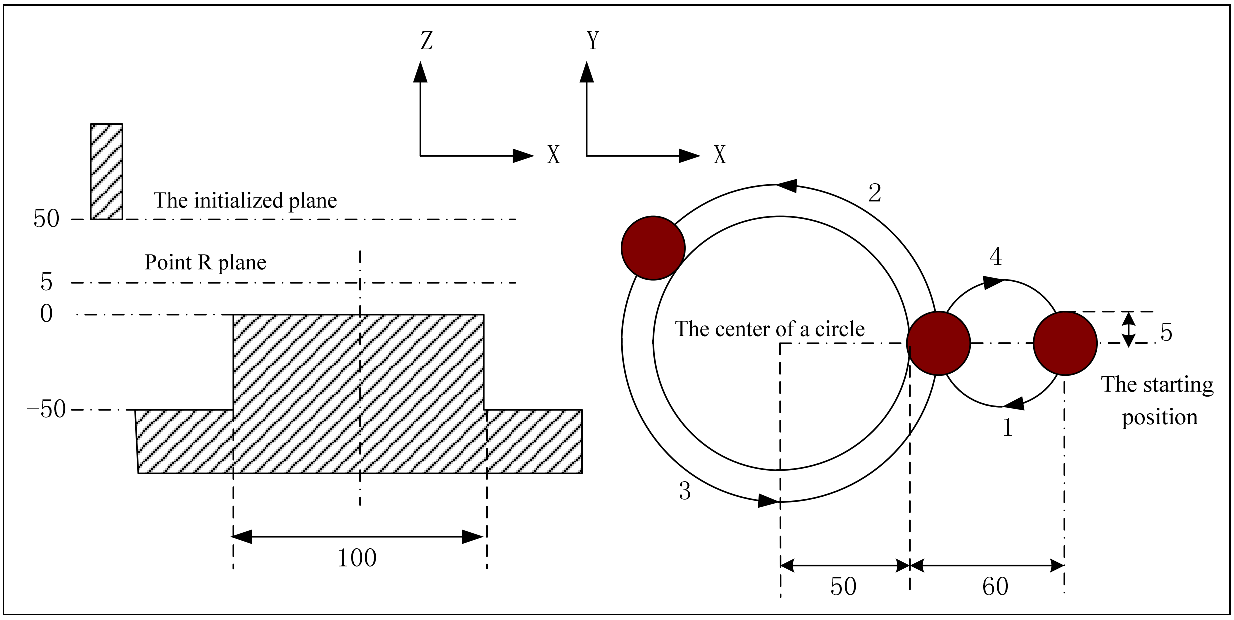


Relative introduction:

(1) During the finish milling outside of the circle, the interpolation directions of the transition arc and the finish milling arc are different When a finishing of external circle is performed, but the interpolation direction in the command introduction means that of the finish milling arc.

(2) In the cycle, commands of Q, P and L are invalid, while the values of Q and P are saved as the mode numerical values in the canned cycle.

For example: The canned cycle G116 commands the finish milling the round groove which has already been rough milled, which is shown as below:



G90 G00 X50 Y50 Z50; （G00 rapid position）

G99 G116 X25 Y25 R5 Z-50 I50 J60 F800 D1; （D1=5 The canned cycle starts and it cuts into the hole bottom for the finish milling cycle outside of the circle）

G80 X50 Y50 Z50; （Cancel the canned cycle, and return from point R plane）

M30;

3.4.20 Roughing Rectangle Groove (G130/G131)

Command formula:

G130
G98/G99 **X_ Y_ Z_ R_ I_ J_ L_ W_ Q_ V_ U_ D_ F_ K_**
G131

Command function: Start from the rectangular center, straightway cut cycle based on the specified parameter data until the rectangular groove of the programming dimension is processed.

Command introduction:

G130: CCW rough milling the rectangular groove

G131: CW rough milling the rectangular groove

I: The width of the rectangular groove in X axis direction

J: The width of the rectangular groove in Y axis direction

L: The cut width increment on X or Y plane should be less than the tool diameter and more than 0.

W: Cut down in Z axis direction for the first time and the distance from R base level should be more than 0; (If the cutting depth for the first time exceeds the groove bottom, directly process at the groove bottom.)

Q: The cutting depth increment each time in Z axis direction

V: When rapidly cut, the distance between the unprocessed face should be more than 0;

U: If the corner arc radius is omitted, it means there isn’t any corner arc transition.

D: The span of the tool radius serial number is 0 ~ 256 and D0 defaults as 0. Based on the specified serial number, the current tool radius value is taken.

K: The repeated times.

Cycle processing:

⑴ Rapidly position on the plane of X or Y;

⑵ Rapidly move downward point R plane;

⑶ Cut down for distance W in the cutting speed

⑷ From the center, helical mill the rectangular section inside out based on L value and L value increases each time.

⑸ Z axis rapidly returns to R base level;

⑹ X and Y axes rapidly positions into the rectangular center;

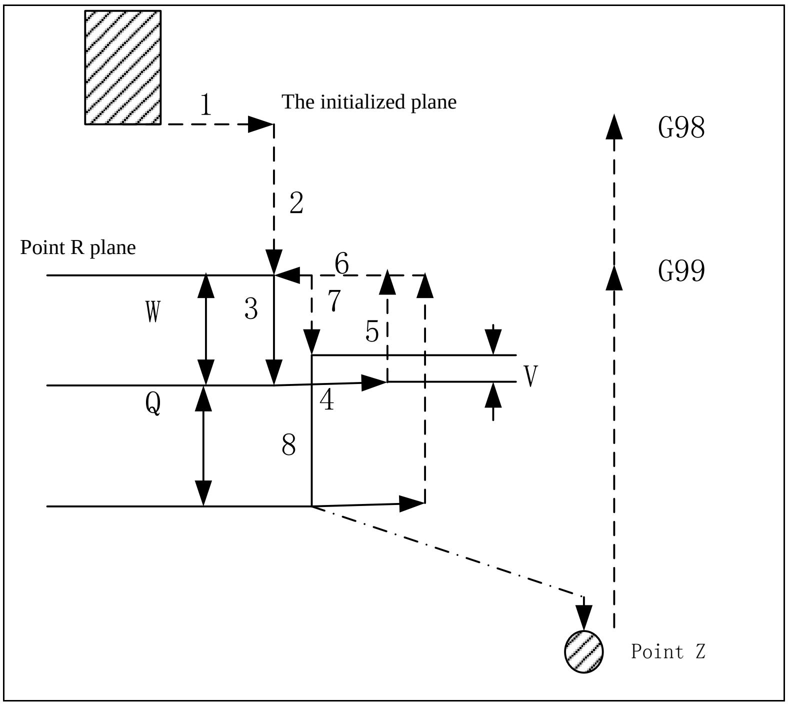
⑺ Z axis rapidly move downward the position which keeps distance Z from the unfinished processed face;

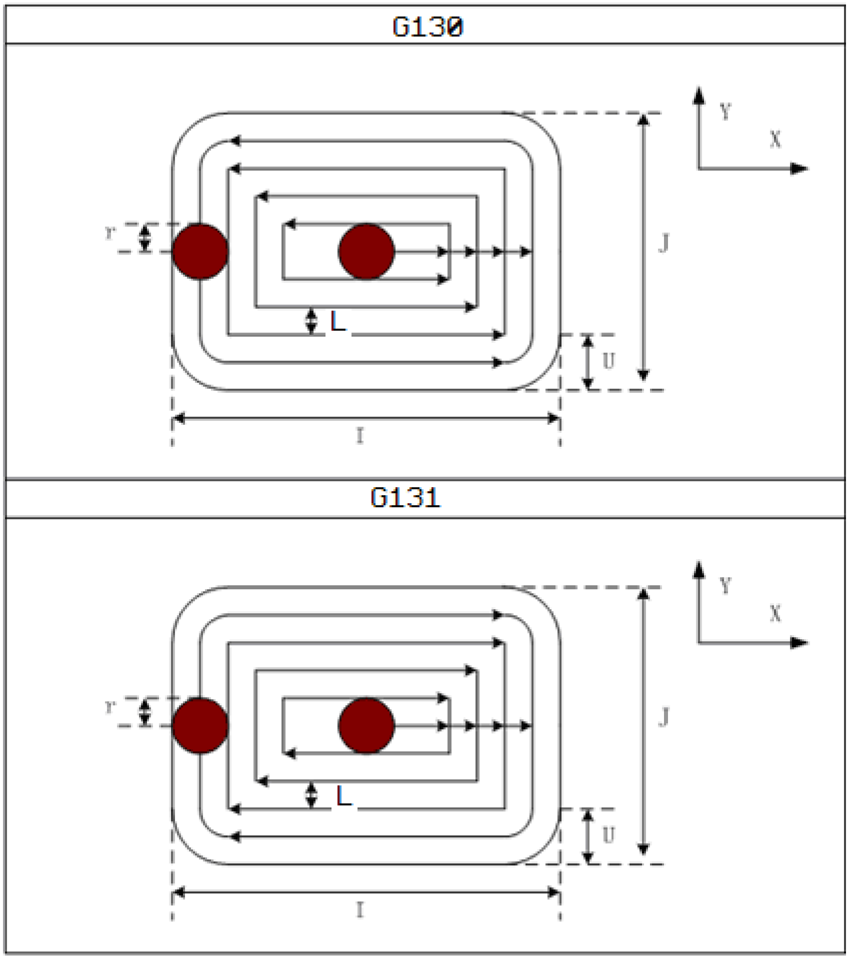
⑻ Z axis cuts down for distance（Q+V）

⑼ Cycle the movements（4）~（8） till the circle face of the total cutting depth is processed.

⑽ Based on the specified G98 or G99, return to the initialized position plane or point R plane.

Command path:

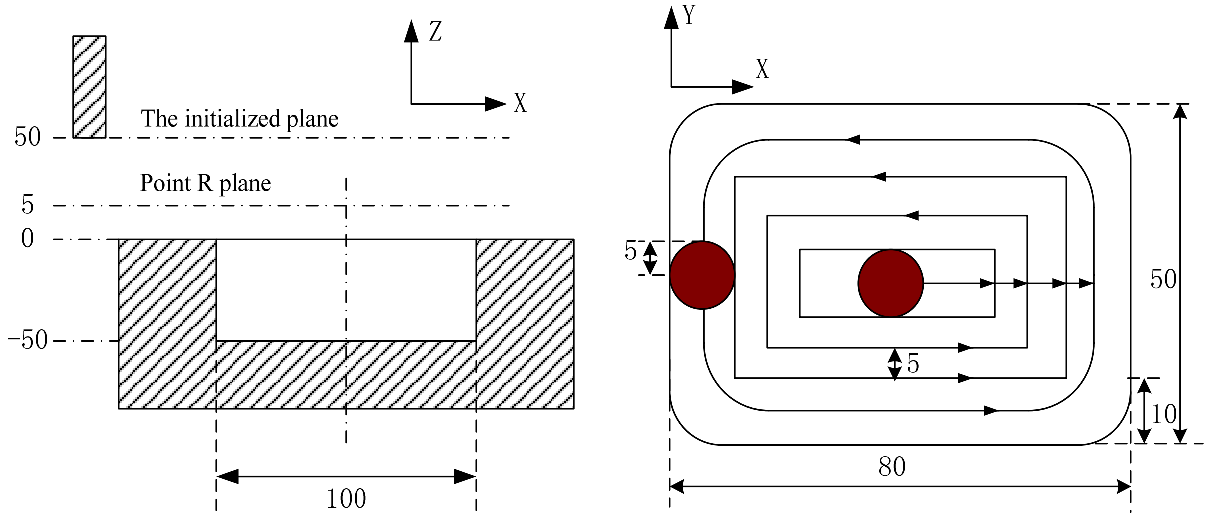




Relative introduction:

In the cycle, commands P and L are invalid, while value P is saved as the mode numerical value in the canned cycle.

For example: The canned cycle G130 commands rough milling one rectangular groove, which is shown as the following figure:



The initialized plane: Point R plane.

G90 G00 X50 Y50 Z50; （G00 rapidly position）

G99 G130 X25 Y25 R5 Z-50 I70 J50 W20 Q10 L5 V10 U10 F800 D1; （ Rough milling the groove cycle in the rectangle D1=5）

G80 X50 Y50 Z50; （Cancel the canned cycle and return from point R plane）

M30;

3.4.21 Finishing Cycle in the Rectangular Groove (G132/G133)

Command formula:

G132

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

G133

Command function: The tool finish mills in the specified width and direction in the rectangle and returns after the finish milling completes.

Command introduction:

G132: CCW finish milling cycle in the rectangle.

G133: CW finish milling cycle in the rectangle.

I: The span of the rectangle width in X direction is 0 mm ~9999.999mm.

J: The span of the rectangle width in Y direction is 0 mm ~9999.999mm.

D: The span of the tool radius serial number is 0 ~ 256, D0 defaults as 0. Based on the specified serial number, the current tool radius value is taken.

L: The span of the distance between the finish milling starting position and the rectangle side in X axis is 0 mm ~9999.999mm.

U: If the corner arc radius is omitted, it means there isn't any corner arc transition. When U is omitted or U=0 and the tool radius >0, it alarms.

K: Repeated times.

Cycle processing:

⑴ Rapidly position on X or Y plane;

⑵ Rapidly cut downward point R plane;

⑶ Cut and feed into the hole bottom;

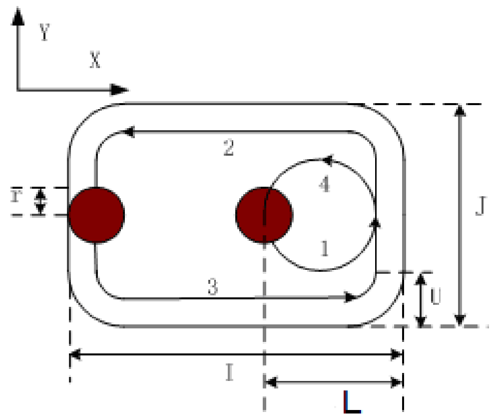
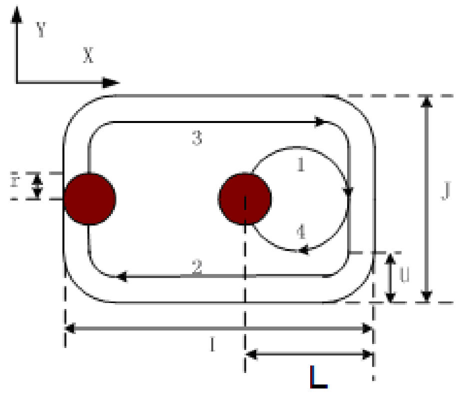
⑷ From the starting position, the arc interpolation takes the transition arc 1 as the path;

⑸ The linear and arc interpolation take the transition arc 2-3-4-5-6 as the paths;

⑹ The arc interpolation takes the transition arc 1 as the path and returns to the starting position;

⑺ Based on G98 or G99, return to the initialization position plane or point R plane.

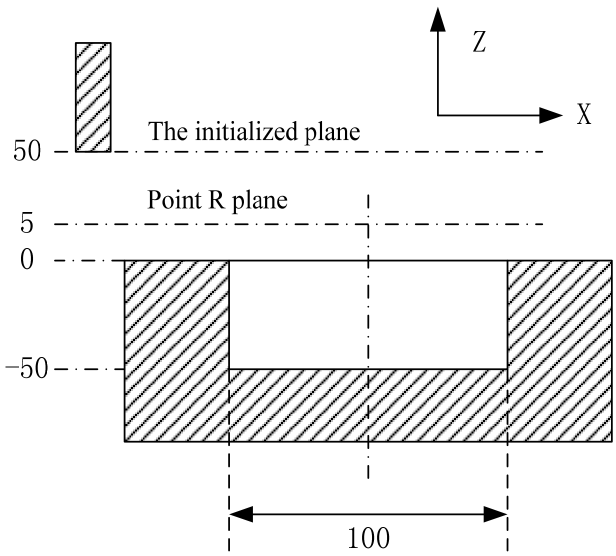
Command path:

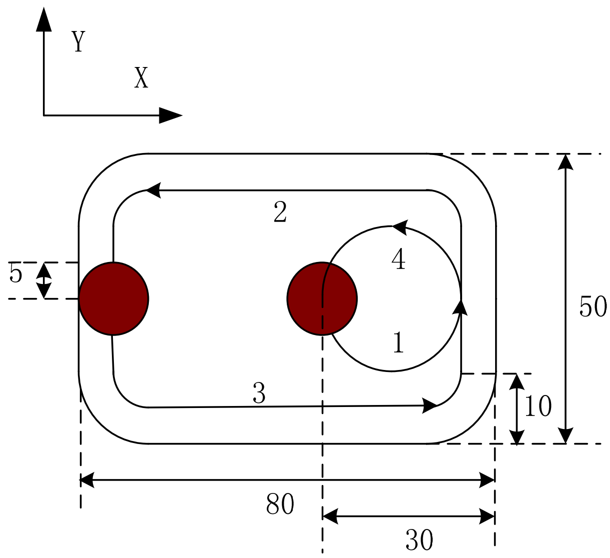
G132	G133
	

Relative introduction:

In the cycle, the commands Q, P and L are invalid, the values of Q and P are saved as the mode numerical value in the canned cycle.

For example: The canned cycle G132 commands finish milling the round groove which has already been rough finished, which is shown as below:





G90 G00 X50 Y50 Z50; （G00 rapid position）

G132 X25 Y25 R5 Z-50 I80 J50 L30 U10 F800 D1; （ In the canned cycle, finish mill in the rectangular groove at the hole bottom D1=5）

G80 X50 Y50 Z50; （The canned cycle is canceled and the tool returns to point R plane）

M30;

3.4.22 Roughing Cycle Outside of the Rectangle (G134/G135)

Command formula:

G134

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

G135

Command function: The tool rough mills in the specified width and direction and returns after the rough milling completes.

Command introduction:

G134: CCW rough milling cycle outside of the rectangle.

G135:CW rough milling cycle outside of the rectangle.

I: The span of the rectangle width in X axis is 0 mm ~9999.999mm.

J: The span of the rectangle width in Y axis is 0 mm ~9999.999mm.

D: The span of the tool radius serial number is 0 ~ 256 and D0 defaults as 0. Based on the specified serial number, the current tool radius value is taken.

L: The starting position of the rough milling, the cutting width increment should be less than the tool diameter and more than 0.

U: If the corner arc radius is omitted, it means there isn’t any corner arc transition.

Ｅ: The times of I and J should be more than 1.

K: The repeated times.

W: When cut down in Z axis for the first time, the distance below R base level should be more than 0.

Q: The cutting depth increment each time in Z axis.

V: When rapidly cut, the distance from the unprocessed face should be more than 0.

Cycle processing:

⑴ Rapidly position on plane of X or Y;

⑵ Rapidly move downward point R plane;

⑶ Cut downward for distance W in the cutting speed

⑷ Helical mill the rectangular face outside-in based on L value and L value is increased each time.

⑸ Z axis rapidly returns to R base level.

⑹ X and Y axes rapidly position on the rectangle center

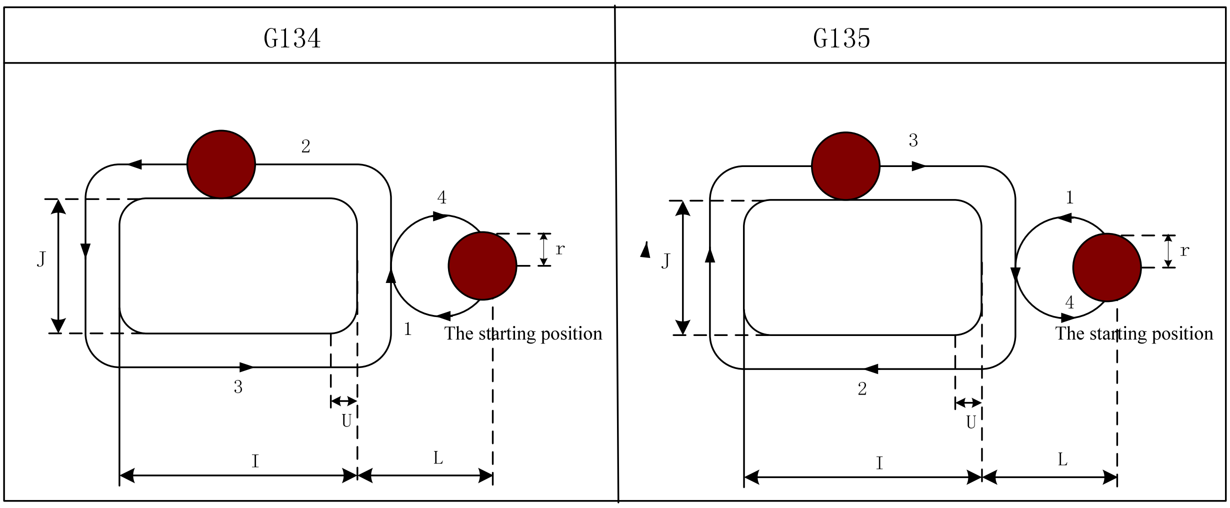
⑺ Z axis rapidly moves downward the distance V from the unprocessed face

⑻ Z axis cuts downward for distance（Q+V）

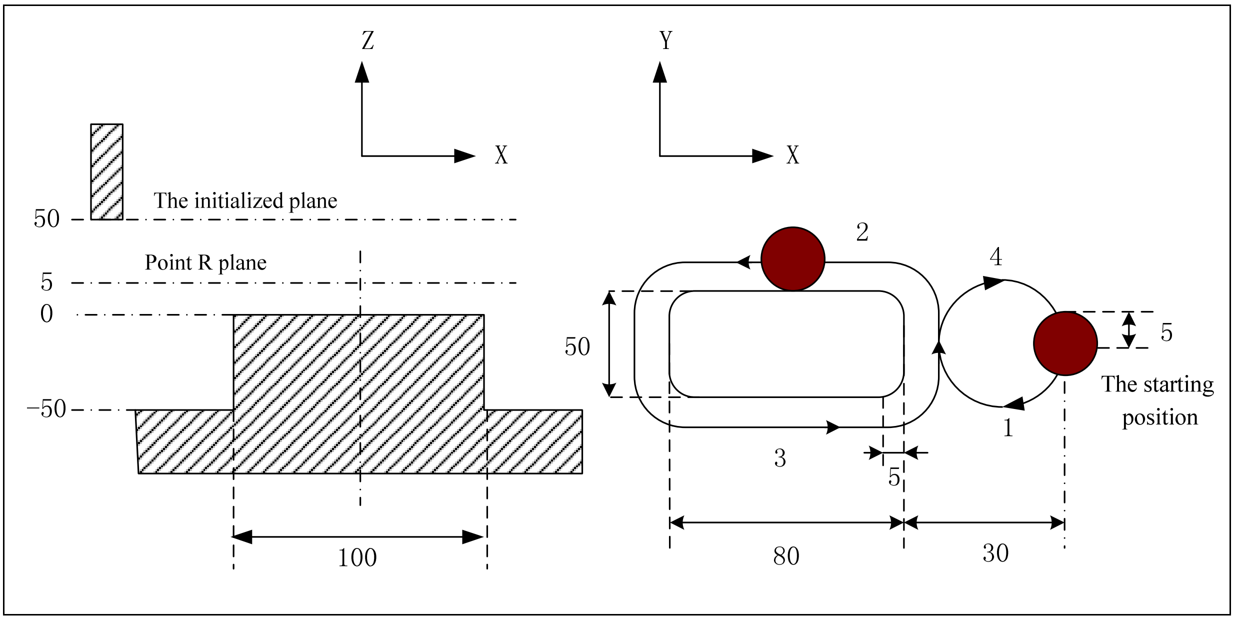
⑼ Cycle the movements（4）~（8）till the circle face of total cutting depth is processed.

⑽ Based on the specified G98 or G99, return to the initialized position plane or point R plane.

Command path:



1.For example: The canned cycle G134 commands rough milling the rectangular section, which is shown as below.



G90 G00 X50 Y50 Z50; （G00 rapidly position）

G99 G134 X25 Y25 R5 Z-50 I80 J50 W10 Q5 V10 L2 E3 U5 F800 D1; (D1=5)

G80 X50 Y50 Z50; （The canned cycle is canceled and the tool returns from point R plane）

M30;

3.4.23 Finishing Cycle outside of the Rectangle (G136/G137)

Command formula:

G136
G98/G99 **X_ Y_ R_ Z_ I_ J_ D_ L_ U_ F_ K_**
G137

Command function: The tool finish mills in the specified width and direction outside of the rectangle, and it returns after the finish milling completes.

Command introduction:

G136: CCW finish milling cycle outside of the rectangle.

G137: CW finish milling cycle outside of the rectangle.

I: The span of the rectangle width in X axis is 0 mm ~9999.999mm.

J: The span of the rectangle width in Y axis is 0 mm ~9999.999mm.

D: The span of the tool radius serial number is 0 ~ 256, D0 defaults as 0. Based on the specified serial number, the current tool radius value is taken.

L: The span of the distance between the finish milling starting position and the rectangle side in X axis is 0 mm ~9999.999mm.

U: If the corner arc radius is omitted, it means there isn't any corner arc transition.

K: The repeated times.

Cycle processing:

(1) Rapidly position on the plane of X or Y;

(2) Rapidly move downward point R plane;

(3) Cut and feed into the hole bottom;

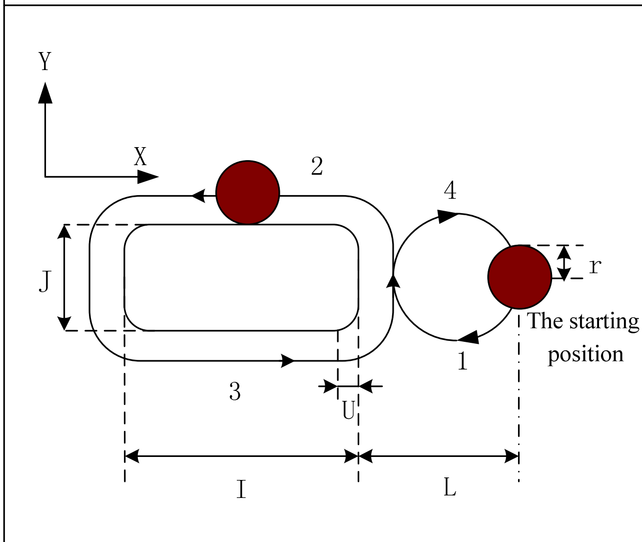
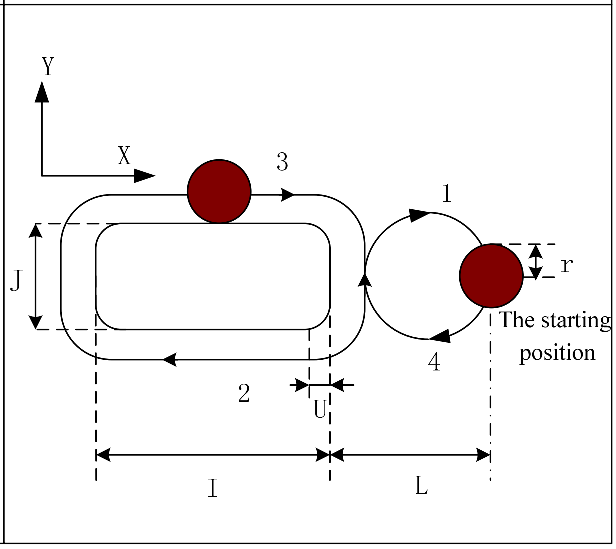
(4) From the starting position, the arc interpolation takes the transition arc 1 as the path;

(5) The linear and the arc interpolation take the transition arc 2-3-4-5-6 as the path;

(6) The arc interpolation takes the transition arc 7 as the path;

(7) Based on the specified G98 or G99, return to the initialization position plane or point R plane.

Command path:

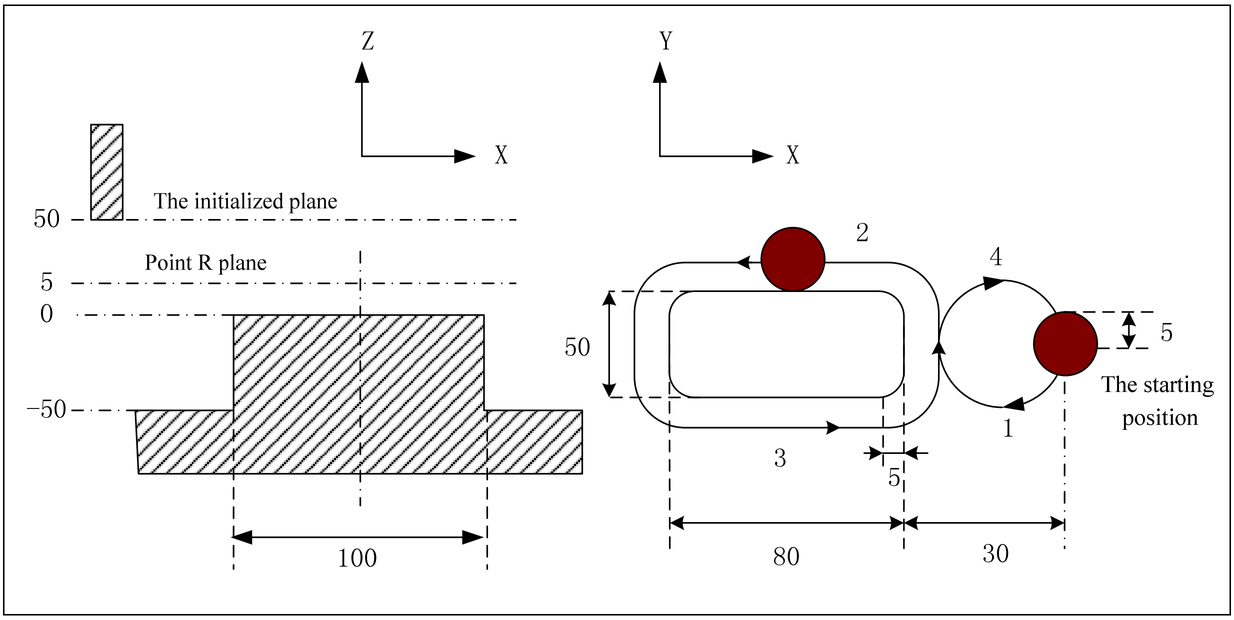
G136	G137
	

Relative introduction:

(1) During the finish milling outside of the rectangle, the interpolation directions of the transition arc and the finish milling arc are different, but the interpolation direction in the command introduction means that of the finish milling arc.

⑵ In the cycle, commands Q, P and L are invalid, but the values of Q and P are saved as the mode numerical value in the canned cycle.

For example: The canned cycle G136 commands finish milling the rectangular section which has already been rough milled, which is shown as below:



G90 G00 X50 Y50 Z50; （G00 rapid position）

G99 G136 X25 Y25 R5 Z-50 I80 J50 L30 U5 F800 D1; （ In the canned cycle, finish mill outside of the rectangle at the hole bottom D1=5）

G80 X50 Y50 Z50; （The canned cycle is canceled, and the tool returns to point R plane）

M30;

3.5 Tool Compensation Function

3.5.1 The Tool Length Compensation G43, G44 and G49

Command formula:

At present, the system supports the two tool length offset modes A/B, which is set by parameter N2600#1.

Mode A:

$$\left.\begin{array}{l}G43\\G44\end{array}\right\}\ Z\_\ H\_\ ;$$

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;

Cancel the tool length offset mode: G49; or H0;

Function:

G43 specifies the positive compensation of the tool length.

G44 Specifies the negative compensation of the tool length.

G49 Cancels the tool length compensation.

Remark Explanation:

The offset value in the memorizer moves the finishing position commanded by Z axis movement command positively or negatively. Through the function, during programming, the offset between the estimated tool length and the actual tool length is set in the memorizer as the offset value. Compensate in the offset direction commanded by G43 and G44 and through the offset value set by command H in the memorizer without changing the program.

1**.** Offset direction

G43: Positive offset (the most ordinary offset mode)

G44: Negative offset

Whether it is the absolute or the increment command, the offset value specified by H code in the memorizer is to add in G43, while subtract from the finishing position coordinate value of the spindle movement command in G44.The coordinate value after calculating becomes the finishing position. When Z axis movement command is omitted, the same mode can be introduced as below:

$$\left.\begin{array}{l}\text{G43}\\ \\ \text{G44}\end{array}\right\}\ \text{G91}\quad \text{ZO H——;}$$

The offset value in G43 is positive direction while negative in G44.

G43 and G44 are mode G codes, after commanding, G codes in different groups always remain valid even they are programmed.

G43 and G44 are mode G codes, and they remain valid before there are other G codes in same group.

2**.** Specifying the offset value

H codes specify the offset number, the offset value in the memorizer of the number adds or subtracts Z axis programming value. The offset number can be specified through H00 or H256.

The offset value corresponds to the offset number and the length offset value is set in the memorizer through MDI/LCD, the span is shown as below:

	Input in mm	**Input in inch**
Offset value	0 ㎜～±999．999 ㎜	0 inch～±999．999inch

The offset value of H00 corresponding to the offset number 00 is usually 0; therefore, the offset value of H00 isn't set.

3**.** Valid sequence of the offset number

Once the length offset mode is set, the current offset number becomes valid immediately; while the offset number changes, the new offset value replaces the old one.

```
O×××××;
H01;
G43 Z10;          (1)  Offset number H01 becomes valid.
G44 Z20 H02;      (2)  Offset number H02 becomes valid.
H03;              (3)  Offset number H03 becomes valid.
G49;              (4)  Cancel the offset, H00 becomes valid.
M30;
```

4. Cancel the tool length compensation

G49 or H00 cancels the tool compensation. After commanding G49 or H00, the system immediately cancels the tool length compensation.

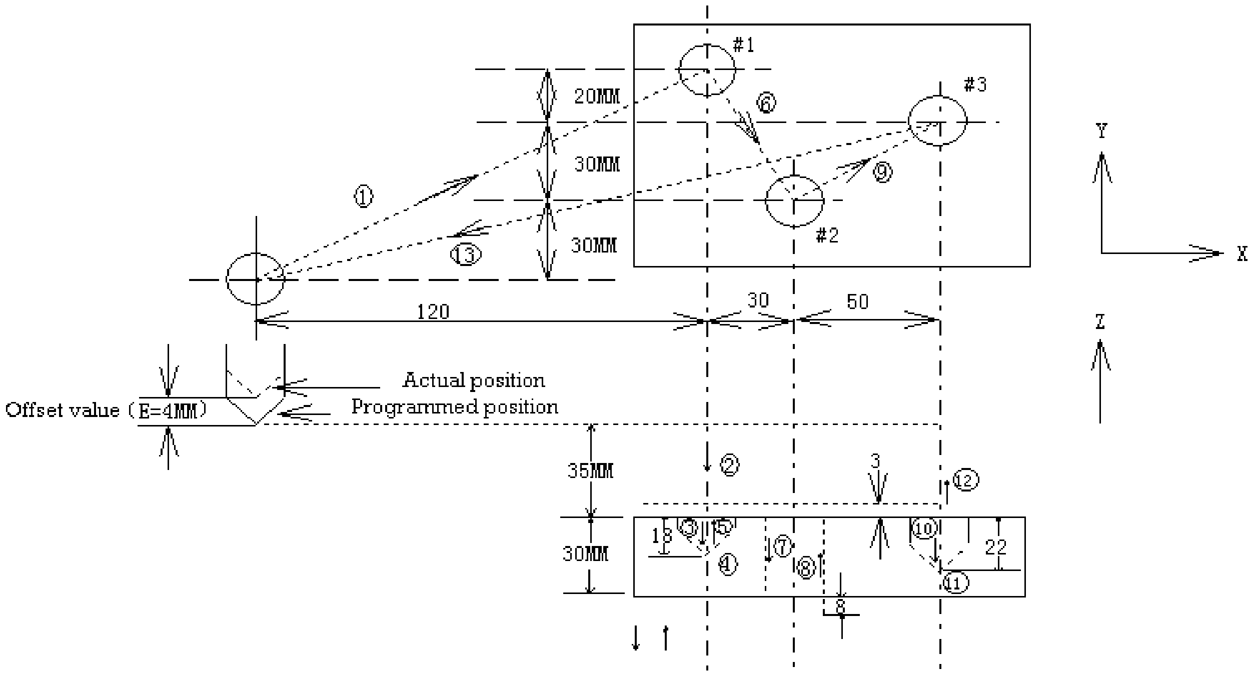
Remark Notice: In the tool length offset mode B, after two or many axes are executed, G94 cancels the offset of all axes, and H0 just cancels the offset of the axis which is vertical to the specified plane.

5. Commands G53, G28 or G30 in the tool length offset mode

In the tool length offset mode, specify G53, G28 or G30, the offset vector of the tool length offset axis is canceled after moving into the specified point. Among them, G53 is canceled after moving toward the commanded position, G28 and G30 are canceled after moving into the intermediate point. However, the mode code doesn't switch into G49, and the axes except for the tool length offset axes aren't canceled. When G53 and G49 are in the same block, the length offset is canceled after all axes move toward the commanded position; when G28 or G30 share the same block with G49, the length offset is canceled after all the axes move toward the intermediate point. In the tool length offset, the tool length offset vector canceled by G53, G28 or G30 restores in the next block.

6. The practical example of the tool length compensation

(1) The examples of the tool length compensation (processing the holes of #1, #2, #3)



H01= －4.0（Offset value）

N1	G91	G00	X120.0	Y80.0	; ……………(1)
N2	G43	Z-32.0	H01		; ……………(2)
N3	G01	Z-21.0	F1000		; ……………(3)
N4	G04	P2000			; ……………(4)
N5	G00	Z21.0			; ……………(5)
N6	X30.0	Y-50.0			; ……………(6)
N7	G01	Z-41.0			; ……………(7)
N8	G00	Z41.0			; ……………(8)
N9	X50.0	Y30.0			; ……………(9)
N10	G01	Z-25.0			; ……………(10)
N11	G04	P2000			; ……………(11)
N12	G00	Z57.0	H00		; ……………(12)

N13 X-200.0 Y-60.0 ; ……………(13)

Remark 1: Because the offset values varies as the offset number changes, the new offset value doesn't add the old one.

H01…………Offset value 20.0

H02…………Offset value 30.0

G90 G43 Z100.0 H01; ……Z will reach 120.0

G90 G43 Z100.0 H02; ……Z will reach 130.0

3.5.2 The Tool Radius Compensation C（G40～G42）

Command formula:

1) G41 D_X_Y_
2) G42 D_ X_Y_
G40 X_Y_

Function:

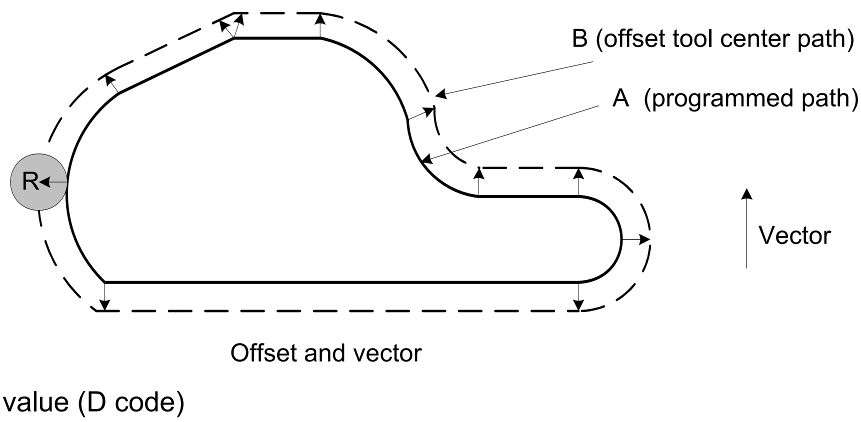
G41 Specifies the left compensation which the tool traverse direction.
G42 Specified the right compensation which the tool traverse direction.
G40 Cancels the tool radius compensation.

Remark:

1. The tool radius compensation function

In the following figure, the tool of radius R processes the work piece specified by A in figure, and the path corresponding to the tool center is position B relative to A and the distance is R. Like this, the distance which the tool leaves the work piece is called as the offset, and the tool compensation function calculates the tool path which is the offset.

Therefore, the programmer can program the work piece shape through the offset mode, and during the processing, if the tool radius (offset value) is measured and set in NC, the tool path is offseted (path B), no matter how the programming path is.



Offset and vector

2. The offset value (D code)

The offset value is set by MDI/LCD based on the command D in the program.

The offset range is set as below:

	Input in mm	input in inch
Offset value	0mm ~±999.999mm	0inch ~±99.9999inch

The offset value corresponding to #00 or D00 is always 0; therefore, the offset value corresponding to D00 isn't set.

Code D specifies the radius offset number, the offset value corresponding to the offset number adds or subtracts the movement command value in the program, which forms the new movement command. The offset number can specify D00～D256. The radius compensation value is set by the diameter value or the radius value, which is set by the bit parameter N02601#7.

The offset value corresponding to the offset number can be preset in the offset memorizer on LCD/MDI panel.

1. Selecting the plane and the vector

Calculate the offset on the plane of G17,G18 or G19, and the plane is called as the offset plane. For example, on the plane of X or Y, calculate the offset value in the program（X, Y）or （I, J）, and the vector. The coordinate values of the axes, which are not on the offset planes, are still executed based on the command value in the command.

When X, Y and Z gang control, the tool path which projects on the offset plane should be compensated.

codes	Offset plane
G17	X－Y planes
G18	Z－X planes
G19	Y－Z planes

2. G40, G41 and G42

G40, G41 and G42 cancel and set the tool radius compensation vector. To set the direction of the offset vector and that of the tool traverse, G40, G41 and G42 can be commanded with G00, G01, G02 or G03 at the same time.

G codes	Function
G40	Cancel the tool compensation
G41	Compensate on the left of the tool
G42	Compensate on the right of the tool

Cancel the tool radius compensation (G40)

On G00 and G01 states, the following commands can be used, G40 X__ Y__ ;

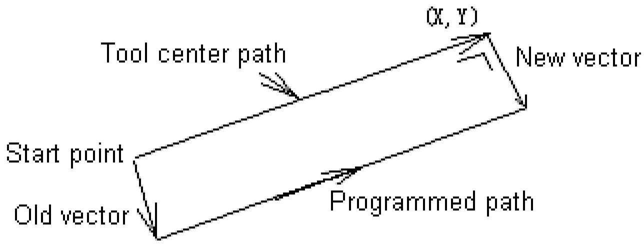
From the starting position of the old vector, the tool linear traverses to the finishing position. In G00 mode, each axis rapidly moves toward the finishing position. The command makes the system access the canceling tool compensation state from the tool compensation state.

The tool doesn't traverse if it is just G40 without commanding X__ Y__.

Compensating the left of the tool radius (G41)

1.G00 and G01

G41 X__ Y__ D__ ; Command at the finishing position of the block, and form a new vector which is vertical to the direction (X, Y), and the tool traverses from the top of the old vector at the starting position to that of the new vector.



When the old vector is 0, the command makes the tool access the tool radius compensation state from the canceling the tool offset state.

2.G02 and G03

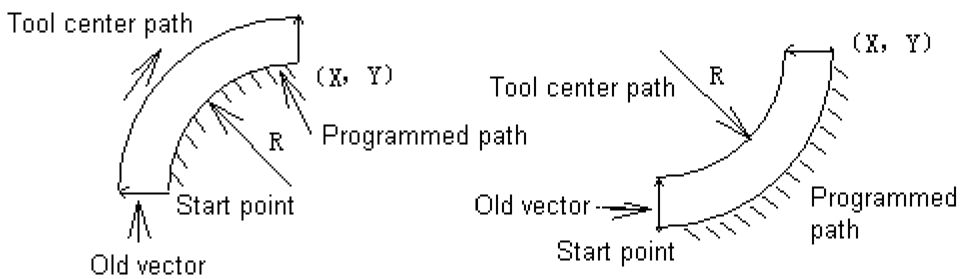
G41……;

……

……

G02 /G03 X__ Y__ R__ ;

Based on the above programs, the new vector is on the line of the arc center and the finishing position. After the old vector has already been jointed, from the arc forward direction, position the left or the right, the tool center traverses along the arc from the old vector top to the new vector one. The offset vector positions the arc center or deviates from the center from the starting position or the finishing position.



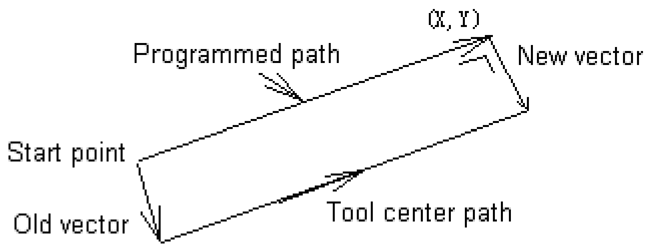
Compensating the right of the tool radius (G42)

G42 is just opposite with of G41 and the tool offsets on the right of the work piece along the tool forward direction. That is to say, the vector direction set by G42 is just opposite with that set by G41. Except for the vector direction, the offset mode is exactly same as that of G41.

1. G00 and G01

G42 X__ Y__ D__ ;

G42 X__ Y__ ;



2. G02, G03

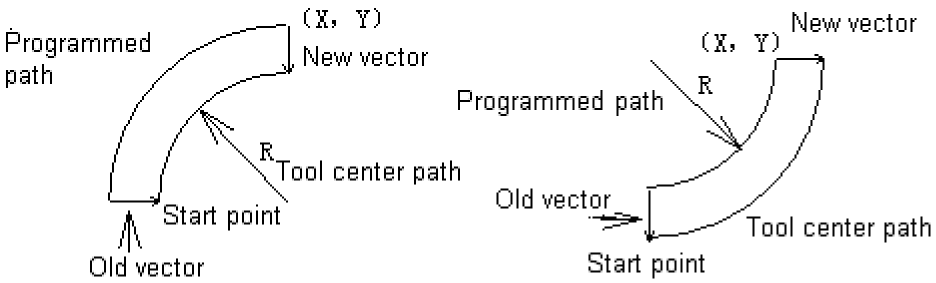


Fig. 3-5-2 (A)

The common positions for attention about the offset:

(A) Specifying the offset number

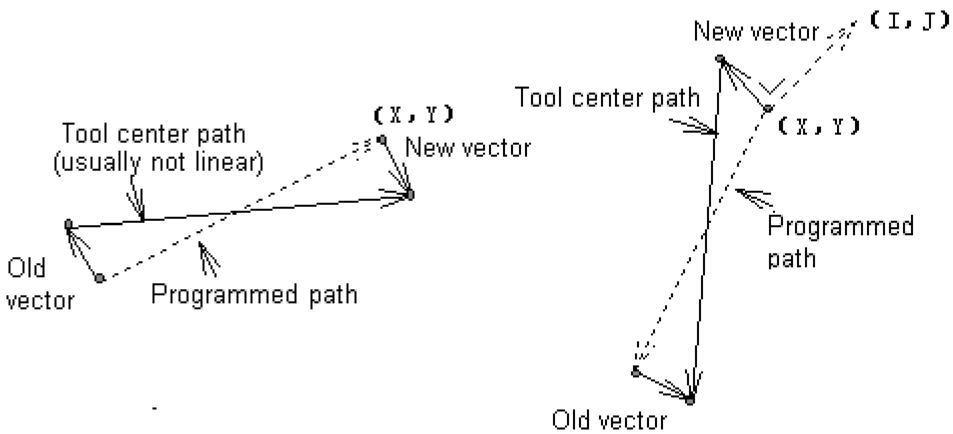
G41, G42 and G40 are mode commands, the offset number is specified by D code. And it can be specified anywhere before the canceling offset state changes into the tool radius compensation state.

(B) The canceling offset state accessing the tool radius compensation state

During the canceling offset state accessing the tool radius compensation mode, the movement commands must be the positioning (G00) or the linear interpolation (G01), which can't be arc interpolation (G02, G03).

(C) Switching between compensating on the left and the right of the tool radius

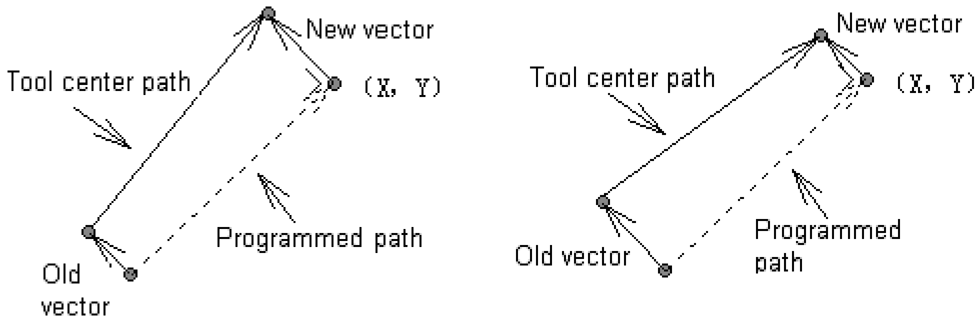
The offset should be canceled before the offset is switched from the left to the right, or from the right to the left. However, the positioning (G00) or the linear interpolation (G01) can be directly switched without offset. And then, the tool path is shown as the following figure:



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

(D) Changing the offset value

During changing the tools, the offset value should be rewritten after the offset is canceled. However, the positioning (G00) and the linear interpolation can be executed in the offset state and the situation is shown as below:



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

If the offset value is negative, it means all G41 and G42 on the program list are interchanged. Therefore, cutting along the outside of the work piece changes into the inside, processing along the inside of the work piece changes into the outside. In the following figure, during the normal programming, and it is assumed that the offset value is positive;

When the tool path programming is shown as figure (A), if the offset value is set as negative, the tool running path is shown as figure (B); Similarly, when the tool path programming is shown as figure (B), if the offset value is set as negative, the tool running path is shown as figure (A).

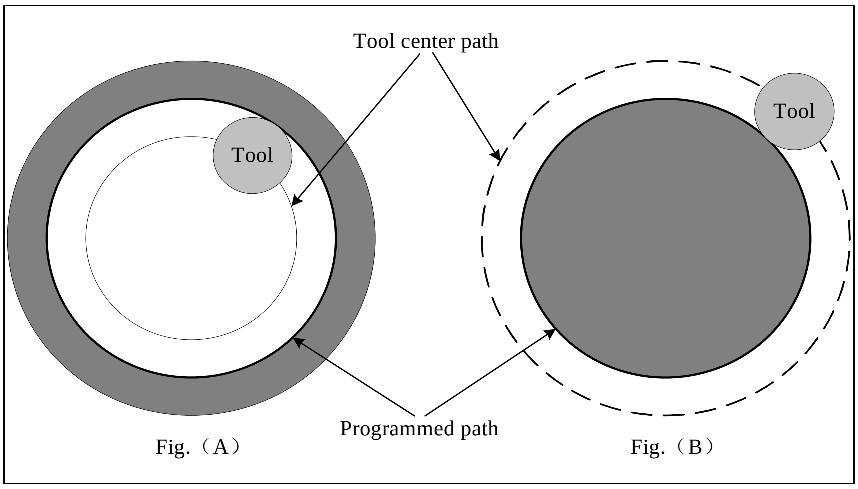


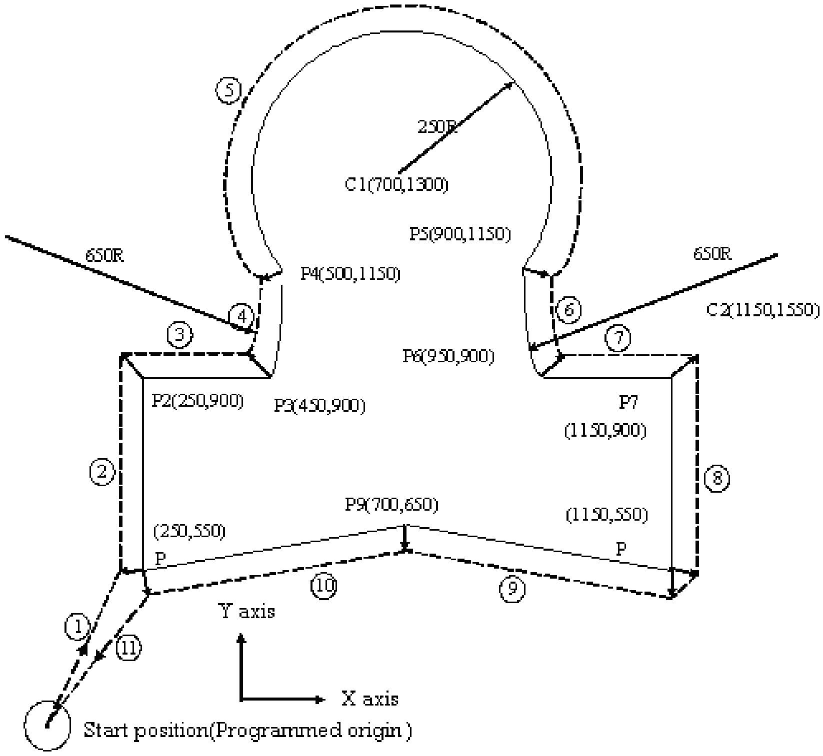
Fig.（A） Fig.（B）

Generally, the figure with the closed angle is very common, that is to say, the figure of arc interpolation with the closed angle. However, after the offset value is set as negative, the inside circle of the work piece can't be processed. When the closed angle of some angle inside, insert the arc with the proper radius over there and it can be cut after rounding off.

Left or right compensation is set by the compensation direction which is in the left or the right of the tool relative work piece(the work piece remains still) traverse direction. G41 or G42 commands the system access the compensation mode, and G40 commands the system cancels the compensation mode.

The example of the compensation program is shown as below:

The program example of the tool path compensation:



G92 X0 Y0 Z0

①N1 G90 G17 G00 G41 D07 X250.0 Y550.0; （ The offset value is preset by MDI in D07）

②N2 G01 Y900.0 F150;

③N3 X450.0;

④N4 G03 X500.0 Y1150.0 I-600.0 J250.0;

⑤N5 G02 X900.0 I200.0 J150.0;

⑥N6 G03 X950.0 Y900.0 I250.0 J0;

⑦N7 G01 X1150.0;

⑧N8 Y550.0;

⑨N9 X700.0 Y650.0;

⑩N10 X250.0 Y550.0;

⑪N11 G00 G40 X0 Y0;

Start the block ①, in the block, canceling the offset mode changes into the offset mode (G41). At the finishing position P_1 of the block, the tool center is offseted by the radius which is vertical to the next block path (from P_1~P_2). The tool compensation value is specified by D07, namely, the offset number is 7 and G41 means compensate on the left of the tool.

The system auto automatically completes the tool compensation after programming and starting the work piece shape $P_1 \rightarrow P_2$———$\rightarrow P_8 \rightarrow P_9 \rightarrow P_1$.

In the block ①, the tool returns to the starting position through command G40 and the offset is canceled. At the finishing position end of the block ⑩, the tool center is vertical to the programming path (from P_9～P_1).

At the end of program, G40 must be commanded and the program is canceled.

3.5.3 The Detailed Introduction of the Tool Radius Compensation

The following is the detailed introduction of the tool radius compensation C.

（1）Canceling

After power on, or during resetting, executing M02 or M30 to end the program and access the canceling offset mode.

In the canceling mode, the vector is always 0, and the tool center path coincides with the programming path, but it is in the canceling mode at the end of the program.

When the program ends in the offset mode, the program finishing position can't be positioned the position can be performed at the end of a program, and the tool position offsets one vector value at the finishing position.

（2）Starting

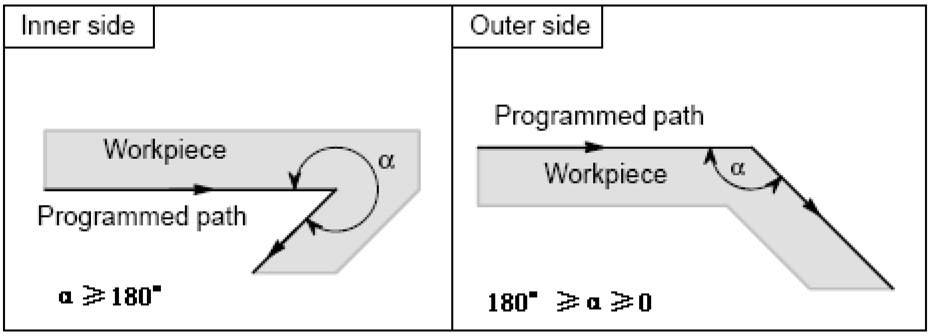
In the canceling mode, when one block which satisfied all the condition is executed, the system accesses the offset mode, and then the block is called as the starting block.

a) G41 or G42 has already commanded and the system accesses G41 or G42 state.
b) The tool compensation number can't be D00.
c) The axis, except for I, J and K, on the offset plane is commanded and its movement value isn't 0.

At the starting program, the arc commands（G02, G03）aren't allowed to use; otherwise, #34 alarms and NC stops running. During starting, NC reads in two blocks, after the 1st block is read and executed, the next block accesses the tool compensation buffer register while the content in the register can't display.

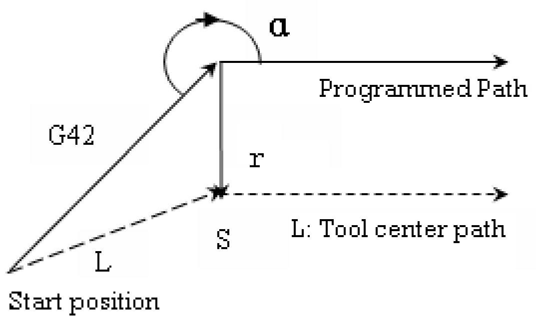
Moreover, in the single block mode, the two continuous blocks are read in and the block which is read firstly stops. And then, the two blocks are usually read in. In NC, there are three blocks and they are respectively the executing block, the next and the following.

Remark Note: The so-called "inside" and the meanings in the other situations are shown as below. The angle of the movement commands of the two blocks is above 180°called as"inside" , and 0°～180°as "outside".



（i） Process around the inside

Straight line→straight line



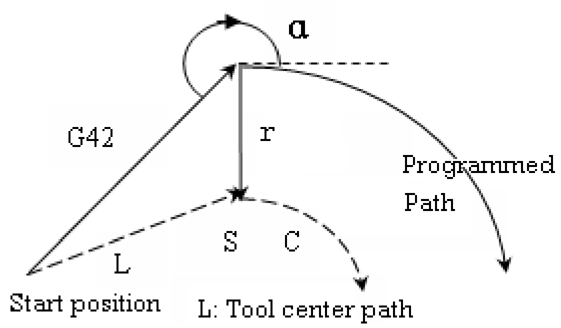
In the figure:

S means the pause position of the single block

L means the linear movement

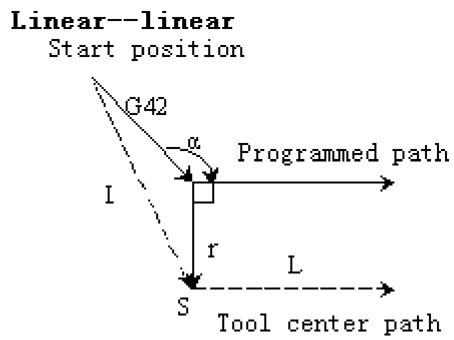
C means the arc movement

Straight line → arc

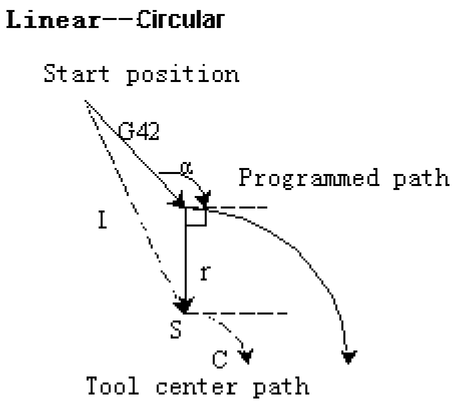


（ii） When the tool feeds the outside of one obtuse angle, （90°≤α≤180°）, there are two inflection positions of A and B which are selected through parameter 2602 #2（TPH） during the starting and the canceling on the tool path.

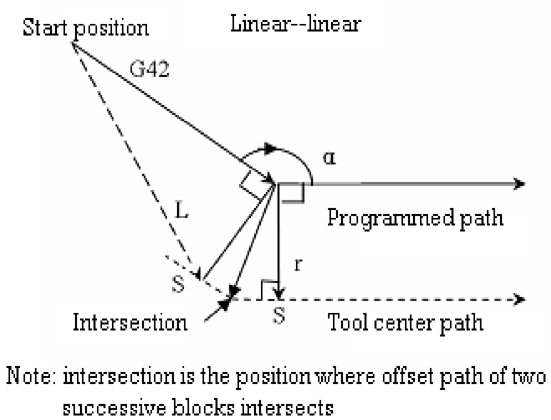
Type A: (Straight line→straight line)



（Straight linear）

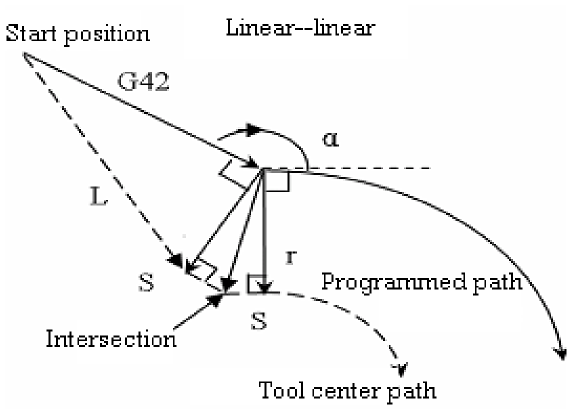


Type B: (Straight line→straight line)



The position of intersect is crossed by the offset paths which are calculated by the two consecutive blocks.

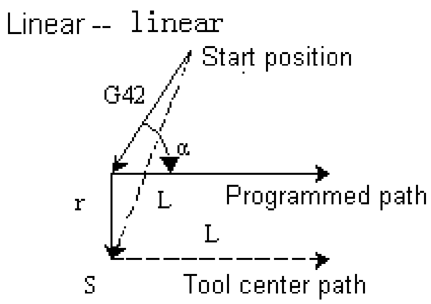
（Straight line→arc）



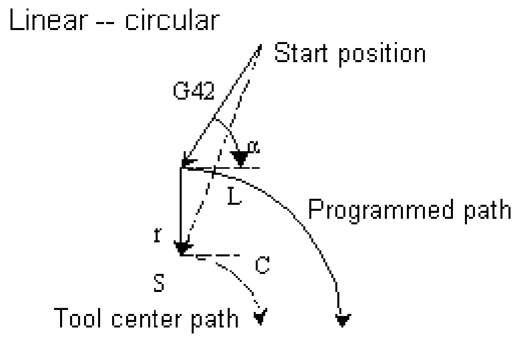
The position of intersect in the above figure is crossed by the offset paths which are offseted by length r of two blocks.

（iii） Feed an acute angle (α＜90°＝outside)

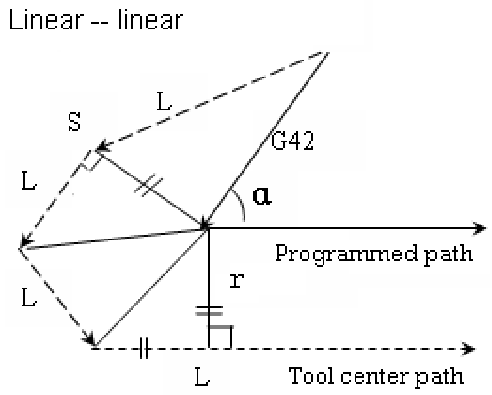
Type A (Straight line→straight line)



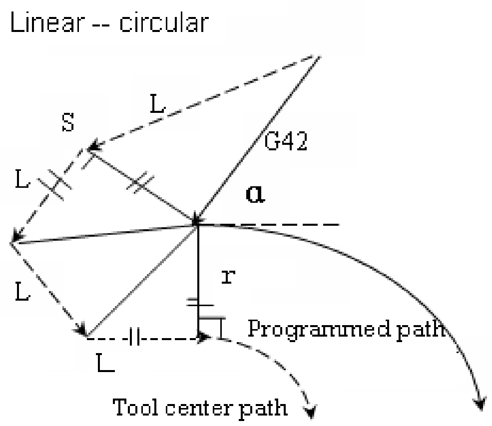
(Straight line →arc)



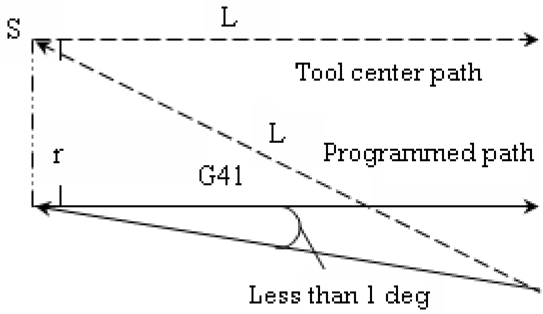
Type B (Straight line→straight line)



(Straight line→arc)



Remark: In the situation of type B, when the tool linear traverses on each side of the angle less than 1°, the compensation is executed as below.



（3）Offset mode

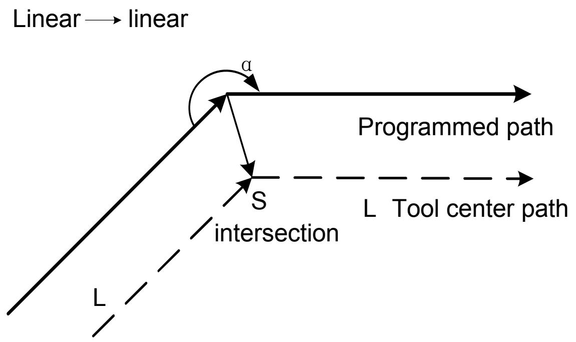
In the offset mode, even the linear interpolation is commanded, the arc interpolation also offseted offsets.

In the offset mode, there aren't any movement commands, only the block of the miscellaneous function and pause, etc can be commanded in the maximum eight consecutive blocks; otherwise, it causes less cutting or cutting too much overcutting.

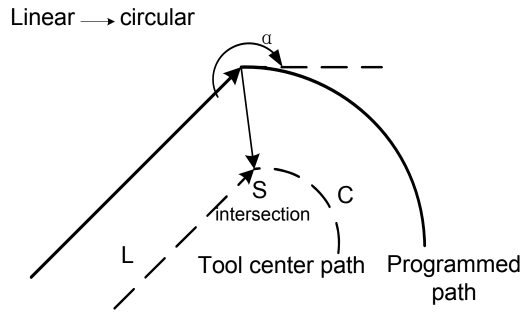
In the offset mode, the offset plane can't be changed, otherwise; N0.37 alarms and the system stops.

（i）The inside corner（180°≤α）

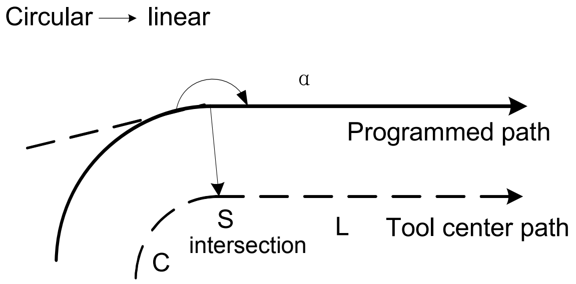
(Straight line →straight line)



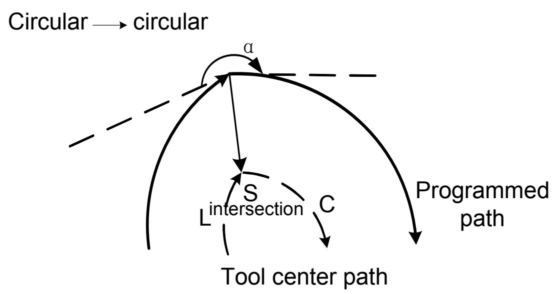
(Straight line →Arc)



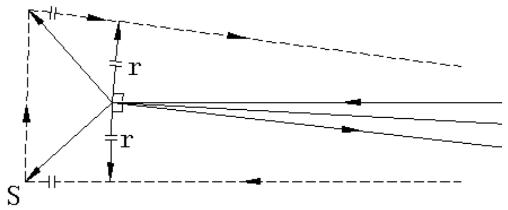
(Arc→ straight line)



(Arc→arc)



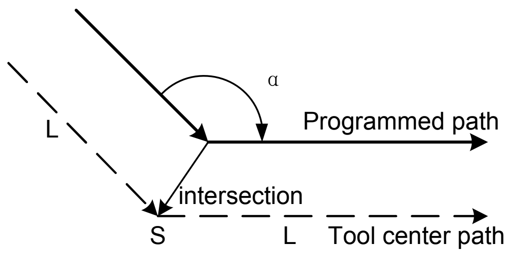
The offset vector is very big when it feeds from the straight line to the straight line and the angle is less than 1°.



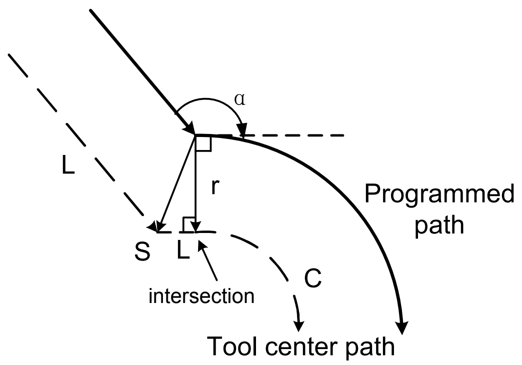
The operator can process in the same method when it feeds from the arc to the straight line, the straight line to the arc and the arc to the arc.

（ii）Feed one obtuse angle along outside（90°≤α＜180°）

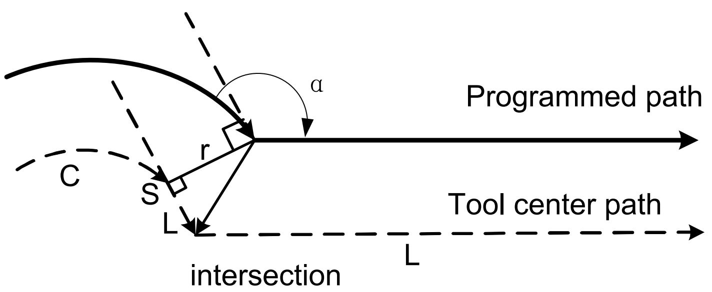
Straight line→straight line



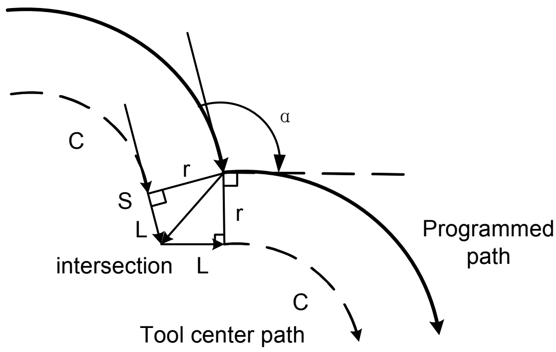
Straight line→arc



Arc→straight line

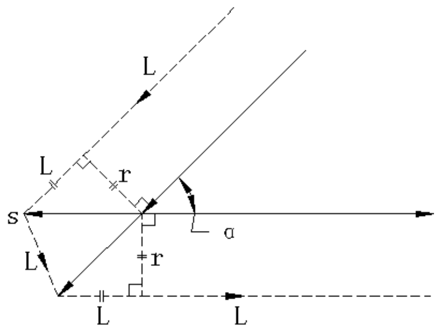


Arc→arc

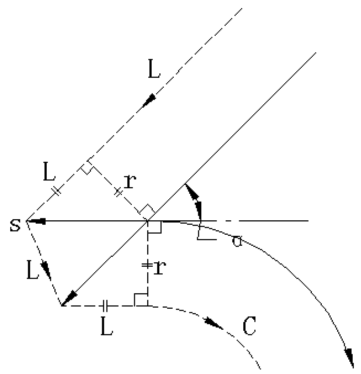


（iii）Feed one acute angle along outside

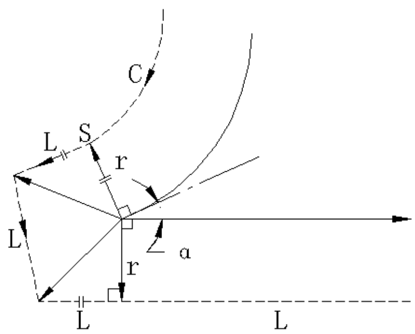
Straight line→straight line



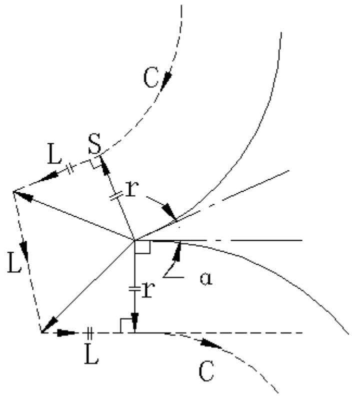
Straight line→ arc



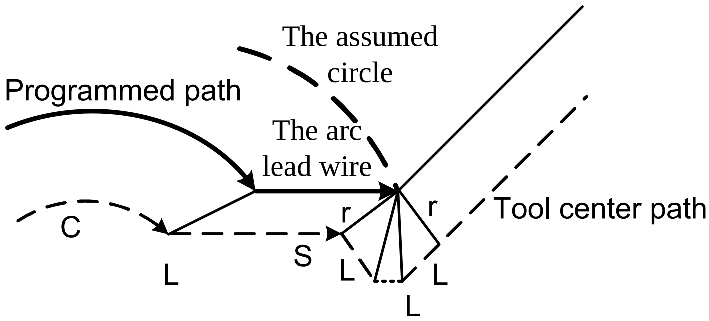
Arc→straight line



Arc→arc



Remark Note 1: In the special situation, the finishing position isn't at the arc. When it is exceptional, the end position for the arc is not on the arc

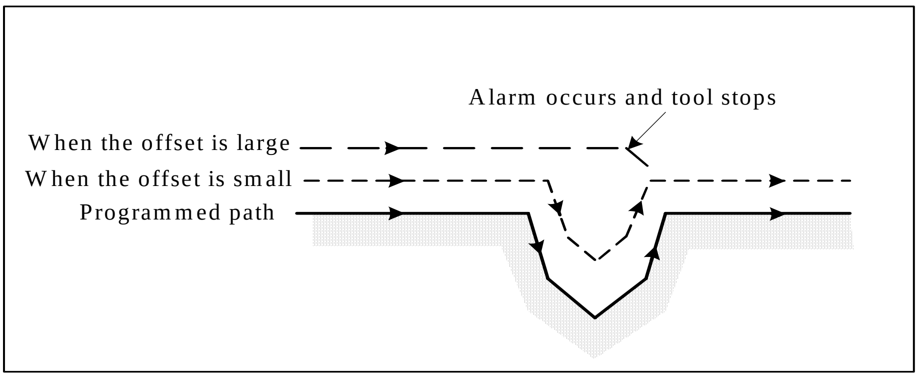


The lead wire in the arc joins the finishing end position of the arc with the arc center as the center of a circle to draw an assumed arc, which is shown in the figure. The assumed arc is taken as one vector to compensate, and its result is different with the tool center path which the arc lead wire is taken as the straight line to compensate.

In the situation from arc to arc, it can be also processed in the same method.

The situation free of the inside position of intersect without an inner intersection

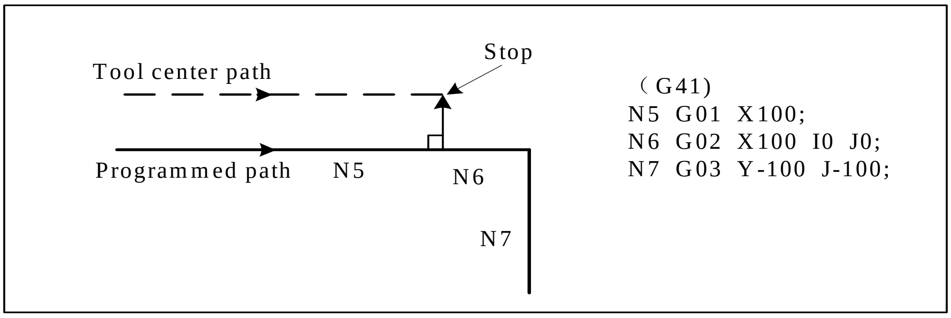
When the offset value is small, the arc position of intersect intersection is on the compensation path, which is shown as below. When the offset value is increased, the position of intersect doesn't exist. Then, NO.33 alarms at the finishing position of the previous block, and the system stops.



In the above figure, when the offset is small, it's on the compensation path of arc A and B, and the position of intersect intersection P exists; if the offset is increased, the position of intersect intersection can be cleared.

When the center coincides with the starting position or the finishing position in the arc,

#38 alarms, and NC stop executing the program at the finishing end position of the previous block.



（4）Canceling the offset

In the offset mode, when the block satisfies one of the following conditions, the system accesses the tool canceling mode, and the block function is called as the canceling offset.

（a）command D00 as the tool compensation number

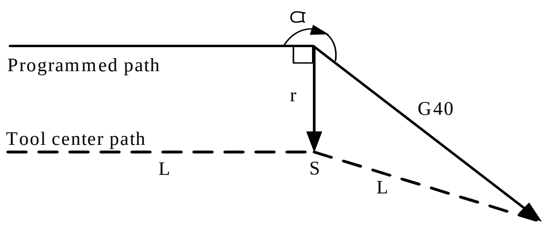
When the offset is canceled, arc commands（G02）and（G03）can't be commanded; otherwise, #34 alarms and NC stops.

In the canceling offset mode, one block is read in, and the second block including the buffer (not display) which saves the tool compensation is executed. In the single block mode, one block is read in and stopped after executing, and it is executed again through pressing the start button and the next block is executed.

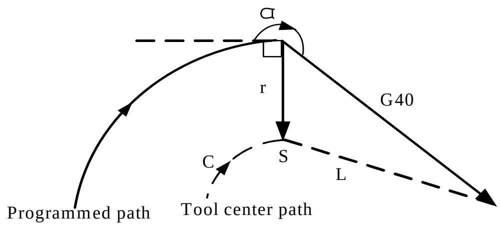
After the control system accesses the canceling mode, in the normal situation, the next executed block is saved in the buffer register without accessing the tool compensation buffer register.

(b) Feed the inside corner（α≥180°）

Straight line→straight line



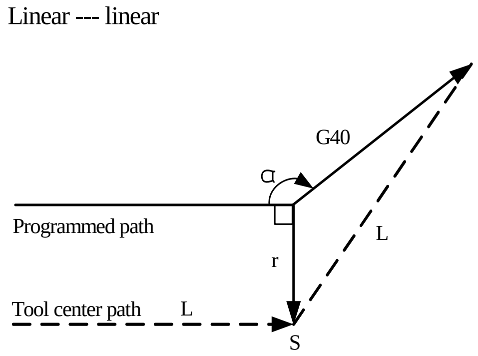
Arc→straight line



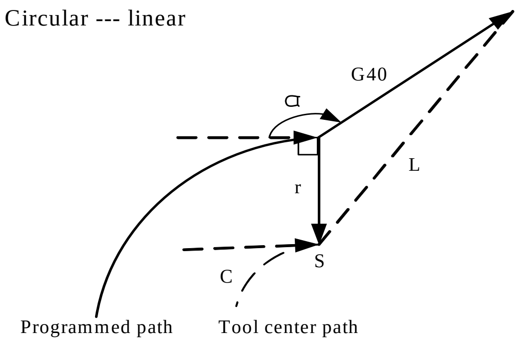
(c) Feed the outside corner （90°≤α＜180°obtuse angle）

（ i ） Type A

Straight line→straight line

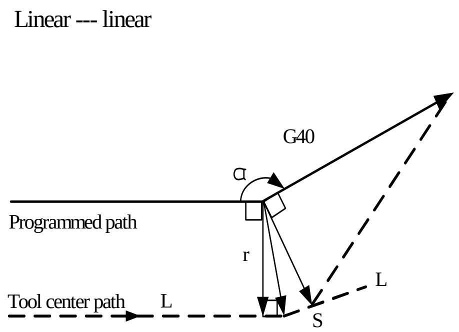


Arc→straight line

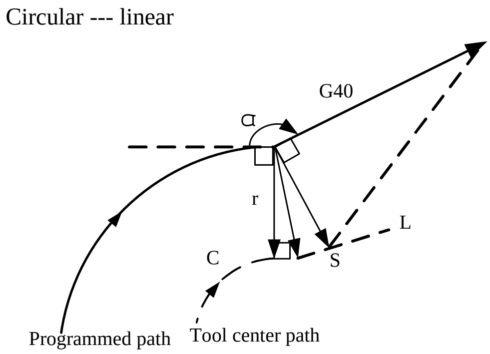


（ ii ） Type B

Straight line→straight line



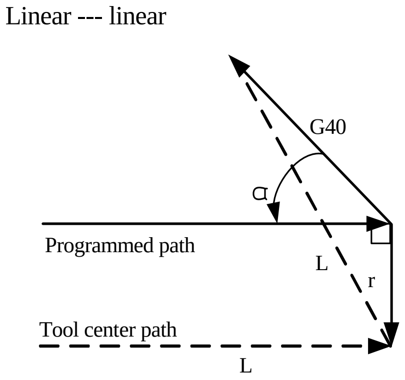
Arc→straight line



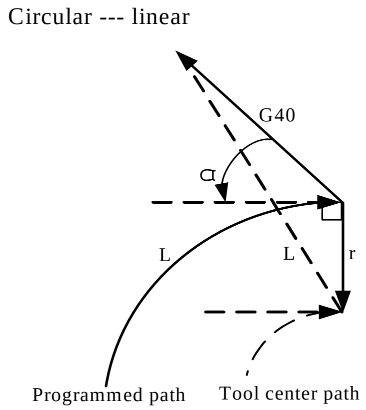
Feed the lateral angle of the acute angle

(i) Type A

Straight line →straight line

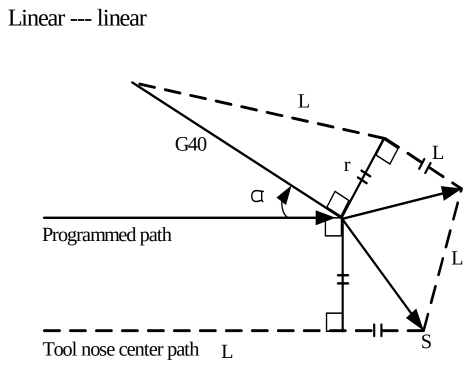


Arc→straight line

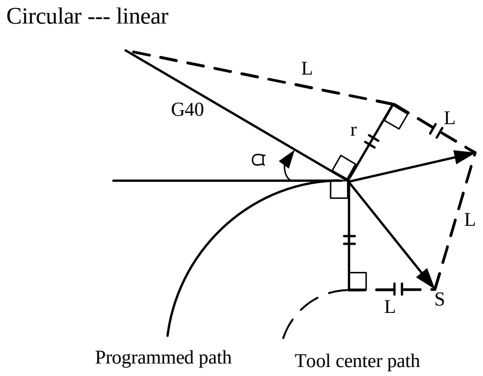


(ii) Type B

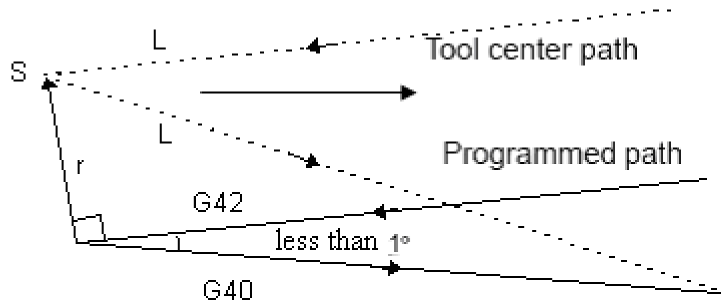
Straight line →straight line



Arc →straight line



Remark Note: In type B, the compensation mode is shown as below when the tool traverses from the straight line to the straight line from outside and the acute angle is less than 1°.



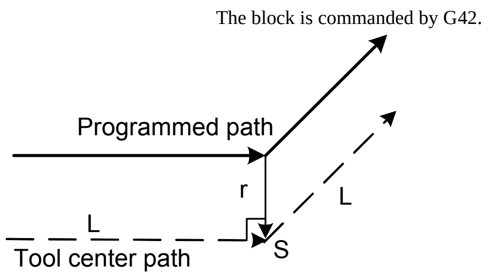
（5）G codes of the tool compensation in the offset mode

In the offset mode, G codes （G41 and G42）of the tool compensation can’t be respectively commanded. Relative with the movement direction of the previous block, the offset vector can be set to form the correct angle and it doesn’t connect with is regardless of the processing inside or outside.

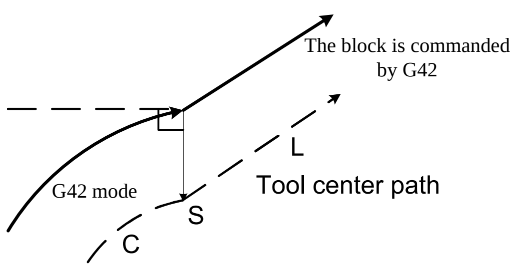
If the codes （G41 and G42）are included in the arc commands, the arc moves wrongly.

Switch the compensation direction through commanding the tool compensation G codes (G41 and G42), refer to “switching the offset direction in the offset mode” in remark 2.

Straight line→straight line



Arc→straight line

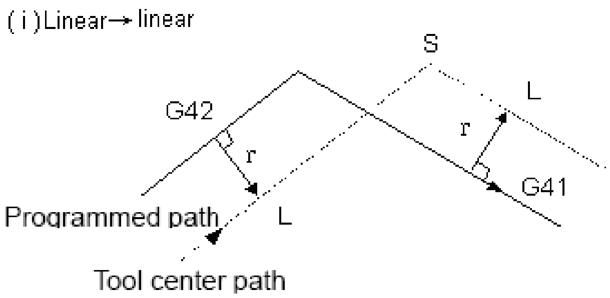


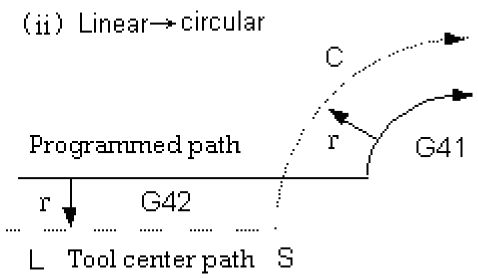
Remark 2: Switching the offset direction in the offset mode

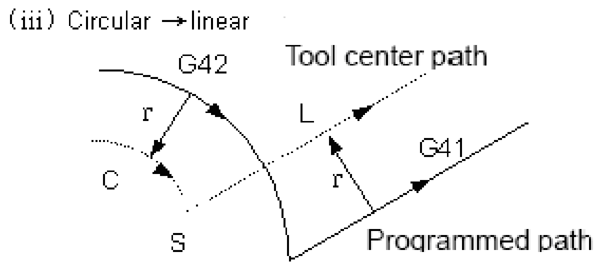
The offset direction is set by G codes（G41 and G42）of the tool compensation and the offset value codes:

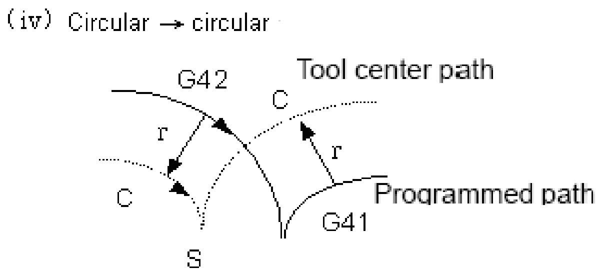
codes	The offset value codes	
	+	−
G41	Left offset	Right offset
G42	Right offset	Left offset

In the special situation, in the offset mode, switching G41 and G42 can change the offset direction, while it can't switch the starting block and the next block. In the situation of switching the offset direction, the concepts of inside and outside are canceled to apply to all the situations. It is assumed that the offset value is position in all the following examples.





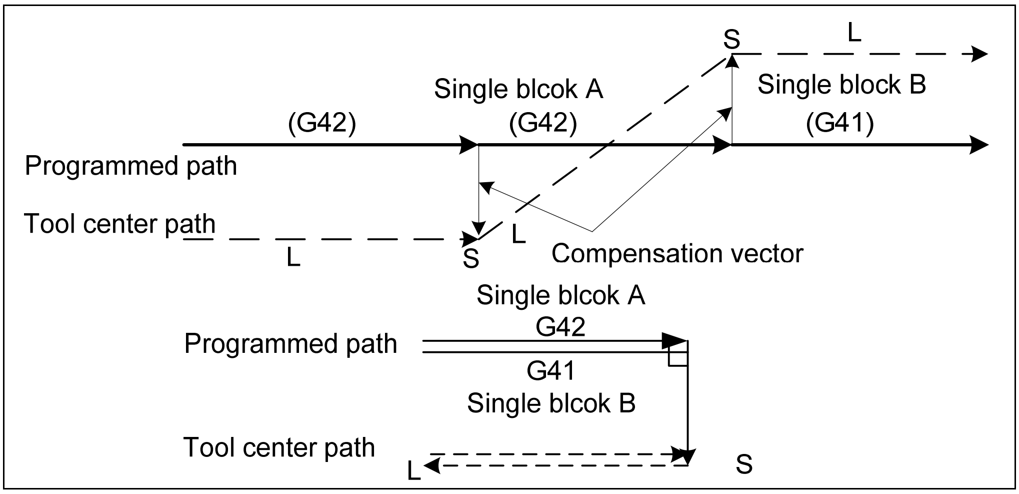




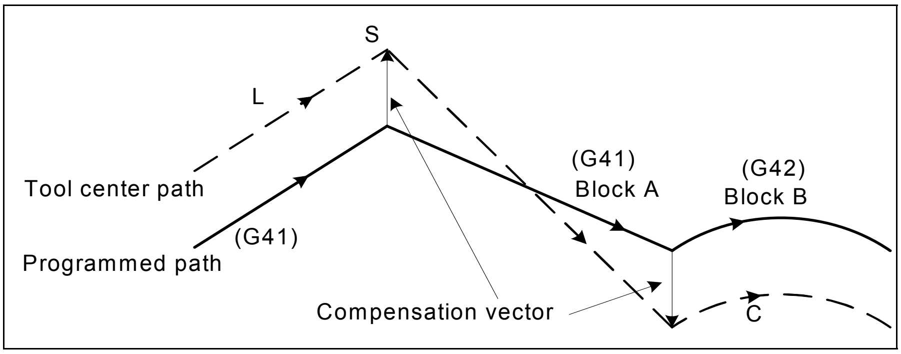
There aren't any positions of intersect intersections in the following path after adding the offset:

From block A to B, if there aren't any positions of intersect of the offset path during switching G41 and G42, the vector vertical to the programming direction is set at the starting position of block B.

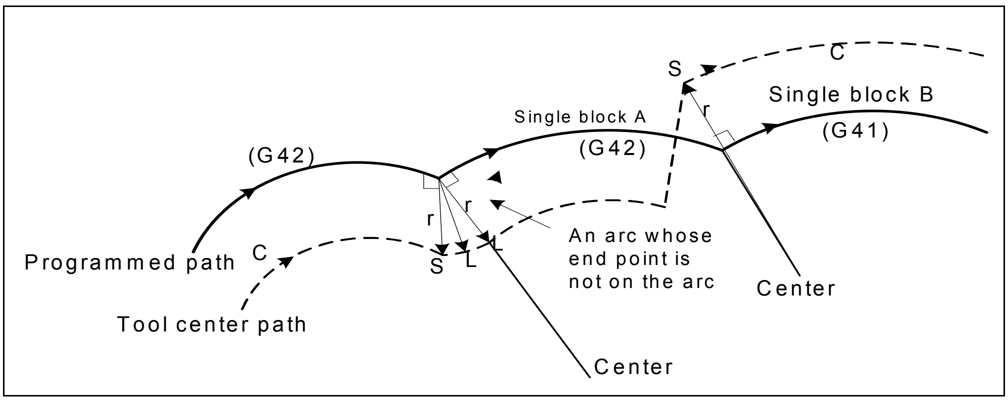
a) Straight line →straight line



b) Straight line → arc

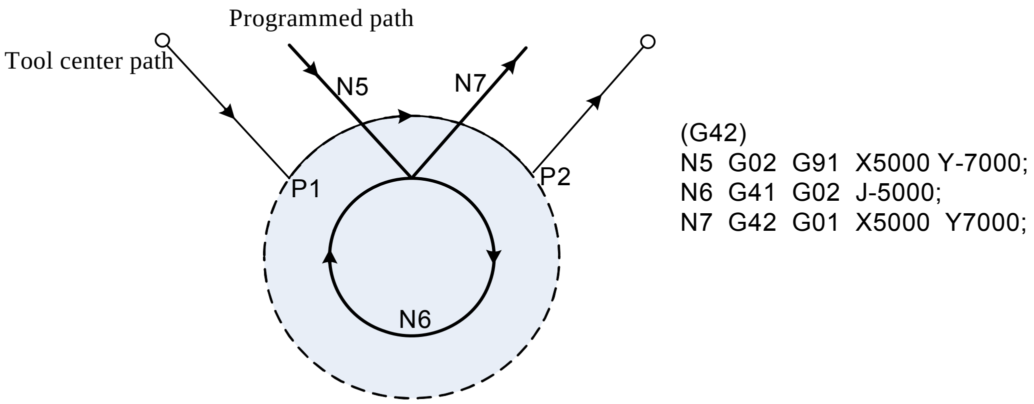


c) Arc→arc



The tool center path length is longer than the circumference, which results from the tool compensation:

Usually, the above situation doesn't happen. Only during switching between G41 and G42, or the addresses I, J and K command G40, it happens.



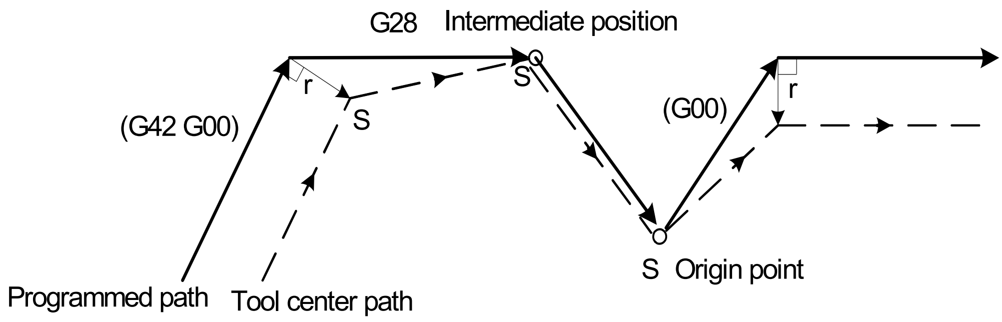
In the above situation, the tool center path just traverses along P1～P2 arc rather than around one circumference and its reason is explained in the alarm resulting from the interference test. If it requires the tool traverse for one circumference, the circumference must be commanded in different blocks.

（6）Canceling the temporary offset, the following commands are executed in the offset mode, “canceling the temporary offset” is caused, auto come back to the offset mode in the system. the system is then restored to the offset mode automatically.

（a） G28 automatic reference point Return

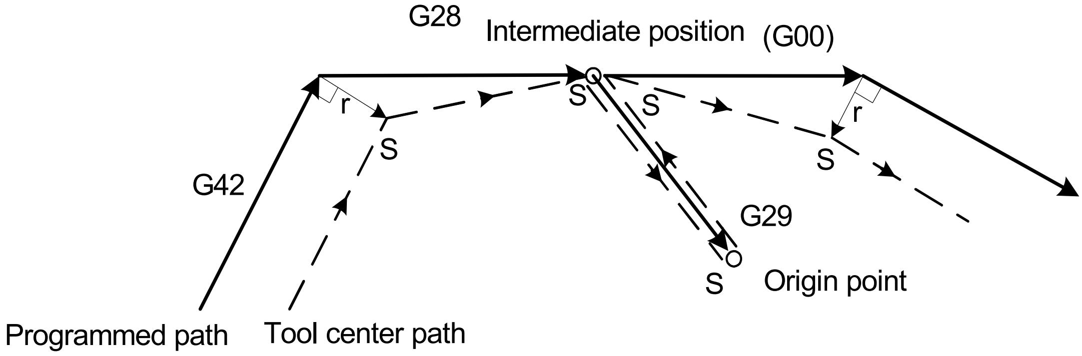
If G28 is commanded in the offset mode, the offset is canceled in the intermediate point, after reaching the reference position, auto come back to the offset mode.

If the offset vector remains in the intermediate point, NC makes each axis vector as zero after the reference point Return.

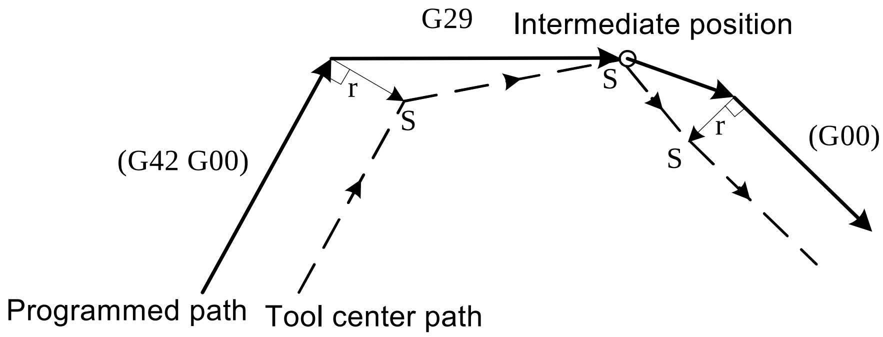


（b） G29 auto return from the reference position

If G29 is commanded in the offset mode, the offset is canceled in the intermediate point and auto restores in the next block. After G28, G29 is directly commanded.

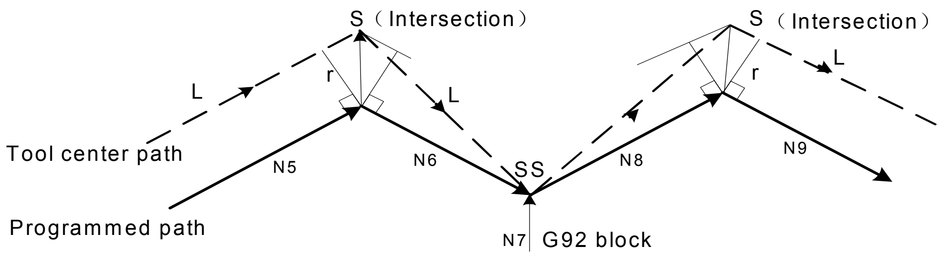


After G28, G29 can be directly commanded in the following situation. Directly command a situation other than G29



（7）When the offset modes are same, if G92 (absolute position 0 programming) is commanded, the offset vector is canceled temporarily, and the following offset vector automatically restores.

In the situation, the tool directly traverses from the position of intersect to the position of the canceling the offset vector without canceling the offset. And when the offset mode is restored, the tool directly traverses to the position of intersect.



(G41 mode)

N5 G01 X3000 Y7000;

N6 X－3000 Y6000;

N7 G92 X1000 Y2000;

N8 G01 X4000 Y8000;

Remark: SS means the position which the tool stops twice in the single block mode

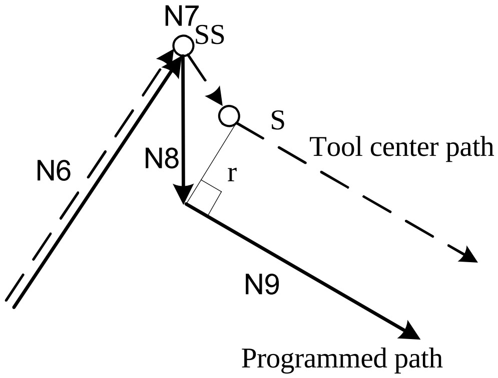
（8）The block free of the tool traverse

The tool doesn't traverse in the following block, even the tool radius compensation in these blocks is valid, the tool doesn't traverse.

M05; ……………………………………………M Output M codes		Not traverse
S21; ……………………………………………S Output S codes		
G04 X1000; …………………………………Pause time		
G22 X100000; ………………………………Setting the processing area		
G10 P01 X100; …………………………… Setting the offset value		
（G17） Z2000; ……………………………… Traverse outside of the offset plane		
G90; …………………………………………… Only G codes		
G91 X0; …………………………………… 0 The traverse value is 0		

a) Command during the starting

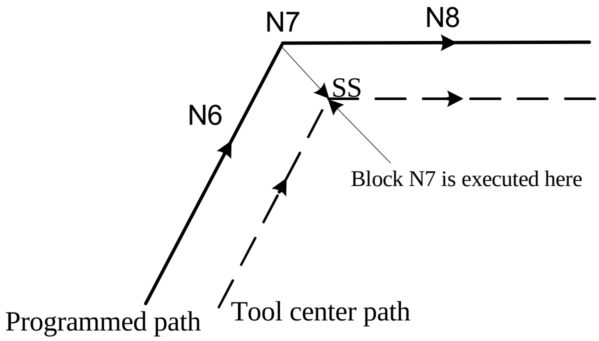
There isn't any offset vector if the block free of the tool traverse is commanded at the starting position of the block.




```
G40  G91……
.
.
N6  X100  Y100;
N7  G41  X0;
N8 Y-100;
N9  X100  Y-100;
```

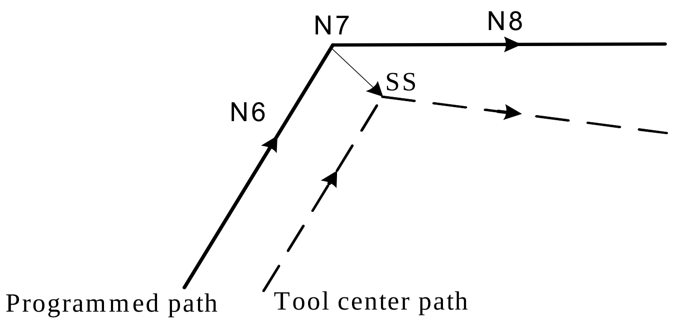
b) Command in the offset mode

In the offset mode, the block is executed at the stopping position of the single block when the single block free of the tool traverse is commanded and its vector and the tool center path are same as that without commanding the block.



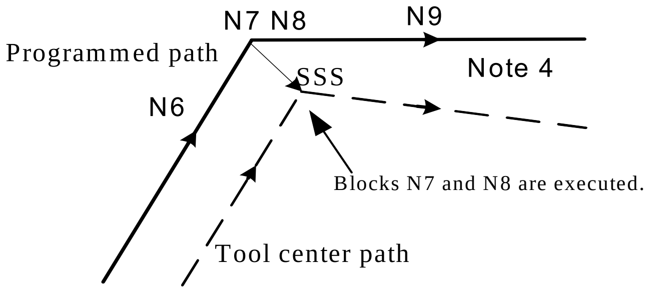

```
N6  G91  X100  Y200;
N7  G04  X100;
N8  X100;
```

But, when the traverse value is 0, even command in the single line, the tool traverse situation of the block is same as that commanding one more block free of the tool traverse. The detailed introduction is shown as below:




```
N6  G91  X100  Y200;
N7  X0;
N8  X100;
```

The block free of the tool traverse can't be specified consecutively more than two blocks; otherwise, the vector is the offset value and its direction is vertical to the tool traverse direction in the previous block; therefore, it may cut too much.

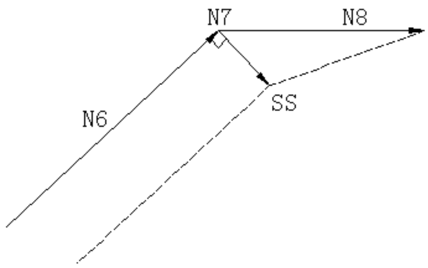


N6 G91 X100 Y200;
N7 S21;
N8 G04 X1.0;
N9 X100.0;

Remark 4: SSS means the position which the tool stops for three times in the single block.

c) Command with the canceling offset

When the block free of the tool traverse and the offset are specified at the same time, the vector is the offset value and its direction is vertical to the tool traverse direction in the previous block; the vector is canceled in the next traverse command.

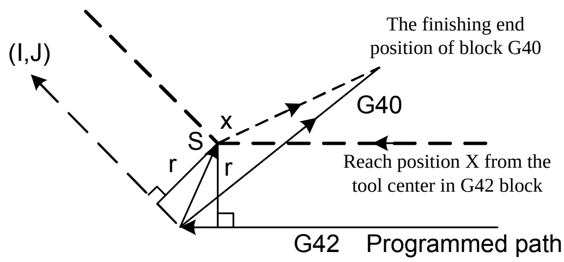


N6 G91 X100.0 Y100.0;
N7 G40;
N8 X100.0 Y0;

（9）The contents of G40 and I—J—K on the offset plane are specified, and the previous block mode is G41 or G42.

In the offset mode, the above are specified, it changes into the situation which is shown on G17 plane, and the situations of the other planes can be processed based on it.

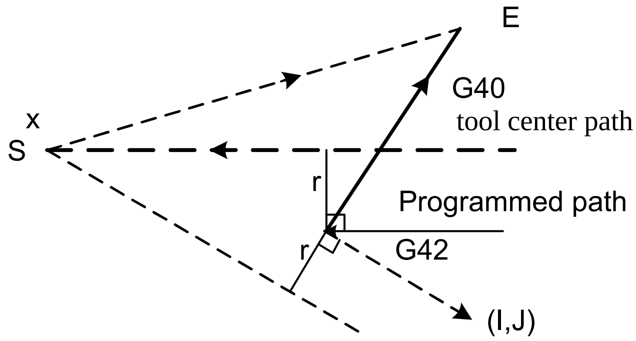
Then, the direction of the vector (I, J) which starts at the finishing position in the previous block is set by the above commands. The offset direction is same as that of the previous block.



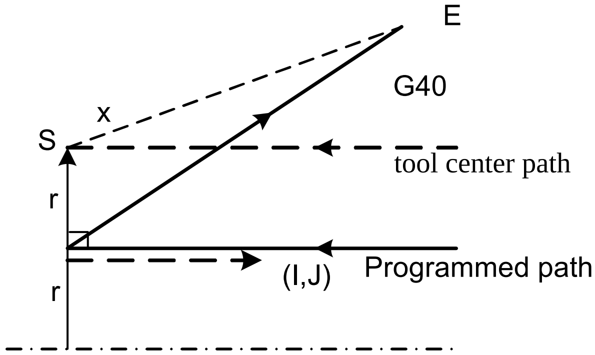
(G42 mode)

G40 X x Y y I — J --;

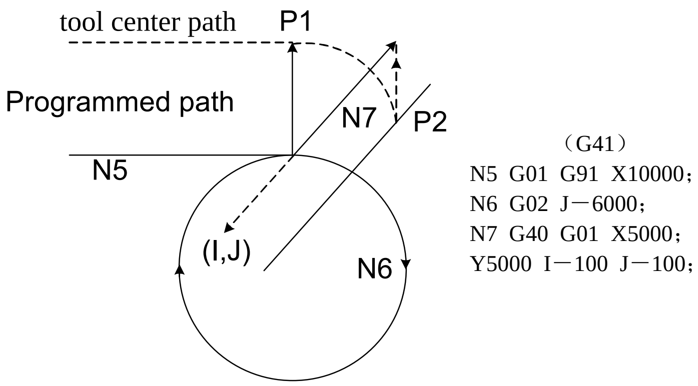
Remark 5: In the situation, pay attention to that NC gets the position of intersect of the tool path, which doesn't connect with the processing inner surface or the outer surface.



Remark 6: When the position of intersect can't be obtained, at the finishing position of the previous block, the tool reaches the position vertical to the previous block path.



Remark 7: The length of the tool center path is longer than the circumference:

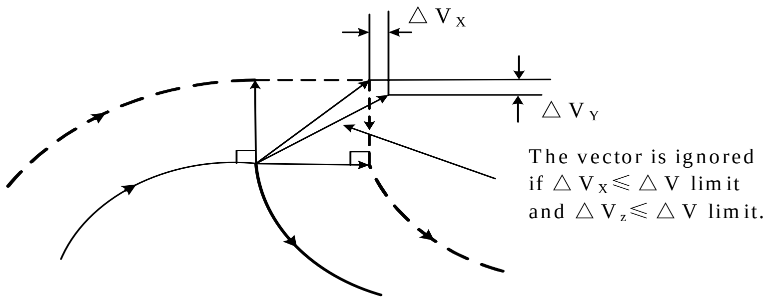


In the above situation, the tool center path just traverses along the arc of $P_1 \sim P_2$ rather than around the circumference.

The alarm caused by the interference check is connected with the following situation. (If it requires the tool move around the circumference, one circumference should be commanded in different blocks.)

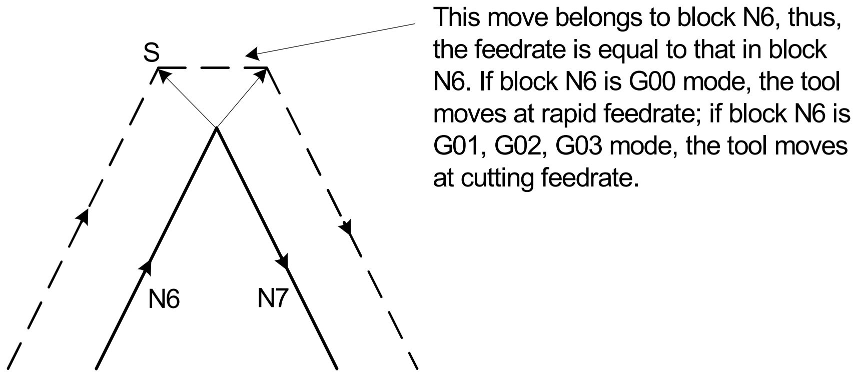
(10) Moving the corner

At the finishing position of the block, there are two or more vectors over there, the tools traverses from one vector to the other vector straight. If the vectors almost coincide, the corner doesn't move and the following vectors are ignored.

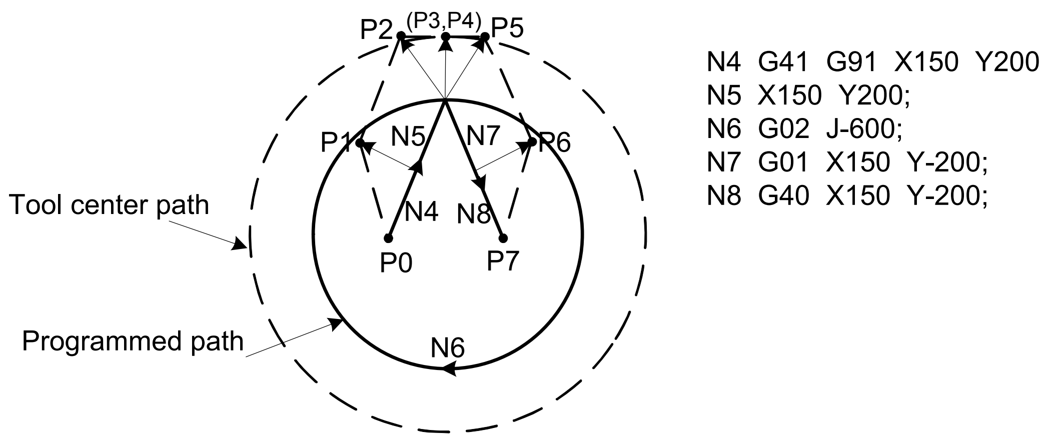


If $\triangle V_X < \triangle V$ and $\triangle V_Y < \triangle V$ are limited, the following vectors are ignored. The limitation of △V is preset by the parameter 069（CRCDL）.

If these vectors don't coincide, it moves around the corner in the following block.



Remark 8: However, the above function isn't executed when the path of the next block is the arc more than the semicircle,
The reasons are as below:



If the vector isn't ignored, the tool path is shown as below:

$P_0 \rightarrow P_1 \rightarrow P_2 \rightarrow P_3$（circumference）$\rightarrow P_4 \rightarrow P_5 \rightarrow P_6 \rightarrow P_7$

However, if the distance from P_2 to P_4, P3 is ignored. And then, the tool path is shown as below:

$P_0 \rightarrow P_1 \rightarrow P_2 \rightarrow P_3 \rightarrow P_4 \rightarrow P_5 \rightarrow P_6 \rightarrow P_7$

The arc cutting commanded by block N6 is ignored.

（11）The common positions cautions for attention about the compensation:

a）Specifying the offset value

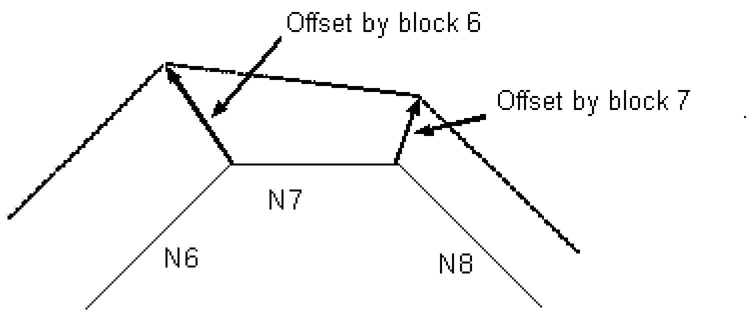
D codes specifying the offset number commands the offset value.

Once they are specified, D codes remain valid till the other D codes are specified or itself is cleared.

D codes not only specify the offset value of the tool compensation, but also the one of the tool position offset.

b) Rewriting the offset value

Usually, in the canceling mode, the offset value is rewritten during changing the tools. If the offset value is rewritten in the offset mode, the vector of the finishing position in the block applies to the new offset value.



c) The positive and negative offset values and the tool center path

If the offset value is negative （－）, G41 and G42 exchanges with each other. That is to say, the original tool center traverses along the outside of the work piece, it traverses the inside, vise versa.

The following figures is one example, normally, the offset value is programmed in the positive value.

The tool path is programmed shown as the figure 3-33 (a); if the offset value is negative, the traverse of the tool center is shown as figure 3-33 (b), vise versa. Therefore, processing the female die and the male die can use the same block and their gap can be adjusted through the offset value (It can also be used if the starting and the canceling are type A.).

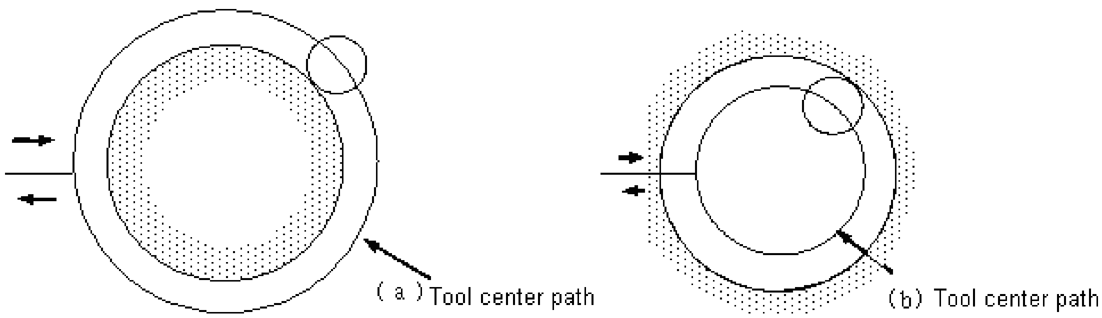
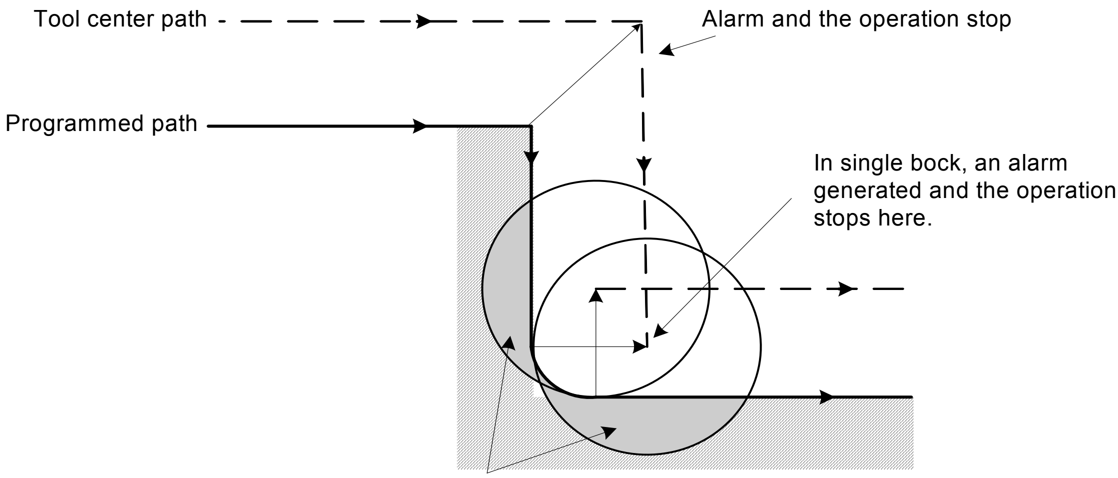


Fig.3-33 The tool path

d) The cutting too much overcutting due to the tool compensation

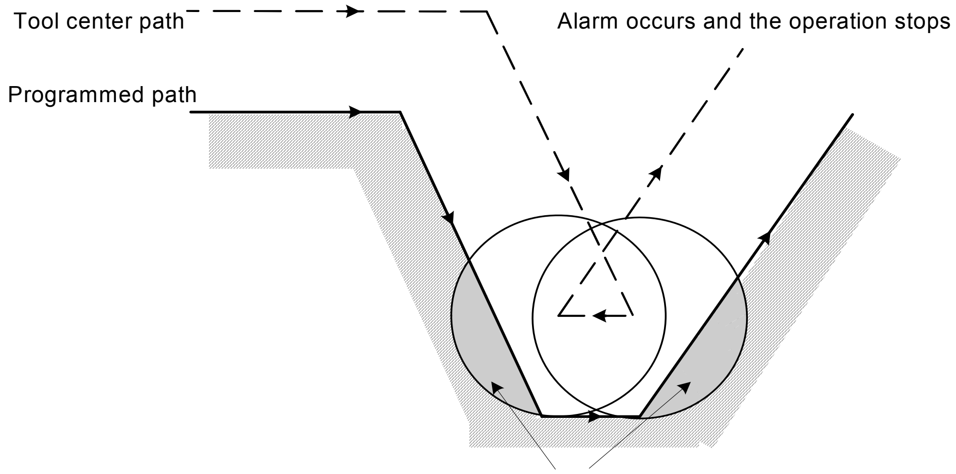
（ⅰ）Processing the arc inside less than the tool radius

When the commanded arc radius is less than the tool one, #41 alarms at the starting position of the previous block and the system stops because the inside offset of the tool may cause cutting too much. However, in the single block, it may cause tool much cutting since the tool stops after the program ends. Then, the tool traverse is same as that when #41 alarms, which is introduced in the following.



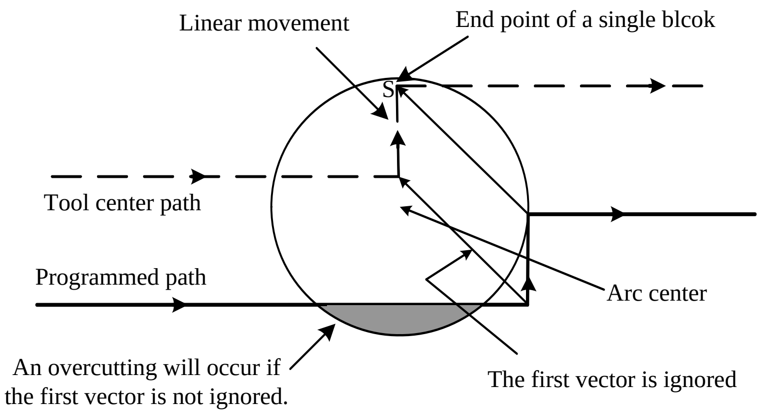
（ii）Process the groove less than the tool diameter

t may cause cutting too much because the tool compensation makes the tool center path traverse in the opposite direction of the programming one. Therefore, #41 alarms at the starting position of the block and NC stops.



（iii）Processing the step less than the tool radius

The tool center path of the common offset traverses in the opposite direction of the programming because there is step less than the tool radius in the program and the circular cutting commands processing the steps. Then, one vector is ignored and the tool traverses toward the position of the 2nd vector. In the single block, the tool stops on the position; otherwise, the program continues executing.

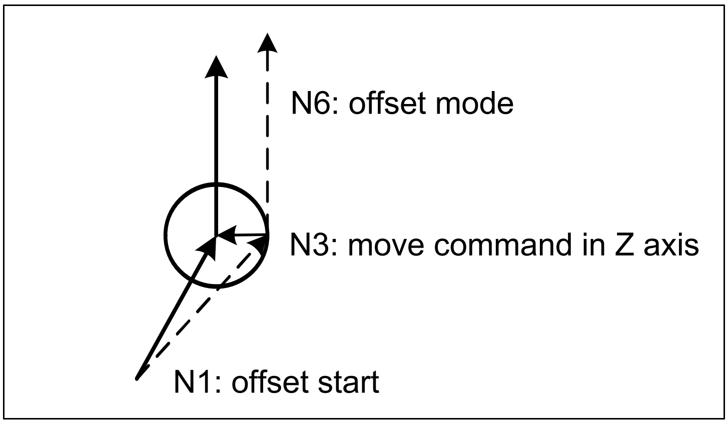


（iv）Starting the tool compensation C and movement in Z axis direction

When the cutting starts, the tool radius compensation which is usually on planes of X and Y is preset on the position which keeps some distance from the work piece, and then, the tool feeds along Z axis. Then, if it requires Z axis rapid feed and cutting feed are together, pay attention to the following problems in the program:

Refer to the following programs:

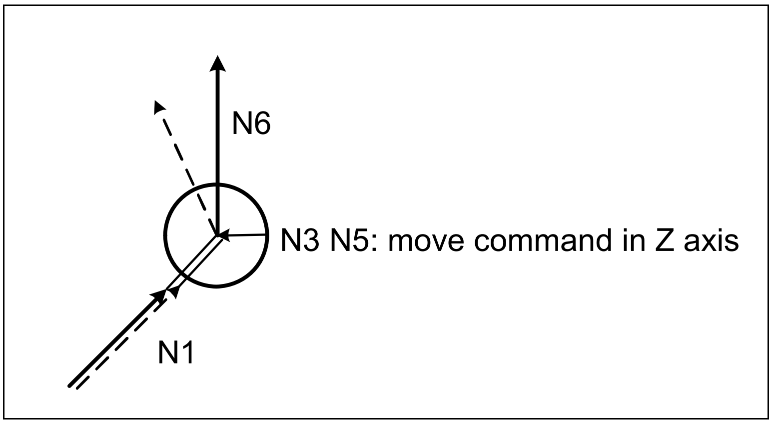
N1 G91 G00 G41 X50000 Y50000 D1;
N3 G01 Z-30000 F1;
N6 Y100000 F2;



In the above example, when N1 block is executed, N3 and N6 are also read in the buffer register, and compensate correctly based on their relation, which is shown as the right figure.

Moreover, if N3 (Z axis traverse command) is separated:

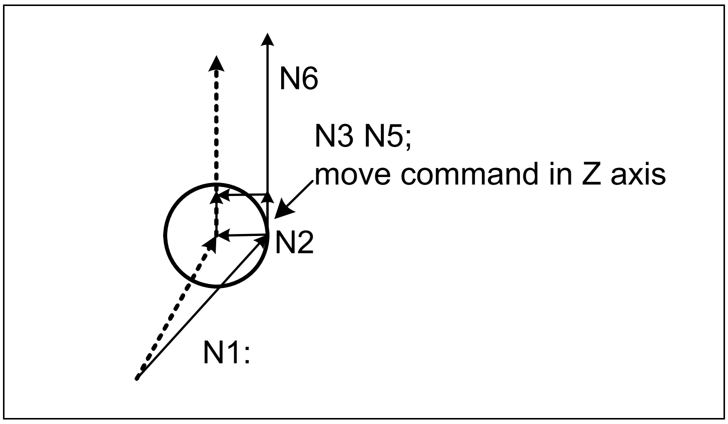
N1 G91 G00 G41 X50000 Y50000 D1;
N3 Z-25000;
N5 G01 Z-5000 F1;
N6 Y100000 F2;



Because the two traverse blocks N3 and N5 are not included on the planes of X and Y, when N1 starts executing, block N6 can't access the buffer register, the result of the tool center path is calculated by N1 information in the right figure. In the situation, the tool vector can't be formed during starting, therefore, cut too much overcutting, which is shown as the right figure.

In the situation, the commands in the specified same movement direction in the blocks before and after Z axis feeding commands based on the above rules, and the cutting too much can be prevented.

N1 G91 G00 G41 X50000 Y40000 D1;
N2 Y10000;
N3 Z-25000;
N5 G01 Z-5000 F1;
N6 Y100000 F2;
（The movement direction commanded by N2 and N6 is same.）



When N1 block is executed, the blocks N2 and N3 read in the buffer register, and correctly compensate based on the relation between N1 and N2.

Remark 9: Interference check:

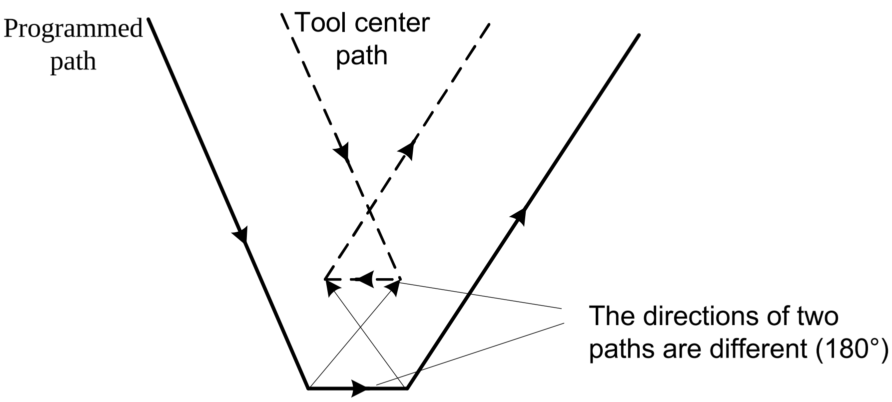
The tool cutting too much is called as "interference" The interference check is the function to check the tool cutting too much in advance. However, not all the further interference is checked by the function, or there isn't the interference but it's still checked.

1） The base of further checking

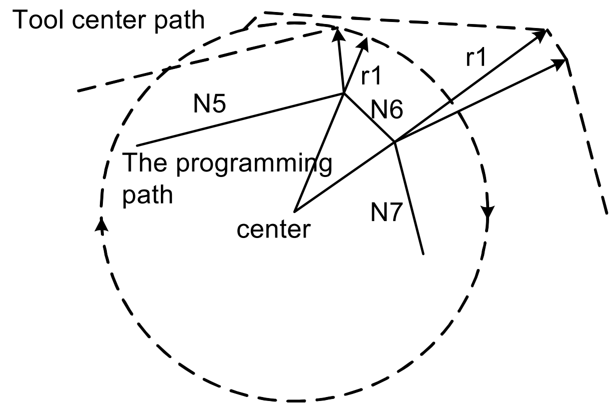
a) In the tool compensation, the tool center path traverse direction is different with that of the arc path and the difference is 90°～270°.

b) During the arc processing, except above state a, the angular difference between the tool center path starting position and its finishing position isn't same as that between the programming path starting position and its finishing position.

Example of state a:



Example of state b:



（G41）

N5 G01 G91 X8000 Y2000 D01;

N6 G02 X3200 Y-1600 I-2000 J-8000 D02;

N7 G01 X2000 Y-5000;

（D01 corresponding offset value: r_1 =2000）

（D02 corresponding offset value: r_2 =6000）

In the above example, the arc in N6 block is in one quadrant. However, after the tool compensation, the arc extends to four quadrants.

2） Rewriting the interference in advance

（a）Moving the vector about the interference

When the tool compensation is executed in the blocks A, B and C, there are the vectors V_1, V_2, V_3 and V_4 in the blocks A and B and the vectors V_5, V_6, V_7 and V_8 in the blocks B and C. Firstly, the vectors together are checked. If there is interference, they are ignored. However, it can't be ignored if the vector is the last one in the corner.

The interference before N4 and N5, check→ interference→ignore V_4 and V_5.

Check V_2 and V_6 →interference→ignore.

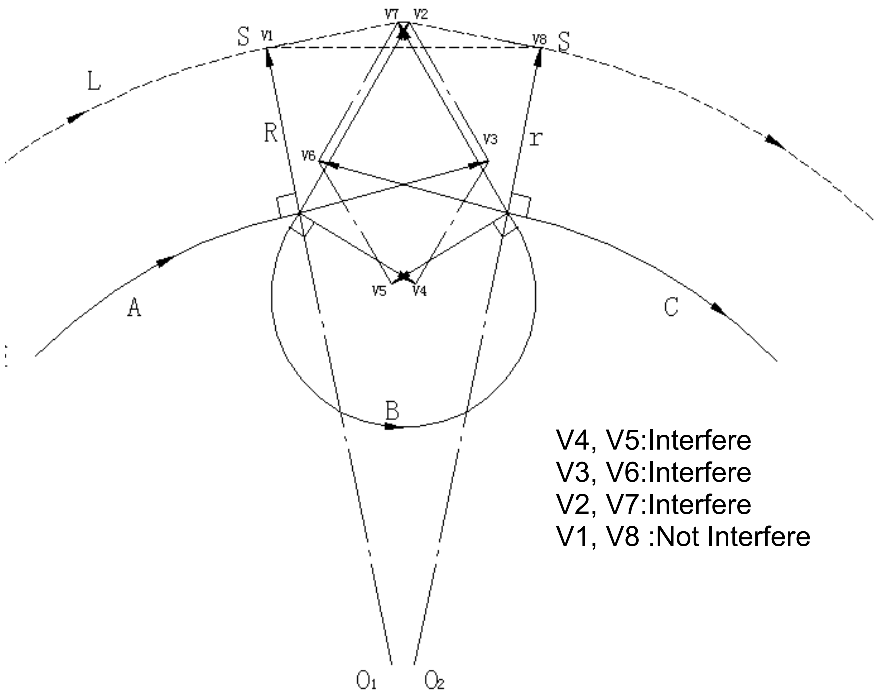
Check V_2 and V_7 →interference→ignore.

Check V_1 and V_8 →interference→not ignore.

During checking, if there isn't vector interference, the checking stops.

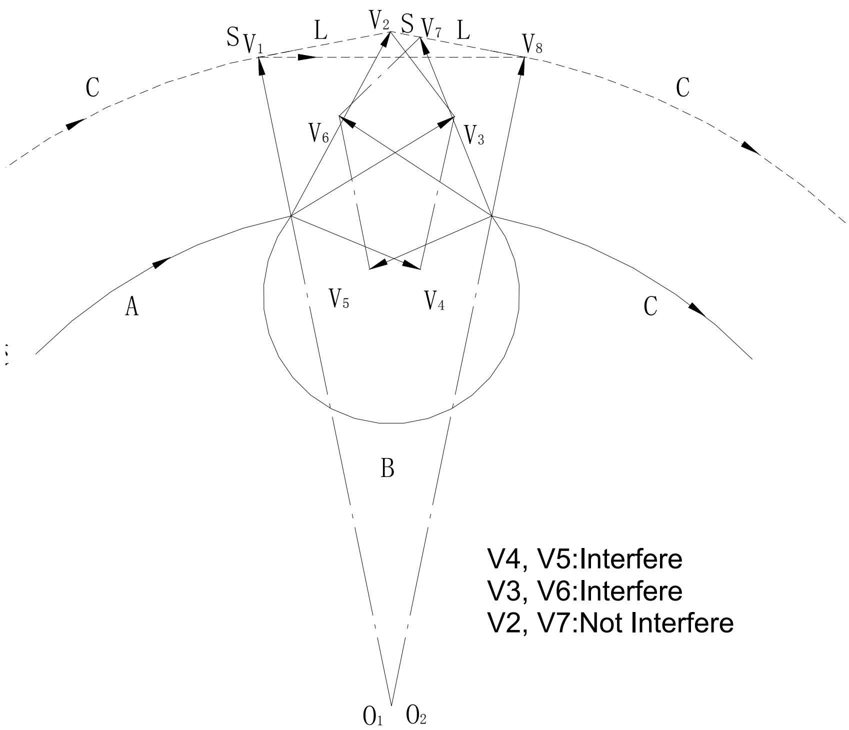
When there is one arc in block B, the arc changes into the linear motion if there is interference.

Example 1: The tool traverses from V_1～V_8 straight.



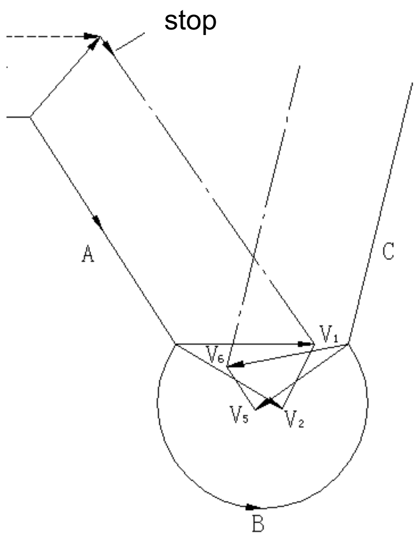
Example 2: The tool linear traverse is shown as below:

The tool path:$V_1 \rightarrow V_2 \rightarrow V_7 \rightarrow V_8$



(b) If there is interference after checking (a), the tool stops after the alarm.

If the interference is in the last vector during checking (a), or only one pair of the vectors are interfered at the starting of checking, #41 alarms and the system stops after the previous block ends.

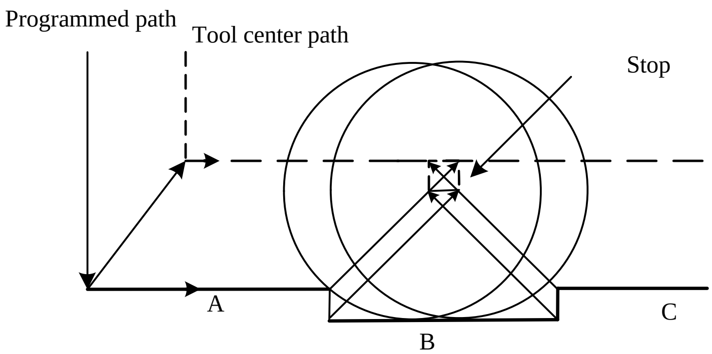


V2, V5:Interfere
V1, V6:Interfere

Due to the interference, the vectors V_2 and V_5 are ignored, the interference is still in V_1 and V_6. Then, it alarms and the system stops.

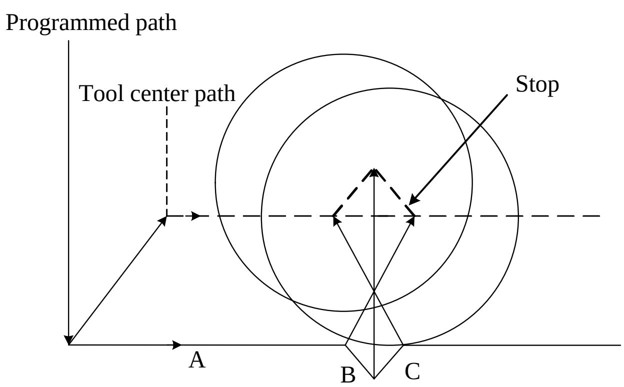
3）Even there isn't the interference, the checking is also executed, there are several examples:

（a） The concave depth is less than the tool compensation value



Although it doesn't interfer actually, the tool stops due to #41 alarm, because the tool path direction after the tool compensation is different with that of the programming path.

（b） Groove depth is less than the tool compensation value



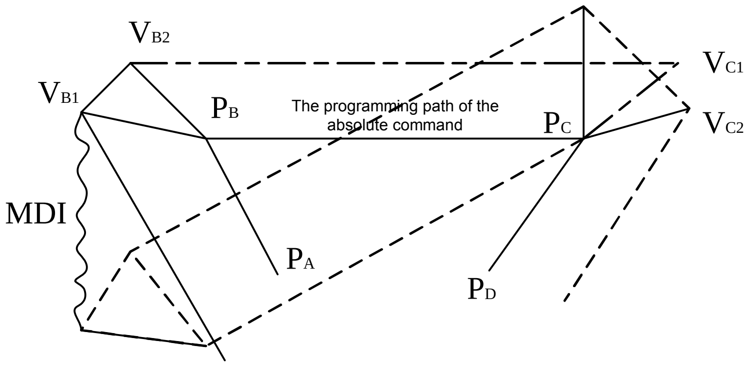
It is same as that of (a), the direction of the tool path is different with that of the programming path.

（12）MDI input commands

It doesn't compensate the command input by MDI. However, the program composed by the absolute commands is auto executed. The single block function pauses, MDI is executed, and restart, the tool path is shown as below:

In some cases, the vector at the starting position of the next block is translated and the other vectors are generated by the following two blocks.

Therefore, auto execute the compensation from position PC.



In the absolute command, P_A, P_B and P_C are specified, the tool stops at the finishing end position of the block from P_A to P_B through the single block function. Then, MDI moves the tool. The vectors V_{B1} and V_{B2} are translated into V'_{B1} and V'_{B2} and the offset vectors V_{C1} and V_{C2} in blocks from P_B——P_C and P_C——P_D should be calculated, again.

However, because vector V'_{B2} isn't calculated anymore, correctly execute the compensation from position P_C.

（13）At present, the system can't execute the tool compensation in the 4th and the 5th axes.

3.5.4 Corner Offset Arc Interpolation （G39）

Command formula: G39 or

$$G39 \begin{cases} I\_\ J\_ \\ I\_\ K\_ \\ J\_\ K\_ \end{cases}$$

Function: During the tool radius compensating, G39 is commanded in the offset mode, the corner offset arc compensation is commanded, the radius of the corner compensation is the compensation value. Whether the corner arc is valid is set by the bit parameter NO:2602#5 in the radius compensation.

Remark;

1. When the above commands are specified, the corner arc interpolation of which radius is the compensation value is can be executed.
2. G41 or G42 before the command sets the arc as CW or CCW and G39 is non mode one-shot G code.
3. When command G39 without I, J and K programs, the arc is formed at the corner;

therefore, the vector at the arc finishing end position is vertical to the starting position of the next block, which is shown as figure 3-34

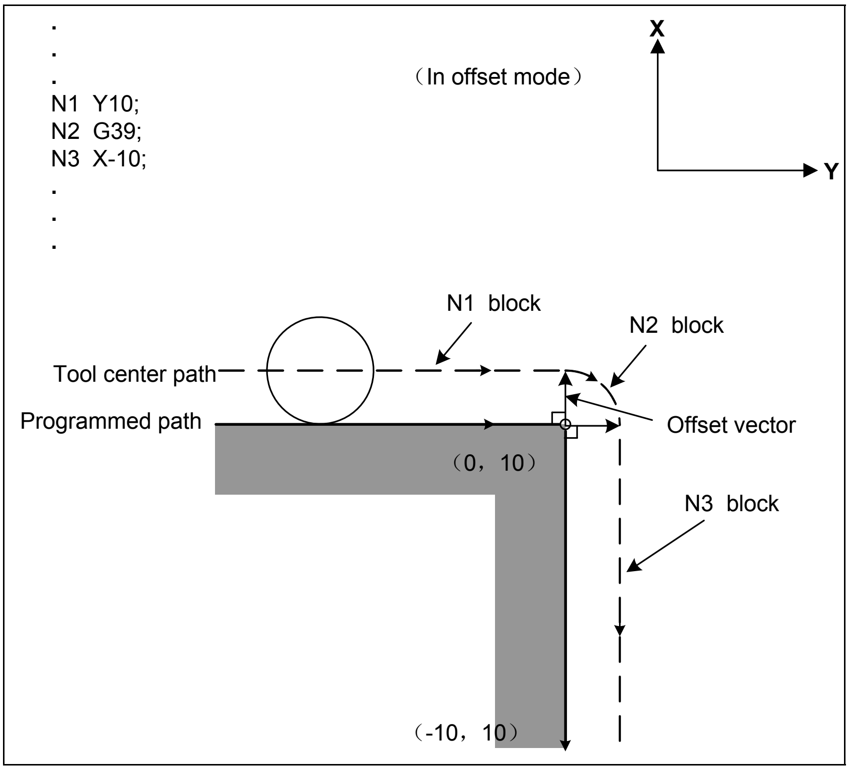


Fig. 3-34 G39 free of I,J and K

4. When G39 and I, J and K are commanded, the arc is formed at the corner, therefore, the vector at the arc finishing position is vertical to the one which is set by values of I, J and K.

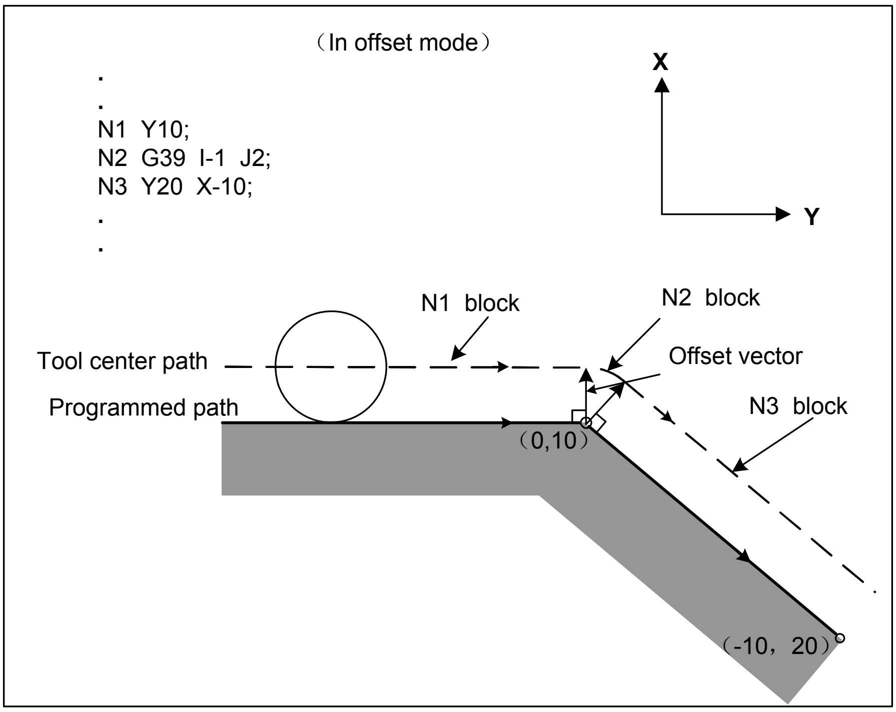


Fig 3-35 G39 with I, J and K

3.5.5 The Tool Compensation Value and Number Input the Compensation Value by the Program

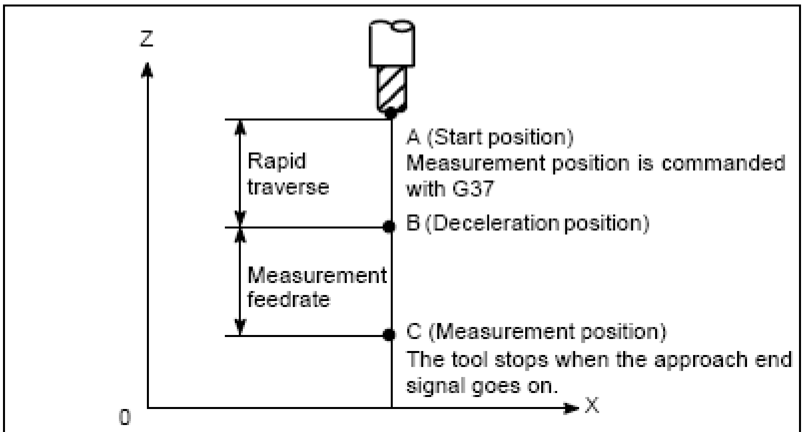
G10 L10 P_ R_ ; Command formula; geometric compensation value of H code
G10 L12 P_ R_ ; geometric compensation value of D code
G10 L11 P_ R_ ; Wearing compensation value of H code
G10 L13 P_ R_ ; Wearing compensation value of D code
P : The tool compensation number
R : The tool compensation value in the absolute value command （G90）mode
The tool compensation value in the incremental value command （G91）mode, the value adds the one of the specified tool compensation number and the sum is the tool compensation value.

Remark: The valid input range of the tool compensation value;
The geometric compensation; input ±999.999mm in the metric system; input ±99.9999 in the inch system.
The wearing compensation; input ±99.999mm in the metric system; input ±9.9999 in the inch system.

Remark**: When switch between the metric system and the inch system, whether the tool offset value is auto switched is set by bit parameter** NO2602#0.

3.5.6 Automatic Tool Length Measurement (G37)

By issuing G37, the tool starts moving to the measurement position and keeps on moving until the approach end signal from the measurement device is output. Movement of the tool is stopped when the tool tip reaches the measurement position. A difference is determined between a coordinate value obtained when the tool reaches the measurement position and a coordinate value specified by G37. The difference is then added to the tool currently used length offset value.



Compensation value = (Current compensation value) + [(Coordinates of the point at which the tool is stopped) – (Coordinates of the programmed measurement position)]

Format:

G92 IP_; Sets the workpiece coordinate system (by G54 to G59 commands).
Hoo; Specifies the offset number for tool length offset.
G90 G37 IP_; Absolute command

G37 is valid only in the block in which it is specified.

IP_ indicates the X_, Y_, Z_, or fourth axis.

Explanations

Setting the workpiece coordinate system

Set the workpiece coordinate system so that a measurement can be made after moving the tool to the measurement position. The coordinate system must be the same as the workpiece coordinate system for programming.

Specifying G37

Specify the absolute coordinates of the correct measurement position. Execution of this command moves the tool at the rapid traverse rate toward the measurement position, reduce the feedrate halfway, then continuous to move it until the approach end signal from the measuring instrument is issued. When the tool tip reaches the measurement position, the measuring instrument sends an approach end signal to the CNC which stops the tool.

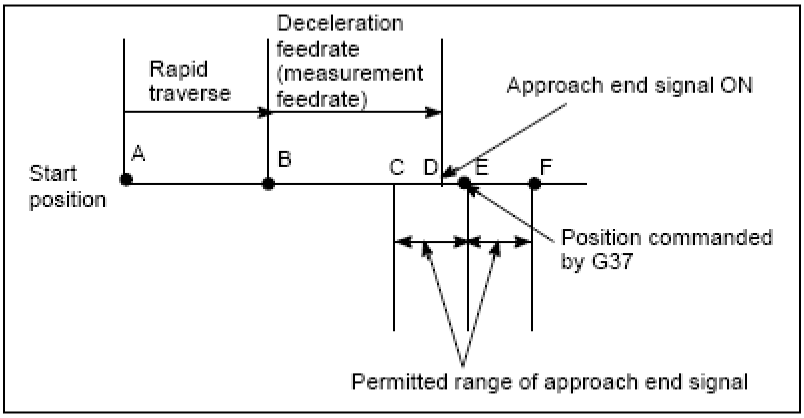
Change the offset value

The difference between the coordinates of the position at which the tool reaches for measurement and the coordinates specified by G37 is added to the current tool length offset value.

Offset value= (current compensation value)+ [（coordinate of the position at which the tool reaches for measurement）-（coordinate specified by G37）]

Alarm

When automatic tool length measurement is executed, the tool moves as shown in figure below. If the approach end signal goes on while the tool is traveling from point B to point C, an alarm occurs. Unless the approach end signal goes on before the tool reaches point F, the same alarm occurs.



Warning

When a manual movement is inserted into a movement at a measurement feedrate, return the tool to the position before the inserted manual movement for restart.

Note

1. When an H code is specified in the same block as G37, an alarm is generated. Specify H code before the G37 block.
2. The measurement speed (parameter No. 2651), deceleration position (parameter No. 2625), and permitted range of the approach end signal (parameter No. 2653) are specified by the machine tool builder.
3. Change the tool wear compensation value of H code.

the offset value is changed when tool offset A is used.
the tool wear compensation value is changed when tool offset B is used.
the wear compensation value of H code is changed when tool offset C is used.

4. The approach end signal is monitored usually every 2 ms. The following measuring error is generated:

   ERR max:Fm×1/60×Ts/1000

   Where :

   Ts::sampling period, for usual 2（ms）

   ERR max: maximum measuring error (mm)

   Fm: measurement feedrate（mm/min）

   For example, when Fm=1000 mm/min, ERR max=0.003 mm

5. The tool stops a maximum of 16 ms after the approach end signal is detected. But the value of the position at which at approach end signal was detected (note the value when the tool stopped) is used to determine the offset amount. The overrun for 16 ms is:

   Qmax= Fm×1/60×16/1000

   Qmax : maximum overrun（mm）

   Fm:measurement feedrate（mm/min）

Example:

G92 Z760.0 X1100.0; sets a workpiece coordinate system with respect to the programmed absolute zero point.

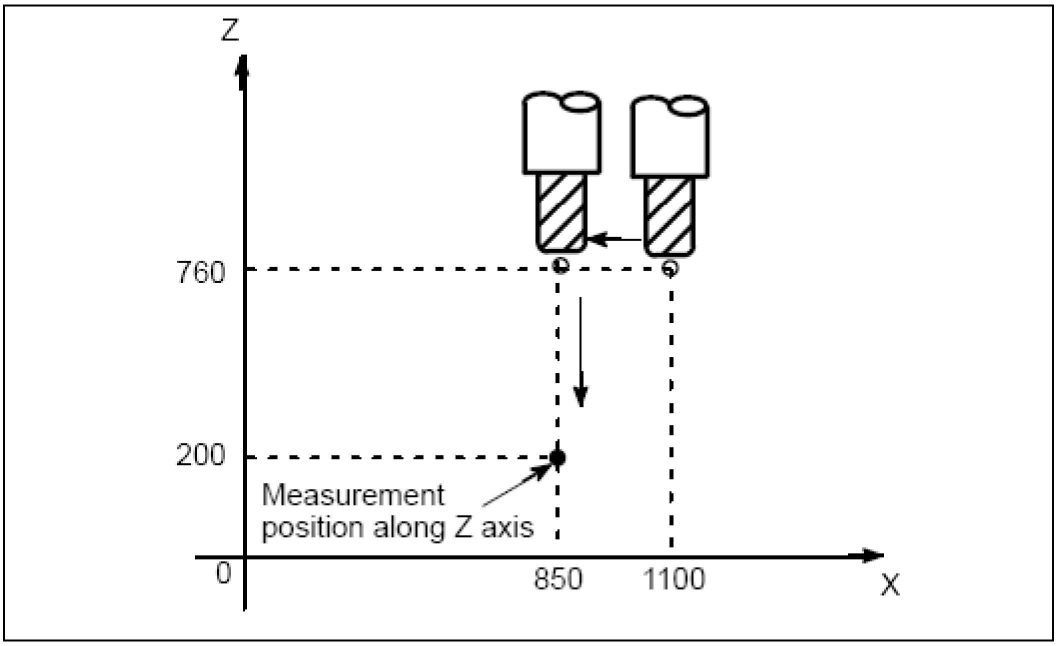
G00 G90 X850.0; moves the tool to X850.0. that is the tool is moved to a position that is a specified distance from the measurement position along the Z-axis.

H01; specifies offset number 1.

G37 Z200.0; moves the tool to the measurement position.

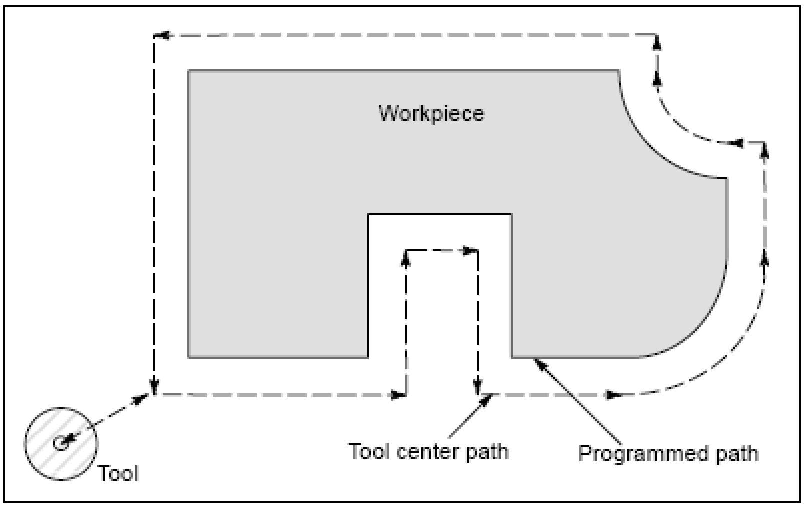
G00 Z204.0; retracts the tool a small distance along the Z-axis.

For example, if the tool reaches the measurement position with Z198.0;, the compensation value must be corrected. Because the correct measurement position is at a distance of 200mm, the compensation value is lessened to 2.00mm (198.0-200.0=-2.0).



3.5.7 Tool Position Offset (G45-G48)

The programmed travel distance of the tool can be increased or decreased by a specified tool offset value or by twice the offset value. The tool offset function can also be applied to an additional axis.



Format:

G45 IP_D_; increase the travel distance by the tool offset value
G46 IP_D_; decrease the travel distance by the tool offset value
G47 IP_D_; increase the travel distance by twice the tool offset value
G48 IP_D_; decrease the travel distance by twice the tool offset value

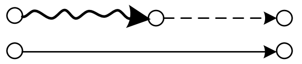
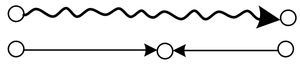
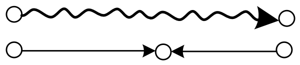
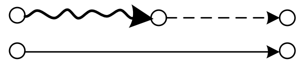
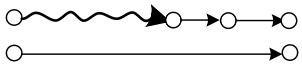
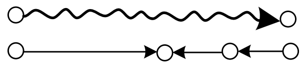
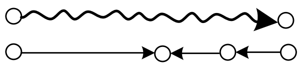
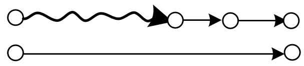
G45～G48: Non-modal G code for increasing or decreasing the travel distance
IP:Command for moving the tool
D:code for specifying the tool offset value

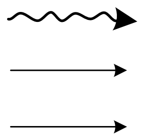
Explanations

1. Increase and decrease

As shown in the table below, the travel distance of the tool is increased or decreased by the specified tool offset value. In the absolute mode, the travel distance is increased or decreased as the tool is moved from the end position of the previous block to the position specified by the block containing G45 to G48.

Table 3-20 Increase and decrease of the tool travel distance

G code	When a positive tool offset value is specified	When a negative tool offset value is specified
G45	 Start position End position	 Start position End position
G46	 Start position End position	 Start position End position
G47	 Start position End position	 Start position End position
G48	 Start position End position	 Start position End position



Programming movement distance

Tool offset value

Actual movement position

If a move command with a travel distance of zero is specified in the incremental command (G91) mode, the tool is moved by the distance corresponding to the specified tool offset value.
If a move command with a travel distance of zero is specified in the absolute command (G90) mode, the tool is not moved.

2. Tool offset value

Once selected by D code, the tool offset value remains unchanged until another tool offset value is selected.

Table 3-21 Range of tool offset values

	Metric input	Inch input
Tool offset value	0 to ±999.999mm	0 to ±999.999 inch
	0 to ±999.999 deg	0 to ±999.999 deg

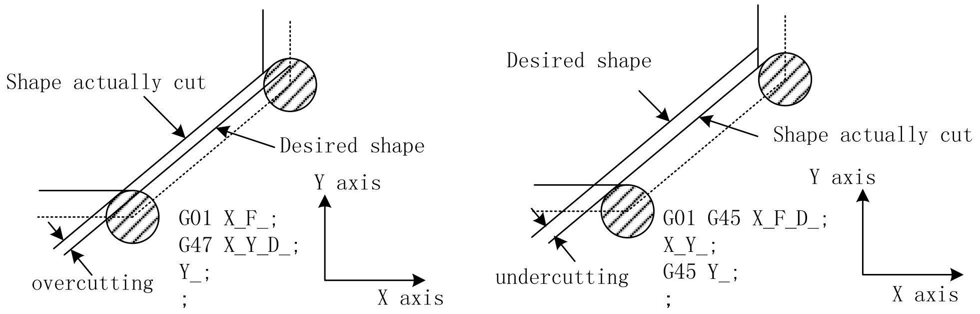
D0 always indicates a tool offset value of zero.

Warning!

1. When G45 to G48 is specified to n axes (n=1-6) simultaneously in a motion block, offset is applied to all n axes.

When the cutter is offset only for cutter radius or diameter in taper cutting, overcutting or undercutting occurs.

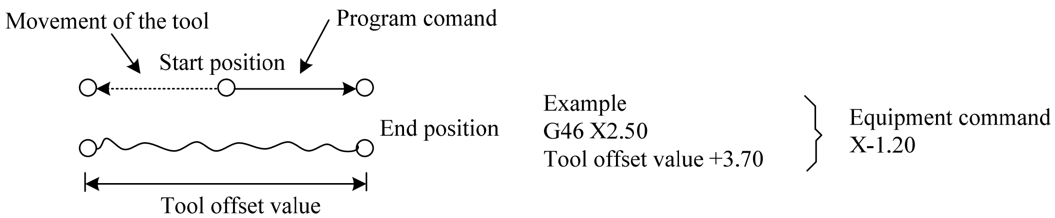
Therefore, use the cutter compensation (G40 or G42).



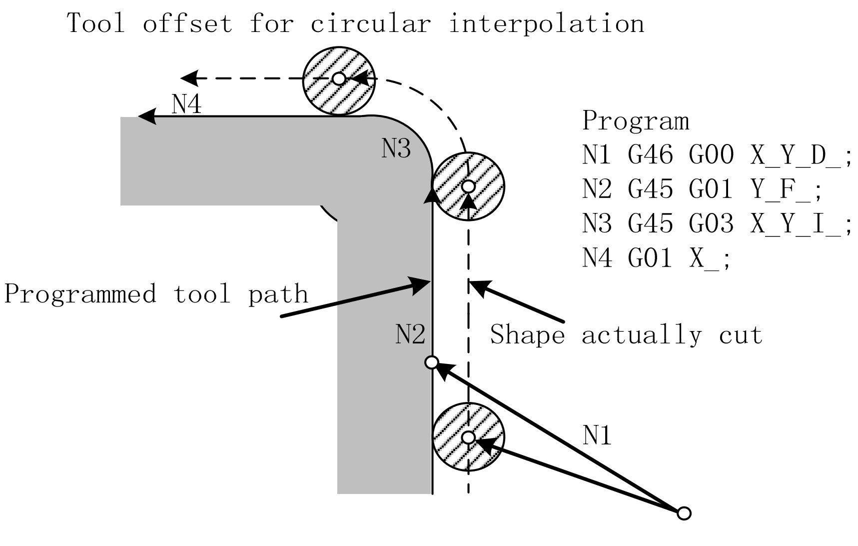
2. G45 to G48 (tool offset) must not be used in the G41 or G42 (cutter compensation) mode.

Note:

1. When the specified direction is reversed by decrease as shown in the figure below, the tool moves in the opposite direction.

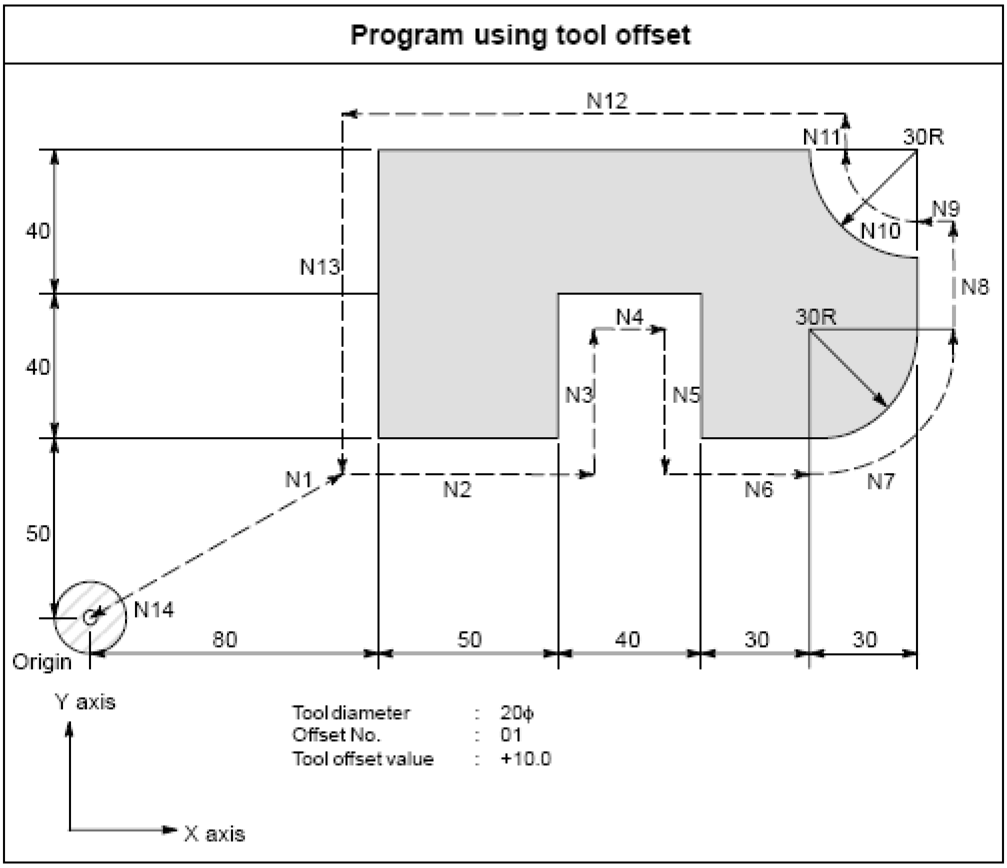


2. Tool offset can be applied to circular interpolation (G02, G03) with the G45 to G48 commands only for 1/4 and 3/4 circles using addresses I,J and K by the parameter setting, providing that the coordinate rotation be not specified at the same time. This function is provided for compatibility with the conventional CNC tape without any cutter compensation. The function should not be used when a new CNC program is prepared.



3. D code should be used in tool offset mode (G45 to G48). However, H code can be used by setting the parameter TPH (No. 5001 #5) because of compatibility with conventional CNC tape format. The H code must be used under tool length offset cancel (G49).
4. G45 to G48 are ignored in canned cycle mode. Perform tool offset by specifying G45 to G48 before entering canned cycle mode and cancel the offset after releasing the canned cycle mode.

Examples:



Program

N1 G91 G46 G00 X80.0 Y50.0 D01 ;

N2 G47 G01 X50.0 F120.0 ;

N3 Y40.0 ;

N4 G48 X40.0 ;

N5 Y-40.0 ;

N6 G45 X30.0 ;

N7 G45 G03 X30.0 Y30.0 J30.0 ;

N8 G45 G01 Y20.0 ;

N9 G46 X0 ; (decreases toward the positive direction for movement amount “0”. The tool moves along the -X direction by the offset value.)

N10 G46 G02 X-30.0 Y30.0 J30.0 ;

N11 G45 G01 Y0 ; (increase toward the positive direction for movement amount “0”. The tool moves along the +Y direction by the offset value.)

N12 G47 X-120.0 ;

N13 G47 Y-80.0 ;

N14 G46 G00 X80.0 Y-50.0 ;

3.6 The Special Canned Cycle Commands

The special canned cycle and the standard canned cycle are used in combination. Before using the canned cycle, the canned cycle selects G commands and the hole processing data for programming, and the hole processing data are saved.（Without the positioning data, the canned cycle isn't executed, only the data are saved. ）Even after the special canned cycle is executed, the saved standard canned cycle still remains before canceling. If it isn't in the canned cycle mode, only the positioning is executed rather than the drilling during specifying the special canned cycle.

Based on the different function of the continuous drilling, this chapter mainly introduces the path of the circle, the straight line, the arc, the chess board or the rectangle to call the canned drilling mode cycle for the drilling holes cycle in the consecutive space.

3.6.1 Circumference Holes Cycle(G120)

G120 X_ Y_ I_ J_ K_ ;

X,Y : The center position of the circumference hole cycles affected by G90/G91.

I : he radius r of the circle, the unit is based on the input setting unit and represented by the positive number.

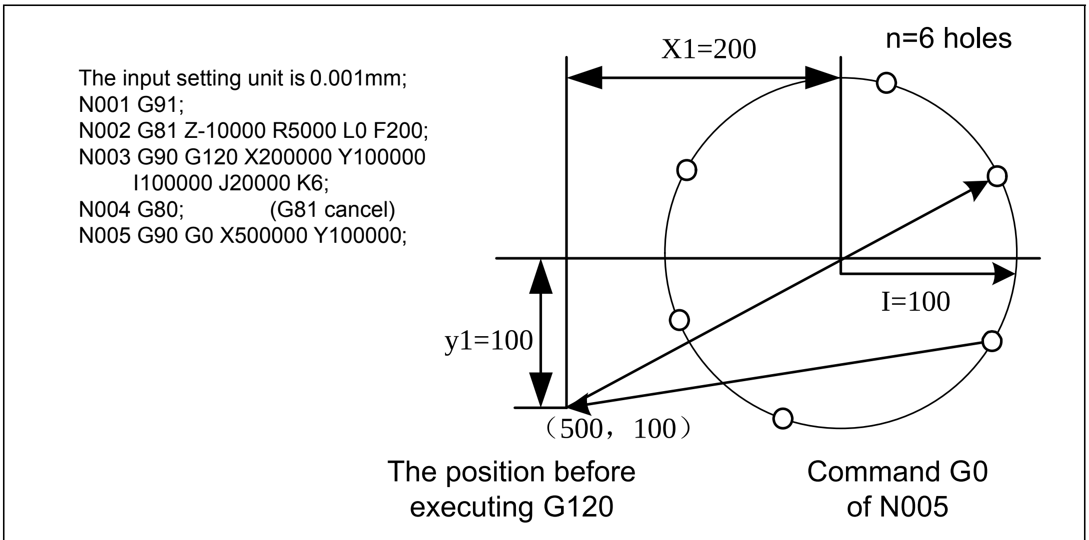
J : The angle of the initial drilling hole position is positive in CCW direction.

（The position of the decimal position is the degree; if there isn't the decimal position, 0.001 is the unit.）

K: The number of the drilling holes is n. The specified quantity is 1～9999 rather than 0. When 0 is specified, P221 alarms:the canned hole number is 0.

Take the coordinate specified by X and Y as the center to form the circumference of radius R, and the circumference is divided equally based on X axis and the angle to drill n holes. The drilling in each hole position saves G81 drilling data of the standard canned cycle.

The movement in the hole position is processed in G00 mode. Moreover, after G120 command ends, the data are not saved.



3.6.2 The Angle Straight Hole Cycle（G121）

G121 X_ Y_ I_ J_ K_ ;

X,Y : The coordinate of the starting position is affected by G90/G91.

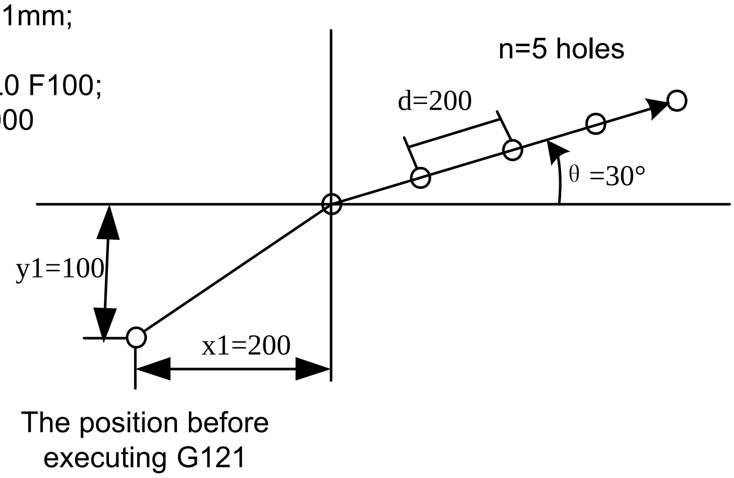
I : The unit of the interval d is based on the input unit; when d is negative value, the starting position is taken as the center and the hole is drilled in the symmetrical direction.

J : The angle is positive in CCW direction. （The position of the decimal position is º; if there isn't the decimal position, it is 0.001º. ）

K : The quantity of the holes n, which includes the starting position, and its range is 1～9999.

It's assumed that the position specified by X and Y as the starting position, the direction formed by X axis and the angle is differed by the interval d and divided drilling hole movement of n times. Based on the standard canned cycle, the data of the drilling holes（the hole processing mode and data）should be saved before drilling in each hole. The movement of each hole position is processed in G00 mode; moreover, after G121 command ends, the data are not saved.

The input setting unit is 0.001mm;
N001 G91;
N002 G81 Z-10000 R5000 L0 F100;
N003 G121 X200000 Y100000
I100000 J30000 K5;



Remark：

1. When K command is K0, K isn't specified or it's out of the range, the program is wrong and the error number is "P221".
2. If G commands of group 0 is with G121 in one block, the following commands are priority.
3. If G121 and G28 are in one block, G121 is ignored and G28 is executed.
4. Commands G72～G89 is with command G121 in one block, the canned cycle is invalid and G121 is executed.

3.6.3 Arc Hole Cycle （G122）

G122 X_ Y_ I_ J_ P_ K_ ;

X,Y : The center coordinate of the arc is affected by G90/G91.

I : The unit of the arc radius r is based on the setting unit and represented in the positive number.

J : The angle of the initial drilling hole position is θ and is positive in CCW direction.
（The position of the decimal position is º ; the unit is 0.001º without the decimal position. ）

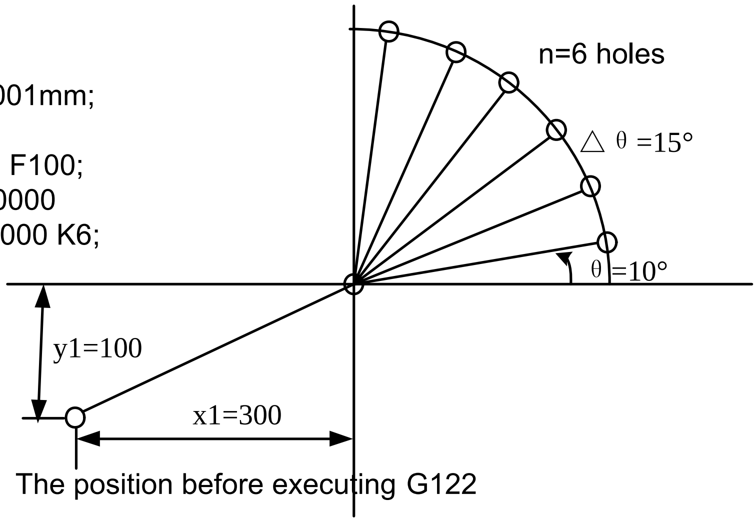
P : The angle interval is θ, and drill the holes positively in CCW direction and negatively in CW.
（The position of the decimal position is º ; the unit is 0.001º without the decimal position.）

K : The quantity of n and the specified range is 1～9999.

The coordinate specified by X and Y is taken as the center to form the circumference of radius r, and from the position set by X axis and the angle, n holes are drilled in △ space. The drilling in each hole is same as that of the circumference hole in cycle, so the drilling hole data should be saved in

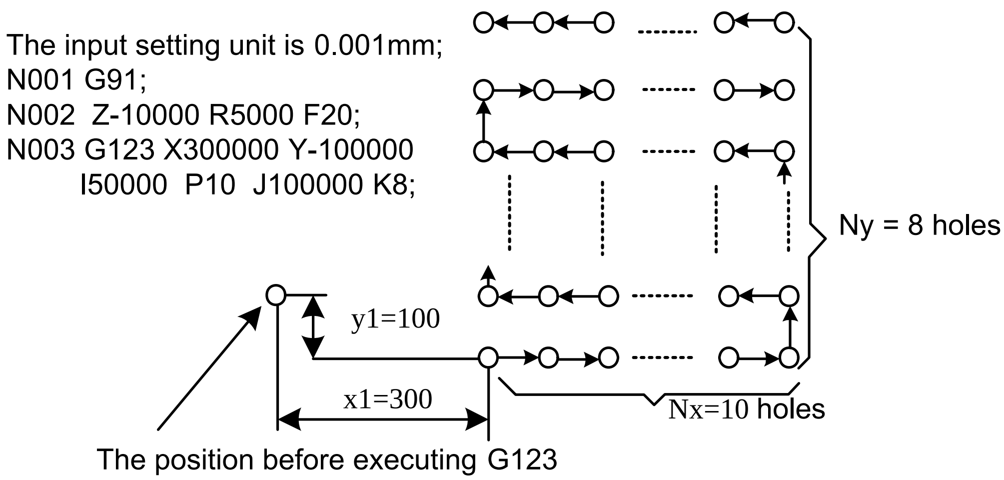
advance during the standard canned cycle.
The movement of the hole position is executed in G00 mode; moreover, the data aren't saved after G122 specifying end.

The input setting unit is 0.001mm;
N001 G91;
N002 G81 Z-10000 R5000 F100;
N003 G122 X300000 Y100000
I300000 J10000 P15000 K6;



3.6.4 The Chess Board Hole Cycle（G123）

G123 X_ Y_ I_ P_ J_ K_ ;
X,Y : The coordinate of the starting position is affected by G90/G91.
I : The interval of X axis is △x. The unit is based on the setting unit. When △x is positive, it is divided in the positive direction from the starting position; negative, in the negative from the starting position.
P : The quantity is nx in X axis direction and its range is 1～9999.
J : The interval of Y axis is △y. The unit is based on the setting unit. When △y is positive, it is divided in the positive direction from the starting position; negative, in the negative from the starting position.
K : The quantity is ny in Y axis direction and its range is 1～9999.
The position specified by X and Y is taken as the starting position, interval △x is taken as nx grid for drilling holes in the direction parallel to X axis. The drilling hole data（the hole processing mode and data）should be saved in advance because the drilling in each hole uses the standard canned cycle. The movement in each hole is processed in G00 mode. Moreover, the data are not saved after G123 command ends.



3.6.5 Continuous Drilling in the Rectangle（G124/G125）

Command formula;

G124

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

G125

Command function; Based on the number of drilling holes in each side, the holes are drilled continuously in each side of the rectangle.

Command introduction:G124 —drill holes in CW direction

G125 —drill holes in CCW direction

Gxx —Drilling mode（G73, G74,G81,G82, G83, G84,G85, G86, G88,G89）

X,Y —The finishing end position coordinate of the 1st rectangular side

R — R Plane R position

Z —The hole depth

P—The quantity of the drilling holes on the 1st and the 3rd sides

K —The quantity of the drilling holes on the 2nd and the 4th sides

I － The length of the 1st and the 3rd sides

J —The length of the 2nd and the 4th sides

F —Cutting feed rate

Relative parameter**:**

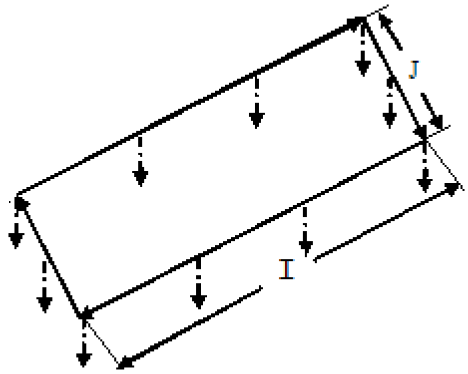
1:The hole positioning of the continuous drilling holes is processed based on the cutting path（G01~G03）.

0: The hole positioning of the continuous drilling holes is processed based on the rapid path（G01~G03）.

For example:The drilling holes on the rectangular path, the starting position coordinate of the 1st side is X90,Y40; the length of the 1st side is 40mm; the length of the 2nd side is 10mm. G81 drilling mode:drill three holes in the 1st and the 3rd sides; the hole depth is 25mm;

The programming is shown as below:

G90 G17 G0 X0 Y0 Z25 M03 F1000;
G124 G81 X90 Y40 R5 Z-25 I40 J10 P3 K2 F800;
G80 G0 X100 Y100 M05;



3.6.6 Milling on the Plane （G126/G127）

Command formula:

G126
X_ Y_ Z_ R_ I_ J_ L_ F_
G127

Command function; The plane is milled based on the specified length and width of each side.
Command introduction:

- G126 —Milling back and forth
- G127 —Milling in one-way
- X,Y —The coordinate of the starting position
- Z —The cutting down length in Z axis direction
- R —RR plane position
- I — The width in X axis direction
- J — The width in Y axis direction
- L —The cut width increment on X and Y planes should be less than the tool diameter more than 0.
- F —Cutting feed rate

Example 1:Milling back and forth on the plane requires the starting coordinate is（0, 0）, the cutting down length in Z axis direction is 25mm, the width in X axis direction is 90mm, the width in Y axis direction is 70mmj, the cut width on the planes of X and Y is 10mm.

The programming is shown as below:

G90 G17 G0 X0 Y0 Z25 M03 F1000;
G126 X90 Y40 Z-10 R5 I70 J30 L10 F800;
M30;

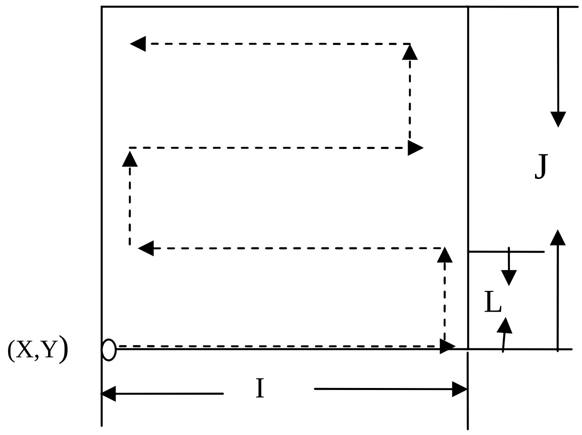


Fig 3-36 G126 instance graph

Example 2:Milling in one-way on the plane requires the starting coordinate is（0, 0）, the cutting depth in Z axis is 25mm, the width in X axis is 90mm, the width in Y axis is 70mm, the cut width on the planes of X and Y is 10mm.

The programming is shown as below;

G90 G17 G0 X0 Y0 Z25 M03 F1000;

G127 X90 Y40 Z-10 R5 I70 J30 L10 F800;

M30;

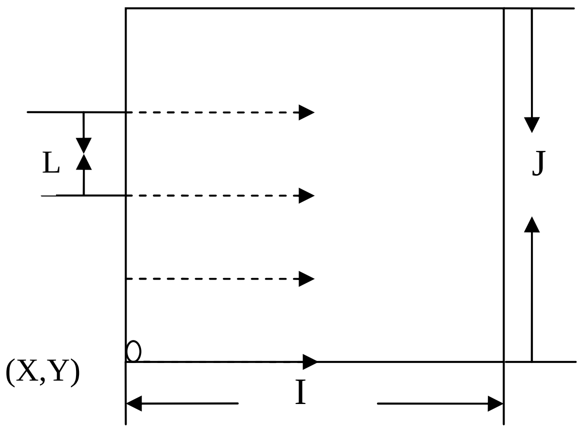


Fig. 3-37 G127 instance graph

Remark:

1. If G124 or G125 is commanded in the canned cycle, it means the continuous drilling on the rectangular path. The rectangular data are set by value J and X and Y coordinate commanded in the block, and the continuous drilling cycle is executed in the drilling hole mode （the canned cycle command）.

2. The maximum command value of the drilling hole number A and B in each side is 9999; the negative value is invalid. If the decimal is commanded, the decimal part is ignored; if A or B isn't specified, 0 is defaulted.

3. The rectangle is set by the current starting position, the finishing end position of the 1st side and the length of the 2nd side; if the finishing end position of the 1st side isn't specified, the current starting position is defaulted; it alarms if the length （value J）of the 2nd side isn't specified.

4. During the continuous drilling holes, the returned planes are point R ones. Only the last hole is processed, return to relative plane based on G98/G99.

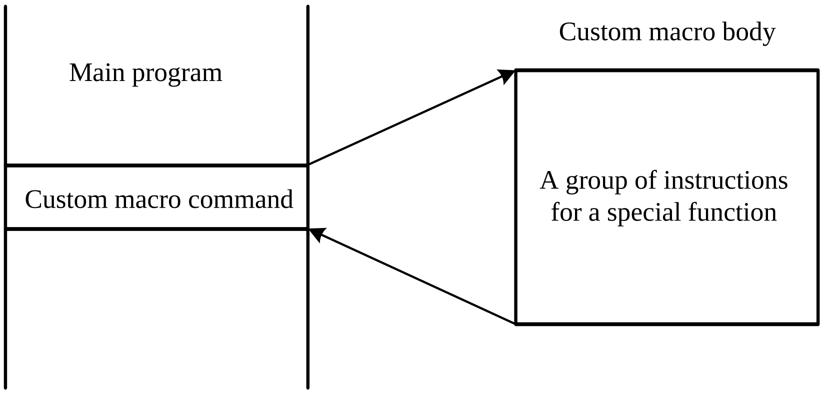
5.The canned cycle G110, G111, G112, G113, G114, G115, G130, G131, G132, G133, G134, G135, G136, G137 isn't with the function of continuous drilling holes.

6. The command character G124, G125, A, B and J are valid in the current block. It alarms if G124 and G125 are commanded without the canned cycle（the drilling mode）command, it alarms; A, B and K are ignored if A, B and K are specified while G124 or G125 isn't. specified

3.7 Macro Function

3.7.1 The User Macro Program General Introduction

The macro program uses the variable, the calculation commands and the control commands, etc as the dedicated control function in the mode of the subprogram, which can be called by the user. In the main program, the dedicated control function (the macro program) calls the commands through the macro program and uses them based on the requirement. The macro program is with the capacity and the flexibility which the standard G codes is lack of. Through the combination of the variable commands, commands of calling, various calculation, input and output the data between PLC and the macro program, control, determining and branch, etc, and the measuring can be executed in the macro program.



3.7.2 The Variable

The common processing program can directly specify G codes and the movement distance through the numerical value; for example, when G01 and X200.0 use the user macro program, the numerical value can be directly specified or through the variable. The variable value can be rewritten through the program or MDI panel.

```
#1=#2+200;
G01 X#1 F300;
```

3.7.2.1 The Variable Formula Representation

The variable is composed by the variable code （#） and its following variable number.

When the variable number is the numerical value;

#i （i=1, 2, 3, 4, 5……）

Example 1 #5
#109
#1005
—#20

The following formula can also be used, and the figure is replaced by the expression formula.

#〔the expression >〕

Example 2 #〔#100〕
#〔#1001-1〕
#〔# 4/2〕

The variable # i in the manual all are replaced by #〔<the expression formula>〕

3.7.2.2 Quoting the Referencing Variable

After the address, the specified variable number can quote its variable value. When the expression formula specifies the variable, the expression formula should be bracketed.

For example;

G01X[#1+#2]F#3;

The quoted referenced variable value auto rounds up is automatically rounded based on the minimum setting unit least input increment of the address.

For example

When the unit of the minimum input increment is 1/1000mm, G00X#1 is commanded, and 12.3456 is assigned to the variable #1, the actual command value is G00X12.346.

If the code of the quoted variable value is rewritten, the negative sign （—）should be in front of #.

For example:

G00X—#1;

When the variable isn't defined, the variable and the address character all are ignored.

For example;

When the variable value #1 is 0, and the value of the variable #2 is void null, the executing result of G00X#1 Y#2 **is** G00X0.

In the program, when the variable is defined, the decimal position can be omitted.

For example;

Defining #1=133:The variable #1 is quoted, its actual value is 133.000.

The program number, the serial number and the optional jumping number in any block can't use the variable.

For example;

The variable can't be used in the following situations;

○#1;

/#2G00X100.0;

N#3Y200.0;

3.7.2.3 Undefined Variable

The variable value which isn't defined is called as the void null value. #0 is always used in the void variable, which can be read rather than written.

The undefined variable is with the following characteristics：

（1）Quotation

When one undefined variable is quoted, the address itself is ignored.

#1=<void value >	#1=0
G90 X100 Y#1 ↓ G90 X100	G90 X100 Y#1 ↓ G90 X100 Y0

（2）Calculation

Except <void value vacant> is replaced,

#1=< void value >	#1=0
#2=#1 ↓ #2=< void value >	#2=#1 ↓ #2=0
#2=#1*5 ↓ #2=0	#2=#1*5 ↓ #2=0
#2=#1+#1 ↓ #2=0	#2=#1+#1 ↓ #2=0

（3）Conditional expression

In the cases of E Q and N E, <void value> and 0 are determined as the different values.

In the cases of GE, GT, LE and LT, <void value> and 0 are determined as the same value.

#1=<void value >	#1=0
#1EQ#0 ↓ Definable	#1EQ#0 ↓ Indefinable
#1 NEO ↓ Definable	#1 NEO ↓ Indefinable
#1GE#0 ↓ Definable	#1GE#0 ↓ Definable
#1GT0 ↓ Indefinable	#1GT0 ↓ Indefinable（not established）

3.7.2.4 Display and Setting the Variable

The variable value can display on LCD and set in MDI mode.

he variable value on LCD is blank, it means the variable is void;

 The variable value displays as ******** on LCD, it means overflow when the absolute value of the variable is above 99999999 or underflow when the absolute value of the variable is less than

0.0000001.

3.7.2.5 The Solution Range of the Variable

The range of the part variable and the public variable is -999999～+999999, it alarms if it's out of the range.

3.7.3 Types of the Variable

The variables are classified into the part variable, the common variable and the system variable, the purposes and the characteristics of the variables in each type are different.

3.7.3.1 The Part Local Variables # 1～# 33

They can be defined as the independent variable argument during calling the macro program, also used as the part local variables between the main program and the subprograms. The part local variable is independent in the macro program of each layer and used repeatedly but not more than four layers.

When the macro program is called, the independent argument variable assigns a value to the part local variable, the part variable which isn't assigned a value is used by the user at random. About the corresponding relation between the part variable and the independent variable, refer to the chapter of calling the macro program.

3.7.3.2 The Public Common Variable #100～ #199 , #500～ #999

The meanings of the common variable in different macro programs are same, one defined public variable is in common use in the main program, or the subprogram called from the main program and the macro program.

When power is off, the variable #100—#199 is initialized as void.

The data of the variable #500——#999 are saved, it doesn't get lost even the power is off.

3.7.3.3＃1000－ System variable

The system variables are the ones of the canned purpose in the system, and it is classified into three types of reading, writing and reading and writing. Read and write the various data of CNC, for example, the current position data and the compensation value of the tool. The system variable is the base of the auto control and the common program development.

The interface signal is from #1000 to #1031 and from #1032 to #1035, from #1100 to #1131 and from #1132 to #1135.

The interface signal is the one interchanging between the programmable machine controller（PLC）and the user macro program.

List 3-8 The system variable of the interface signal

The system variable number	Property	Function
#1000—#1031	Read	The signal in 32 digits is sent from PLC to the user macro program, and the signal is read from the variable #1000 to #1031 based on the bit, the interface input signal is from UI000 to UI031.
#1100—#1131	Read/write	The signal in 32 digits is sent from the user macro program to PLC, and the signal is written from the variable #1100 to #1131 based on the bit, the interface input signal is from UI000 to UI031.
#1032 —#1035	Read	The signal in 32 digits is output from PLC to the variable of the user macro program and the span of the variable value is from —99999999 to +99999999.
#1132 —#1135	Read/write	The signal of 32 digits is written into the variable of the user macro program and the variable value range is from -99999999 to +99999999.

（the tool offset value）#2001～#2400

The system variable can read and write the tool compensation value.

The usable variable number is set by the tool compensation number, the appearance geometric compensation and that of the wearing are differed and the tool length compensation and that of the tool radius are also differed.

When the offset group number is less or equal to 200, #2001～#2400 can also be used.

List 3-9 The system variable of the tool compensation value

Compensation number	Property	The tool length compensation（H）		The tool radius compensation（D）	
		Appearance geometric compensation	Wearing compensation	Appearance geometric compensation	Appearance wear compensation
1 : 201 : 400	Read/write	#11001 （#2201） : #11201 （#2400） : #11400	#10001 （#2001） : #10201 （#2200） : #10400	#13001 : : : #13400	#12001 : : : #12400

For example: #30=#2005

In the tool offset number, the tool offset value is substituted into the variable #30.

When the offset value is 1.500mm, the value of #30 is changed into 1.5.

#2210=#8

The offset value of the current offset number #10 is written and equal to that of #8 variable.

（3）Macro program alarm #3000

Only the variable is written, when the value of the variable #3000 is 0～200, CNC stops running and alarms. After the expression formula, the alarm information within 26 characters is specified. The alarm number and information display on CRT screen, and the value of which alarm number is #3000 adds 3000.

For example :

#3000=1（The tool can’t be found.）

on the alarm screen.

（4）The time information #3001, #3002, #3011, #3012

List 3-10　The system variable list of the time information

VARIABLE NUMBER	PROPERTY	FUNCTION
#3001	Read/write	The variable is a counter, and one minisecond is the unit. When it powers on, the variable value is reset as 0. When it reaches millsecond of 2147483648, the value of the counter returns to 0.
#3002	Read/write	The variable is a counter, and one hour is the unit. Start counting when the auto running starts, and the counter also saves the numerical value even it powers off. When it reaches 9544.371767 hours, the value of the counter returns to 0.
#3011	Read/write	The variable is for reading the current date（year/month/day）. The information of year/month/day is switched in the decimal system.
#3012	Read/write	The variable is to read the current time　（hour/minute/second）. The information of hour/minute/second is switched in the decimal system. For example, pm 15:34:56 is represented as 153456.

（5）Prohibition of stopping the single block and waiting the miscellaneous function finish signal#3003

List 3-11 System variable of the auto running control（#3003）

#3003	**PROPERTY**	**SINGLE BLOCK**	**FINISH SIGNAL OF THE MISCELLANEOUS FUNCTION**
0	Read/write	Valid	Wait
1	Read/write	Invalid	Wait
2	Read/write	Valid	Not wait
3	Read/write	Invalid	Not wait

Remark Notice;

·When the power is on, the variable value is 0.

·When stopping the single block is invalid, even the single block switch is ON, the single block doesn’t stop.

·When the no waiting miscellaneous function（M, S and T function）is not specified end, the program executes the next block before the miscellaneous function ends. Moreover, the distribution finish signal DEN doens't output.

· The variable value of #3003 can be cleared through resetting.

For example; The drilling holes in cycle（relative to the incremental programming）is equivalent to G81.

Macro program calling commands

G65　P9081L（repeated times）R（point R）Z（point Z）;

Editing the macro program itself is shown as below：

09081;

#3003=1;
G00 Z#18;
G01 Z#26;
G00Z-[ROUND（#18）+ROUND（#26）];
#3003=0;
M99;
The single block doesn’t stop, #18 is relative to R, #26 to Z.

（**6**）Feed hold. The valid and invalid conditions for the feed rate override and exactly stop **#3004.**

#3004	PROPERTY	FEED HOLD	FEED RATE OVERRIDE	EXACT STOP
0	Read/write	Valid	Valid	Valid
1	Read/write	Invalid	Valid	Valid
2	Read/write	Valid	Invalid	Valid
3	Read/write	Invalid	Invalid	Valid
4	Read/write	Valid	Valid	Invalid
5	Read/write	Invalid	Valid	Invalid
6	Read/write	Valid	Invalid	Invalid
7	Read/write	Invalid	Invalid	Invalid

Remark：

·When it powers on, the variable value is 0.

·When the feeding pause hold is invalid：

a. When the feed hold button is pressed, the machine stops in single block stopping mode. However, when the variable #3003 makes the single block mode invalid, the single block doesn’t stop.

b. The feeding pause indicator is on when the feeding pause button is released after being pressed. However, the machine doesn’t stop; the program continues executing, and the machine stops in the first block which the feeding pause is valid.

·When the feed rate override is invalid, the override is always 100% and it doesn’t have any connection with the feed rate override switches on the machine operational panel.

·When exact stop detection is invalid, even the block which doesn’t execute the cutting doesn’t execute the exact stop detection（position detection）.

For example; The tapping cycle （Relative to the incremental programming）（equivalent to G84）
The macro program calling commands

G65 P9084 L（Repeated times）R（point R）Z（point Z）;

Editing the macro program itself is as below：

09084;
#3003=1; : Prohibit stopping the single block
G00Z#18;
#3004=7;
G01Z#26;
M05; The feed hold, the feed rate override and the exact stop checking are invalid.
M04;

Z-#26;
#3004=0;
M05;
M03;
G00Z-#18;
#3003=0;
M99;

（7）Stopping and information display ＃3006

SYSTEM VARIABLE NUMBER	PROPERTY	FUNCTION
#3006	Write	In the macro program, when"#3006=1（MESSAGE）; "is commanded, the program stops after the previous block ends. The message in maximum 26 characters is specified in one block and bracketed by the control input "（" and output "）", and the relative information displays on the operation information screen.

（8）Number of parts #3901,#3902

The part number （target number）and the processed part number （processed number） can be read and written.

VARIABLE NUMBER	PROPERTY	FUNCTION
#3901	Read/write	The processed part number （the processed number）
#3902	Read/write	The required part number（target number）

Remark Notice; The numerical value can't be negative.

（9）The mode information #4001～#4130

The mode information before the processing block can be read.

SYSTEM VARIABLE	PROPERTY	MODE INFORMATION	GROUPS
#4001	Read	G00, G01, G02, G03, G33	01
#4002	Read	G17, G18, G19	02
#4003	Read	G90, G91	03
#4004	Read	04	04
#4005	Read	G94, G95	05
#4006	Read	G20, G21	06
#4007	Read	G40, G41, G42	07
#4008	Read	G43, G44, G49	08
#4009	Read	G73, G74, G76, G80~G89	09
#4010	Read	G98, G99	10

#4011	Read	G50, G51	11
#4012	Read	G65, G66, G67	12
#4013	Read	G96, G97	13
#4014	Read	G54~G59	14
#4015	Read	G61~G64	15
#4016	Read	G68, G69	16
. . .	Read	. . .	. . .
. . .	Read	. . .	. . .
#4022	Read	G50.1, G51.1	22
#4107	Read	codes D	
#4109	Read	codes F	
#4111	Read	codes H	
#4113	Read	codes M	
#4114	Read	serial number N	
#4115	Read	program number O	
#4119	Read	S codes	
#4120	Read	T codes	
#4130	Read	P Additional work piece coordinate system number P	

For example：The combined programming of the incremental value/the absolute value, the boring hole cycle（equivalent to G86）

The macro program calling commands

G65 P9086L（Repeated times）R（point R）Z（point Z）:

The macro program itself is edited as below：

```
09086;
#1=#4003;        : Save G codes in group 03
#3003=1;         : Prohibit stopping the single block
G00 G91 Z#18;
G01  Z#26;
M05;
G00 Z-[#18+#26];
M03;
#3003=0;
G#1 M99;         : Restore G codes in group 03
```

The system variables #4001～#4130 can’t use in the items in the left of the operational commands.

（10）The position information #5001～#5105

The position information can be set by the system variables #5001～5105 and its unit is the millimeter or the inch set by the input system. The system variables #5001～5105 can’t use in the items in the left of the operational commands.

SYSTEM VARIABLE	PROPERTY	POSITION INFORMATION	COORDINATE SYSTEM	TOOL COMPENSATION VALUE	READ DURING MOVING
#5001 #5002 #5003 #5004 #5005	READ	X The finishing end position of X axis block（ABSIO） Y The finishing position position of Y axis block（ABSIO） Z The finishing position position of Z axis block（ABSIO） The finishing position position of the 4th axis block（ABSIO） The finishing position position of the 5th axis block（ABSIO）	The work piece coordinate system	Exclude	Possible
#5021 #5022 #5023 #5024 #5025	READ	X The current position of X axis（ABSMT） Y The current position of Y axis（ABSMT） Z The current position of Z axis（ABSMT） The current position of the 4th axis（ABSMT） The current position of the 5th axis（ABSMT）	The machine coordinate system	Include	Impossible
#5041 #5042 #5043 #5044 #5045	READ	X The current position of X axis（ABSMT） Y The current position of Y axis（ABSMT） Z The current position of Z axis（ABSMT）	The work piece coordinate	Include	Impossible

		The current position of the 4th axis（ABSMT） The current position of the 5th axis（ABSMT）			
#5061 #5062 #5063 #5064 #5065	READ	X Skipping the signal position in X axis（ABSKP） Y Skipping the signal position in Y axis（ABSKP） Z Skipping the signal position in Z axis（ABSKP） Skipping the signal position in the 4th axis（ABSKP） Skipping the signal position in the 5th axis（ABSKP）	The work piece coordinate system	Include	Possible
#5083	READ	The tool length offset value			Impossible
#5101 #5102 #5103 #5104 #5105	READ	X The servo position offset in X axis Y The servo position offset in Y axis Z The servo position offset in Z axis The servo position offset in the 4th axis The servo position offset in the 5th axis			Impossible

Remark Notice;

·The 1st bit represents the axial number （from 1 to 5）.

·The tool length compensation value saved by the variables #5081~#5083 is the current executed value rather than the processing value in the following blocks.

·In G31（jumping function）block, when the jumping signal is connected, the tool position is saved in the variables from #5061 to #5063. When the jumping signals in G31 block isn't connected, the tool position saved in these variables specifies the finishing position value of the block.

·During the traverse, the expected value can't be read due to the buffer function（preread）.

（11） The work piece coordinate system compensation value （the work piece zero position offset value）＃2500～#2906

The work piece zero position offset value can be read and rewritten.

Axes	FUNCTION	PROPERTY	VARIABLE NUMBER
The first axis	The external work piece zero position offset	Read	#2500
	G54 The work piece zero position offset		#2501
	G55 The work piece zero position offset		#2502
	G56 The work piece zero position offset		#2503
	G57 The work piece zero position offset		#2504
	G58 The work piece zero position offset		#2505
	G59 The work piece zero position offset		#2506
The 2nd axis	The external work piece zero position offset	Read	#2600
	G54 The work piece zero position offset		#2601
	G55 The work piece zero position offset		#2602
	G56 The work piece zero position offset		#2603
	G57 The work piece zero position offset		#2604
	G58 The work piece zero position offset		#2605
	G59 The work piece zero position offset		#2606
The 3rd axis	The external work piece zero position offset	Read	#2700
	G54 The work piece zero position offset		#2701
	G55 The work piece zero position offset		#2702
	G56 The work piece zero position offset		#2703
	G57 The work piece zero position offset		#2704
	G58 The work piece zero position offset		#2705
	G59 The work piece zero position offset		#2706
The 4th axis	The external work piece zero position offset	Read	#2800
	G54 The work piece zero position offset		#2801
	G55 The work piece zero position offset		#2802
	G56 The work piece zero position offset		#2803
	G57 The work piece zero position offset		#2804
	G58 The work piece zero position offset		#2805
	G59 The work piece zero position offset		#2806
The 5th axis	The external work piece zero position offset	Read	#2900
	G54 The work piece zero position offset		#2901
	G55 The work piece zero position offset		#2902
	G56 The work piece zero position offset		#2903
	G57 The work piece zero position offset		#2904
	G58 The work piece zero position offset		#2905
	G59 The work piece zero position offset		#2906

3.7.4 The Operational Commands

Various operation can be operated among the variables, the operational command is like the program in the common arithmetic expression.

#i=＜Expression ＞

The right ＜expression formula＞ of the operational command is the combination of the constant, the variable, the function and the operator. The constant can replace #j and #k. The constant free of the decimal position in ＜expression formula＞ can be processed as it's with the decimal position at its end.

3.7.4.1 Defining and Replacing the Variable

#i=#j Define and replace

3.7.4.2 Addition

#i=#j+#k Summation
#i=#j-#k Subtraction
#i=#joR#k Logic sum （for each digit of 32 digits）
#i=#jXOR#K Anticoincidence X OR/Exclusive OR（for each digit of 32 digits）

3.7.4.3 Multiplication

#i=#j*#k Multiplication
#i=#j/#k Division
#i=#jMOD#k Remainder
#i=#jAND#k Logic multiply（for each digit of 32 digits）

3.7.4.4 Function

#i=SIN[#j] Sine（Unit：degree）
#i=COS[#j] Cosine （Unit：degree）
#i=TAN[#j] Tangent （Unit：degree）
#i=ATAN[#j] Arc tangent （Unit：degree）
#i=SQRT[#j] Square root
#i=ABS[#j] Absolute value
#i=BIN[#j] Switch from BCD to BIN
#i=BCD[#j] Switch from BIN to BCD
#i=ROUND[#j] Round off
#i=FIX[#j] Round up the part after the decimal position
#i=FUR[#j] The decimal position part is forward into the integer part.

Remark Notice： How to use the function ROUND.

（1）If the function ROUND is used in the operation command or in the conditional formula of IF or WHILE, the original data with the decimal position should be rounded up.

For example: #1=ROUND[1.2345];

#1= 1.0

IF[#1 LEROUND（#2）]GOTO 10;

If #2=3.567, ROUND[#2]=4.0

（2）There is the function ROUND in the address command, round up based on the minimum setting unit of its address

For example: G01 X[ROUND（#1）];

If #1 is 1.4567 and X minimum input increment is 0.001, the block changes into G01 X1.457;

The function ROUND in one address command is mainly used in the following situations.

For example：[It just moves in the increment of #1 and #2, and then return to the starting position].

N1 #1=1.2345;

N2 #2=2.3456;

N3 G01 X#1 F100; X moves 1.235

N4 X#2; X moves 2.346

N5 X-[#1+#2]; X moves（Because #1+#2=3.5801）

Because 1.235+2.346=3.581so the program can't return to the starting position througn N5.

Use N5X-[ROUND[#1]+ROUND[#2]];

It's equal to N5x-1.235+2.346], and the program can return to the starting position.

3.7.4.5 Combined Calculation

The above operation and the function can be combined. The operational preferential order is the function, and then the multiplication, finally is the addition.

Example 1: #i #j+#K*SIN[#ℓ]

——①——The operational order

——②——

——③——

3.7.4.6 Changing the Operational Order Through [ ]

The preferential part can be bracted in [ ]. [ ] can be nested for 5 layers including the bracket in the function itself.

Example 1: #i=SIN [[[#j+#K] *#ℓ +#m] *n]（Nesting for three layers）

—①——

——②——

————③——

—————④——

——————⑤————

3.7.4.7 Precision

Pay attention to the precision during programming through the macro program function.

（1）Data formula

The formula of the data floating position processed by the macro program is shown as below：

M*2E

M:1 Code in one digit + the binary number in 31 digits

E: 1 Code in one digit + the binary number in 7 digits

（2）Operational precision

The operation is executed for one time, the following error exists, and these errors are accumulated after the repeated operation.

OPERATION FORMULA	MEAN ERROR	MAX ERROR	TYPE OF ERRORS
a=b*c	1.55×10^{-10}	4.66×10^{-10}	Relative error
a=b/c	4.66×10^{-10}	1.86×10^{-9}	$\left\lvert \frac{\varepsilon}{a} \right\rvert$
a=$\sqrt{b}$	1.24×10^{-9}	3.73×10^{-9}	
a=b+c a=b-c	2.33×10^{-10}	5.32×10^{-10}	min（$\frac{\varepsilon}{b}, \frac{\varepsilon}{c}$）
a=SiNb a=comb	5.0×10^{-9}	1.0×10^{-9}	Absolute error $\lvert\varepsilon\rvert$ degree
a=ATANb/c	1.8×10^{-6}	3.6×10^{-6}	

Remark; The function TAN executes SIN/COS.

3.7.4.8 Positions for Attention the Loss of the Precision

（1）Addition and subtraction

When the absolute value is subtracted, the relative error can't less than 10^{-8}. For example, if the actual values of #1 and #2 are shown as below：

#1=9876543210123.456

#2=9876543277777.777

Execute the operation of #2-#1：

#2-#1=67654.321

The above numerical values can't be obtained, the precision of #1and #2 gets lost because the precision in the macro program is eight digits in the decimal system.

#1=9876543200000.000

#2=9876543300000.000

Restrictly speaking, the above values and the internal values are different because the internal are the binary number.

#2-#1=100000.000

Therefore, it causes bigger error.

（2）Logic calculation

Basically, EQ, NE, GT, LT, GE are LE are same as the addition and subtraction. Therefore, pay attention to the error and confirm #1and #2 are equal in to the above example.

When IF[ABS[#1-#2]LT50000] determines, if the difference between #1and #2 is in the error range, #1and #2 are taken as equal.

（3）The triangle function

In the triangle function, the absolute error can be ensured. But, they are not less than 10^{-8} , pay attention to the multiplication and division after the triangle function calculation.

3.7.5 The Control Command

The executing sequence can be executed through the following commands.

3.7.5.1The Branch （GOTO）

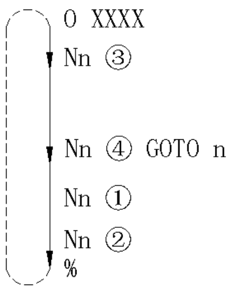
IF[＜Conditional formual＞]＝GOTOn

If it satisfies ＜Conditional formual＞, it transfers to the block of n. The serial number of n can be replaced by the variable or [＜formula＞] .

Remark：

1. The blocks after the serial number n are executed after GOTOn command, and the serial number n must be written at the beginning of the block.

2. When GOTOn is executed, the farther Nn block is in the executing program direction, the longer the executing time is.



In the above figure, the time is based on the sequence of ①②③④. Therefore, the high efficiency of executing GOTOn depends on the two conditions： 1. The times of executing GOTOn; 2. The distance with block Nn is very short in the executing sequence. The variable content is taken as the detection and the alarm, it's suggested that the alarm program shouldn't be closed to GOTOn sentence and the alaram program should be far away from GOTOn.

For example：When #1≥10, #150 alarms.

┆

IF[#1GE10]GOTO150;

When it doesn't alarm：

M99;

┆

N150 #3000=150;

M99;

3. During executing GOTO, it alarms in the following situation.

①In one address, the macro program operation can't be executed correctly.

When #1=-1, if GOTO is executed, in the block of X[SQRT[#1]], NO.119 alarms.

②The conditional formula specified by WHILE can’t be executed correctly.

When #1=0, if execute GOTO; in WHILE[10/#1 GE2]D0 1 block, NO.112 alarms.

In this case, the following programs should be rewritten：

①#2=SQRT[#1];
x#2;
②#2=10/#1
WHILE[#2 GE 2]D0 1;
┆
#2=10/#1;
END　1;

Even GOTO is executed, the operational commands don’t alarm.

3.7.5.2 Repeating

WHILE[＜the conditional formula＞］＝DOm（m=1,2,3）
ENDm

When <the conditional formaula> is satisfied, the blocks from Dom to ENDm are repeatedly executed; that is to say, <the conditional formula> which judges DOm block is satisfied, the block transfers to the next block. If the conditional formula isn’t satisfied, the blocks after ENDm are satisfied.

When the situation of WHILE [＜conditional formula＞] are same as that of IF, it can be omitted. If it is omitted, the blocks from Dom to ENDm are executed repeatedly.

WHILE [＜conditional formula＞］＝Dom and ENDm must be used in pair, and the recognition number m can recognize each other.

For example:
#120=1;
N1 WHILE [#120LE10] DO1;

N2 WHILE [#30EQ1] DO2; ← Repeat for 10 times
N3 END 2; ← Repeat when #30=1.

#120=#120+1;
N4　END1;

REPEAT :When it is programmed repeatedly, pay attention to the following:

① DOm must be specified before ENDm:
┆
END1;
┆（Can’t ）
DO1;
┆

② In one program, DOm and ENDm must be corresponded to each other.

```
┆
DO1;
┆
DO1;  ( Can't)
┆
END1;
┆
DO1;
┆
END1 (Can't)
┆
END1;
┆
```

③ The same recognition number can be used for many times.

```
┆
DO1;
┆
END1;
┆   (Can)
DO1;
┆
END1;
┆
```

④ DOD sentense can be nested for 3 times.

```
┆
DO1;
┆
DO2;
┆
DO3;
┆
END3;
┆
END2;
┆
END1;
┆
```

⑤ DO area can't be crossed.

```
┆
DO1;
┆
DO2;
┆
END1;
┆
END2;
```

┆

⑥ Transfer from the inside of DO area to the outside

```
  ┆
DO1;
 ┆
GOTO 9000;
 ┆      （Can）
END1;
 ┆
N9000….;
 ┆
```

⑦ It’s not allowed to transfer from the outside of DO area to the inside.

```
  ┆
GOTO 9000;
  ┆
DO1;   （Can’t）
  ┆
N9000…..;
  ┆
END1;
  ┆
DO1;
  ┆
N9000…..;
  ┆  （Can’t ）
END1;
  ┆
GOTO 9000;
```

⑧ The macro program and the subprogram can be called from DO area inside. In the macro program or the subprogram, DO sentence can be nested for three times.

```
  ┆
DO1;
  ┆
G65…..; （ Can）
  ┆
G66…..; （ Can）
  ┆
G 6 7 ;    （Can）
  ┆
E N D 1 ;
  ┆ ;
DO1 ;
  ┆
M 9 8….. ; （ Can）
  ┆
```

E N D 1;
┆
Remark Note: When the program is transferred and repeated, the executing time is very short.

Example 2: Wait for the cycle program and one of its signal（＃10000）should be 1.
┆
N 10 I F[#1000 EQ 0]GOTO 10;
┆
IF
┆
WHILE [#1000 EQ 0] DO 1
END1
The programming is through , the executing time is very short.

3.7.6 Macro Program Calling Commands

3.7.6.1 Simple Calling

(1) The macro program calling:
Non-mode One-shot calling （G65）
Mode calling （G66, G67）
The macro program is called through G codes.
The macro program is called through M codes.
The subprogram is called through M codes.
The subprogram is called through T codes.
(2) The difference between calling the macro program and the subprogram
The macro program calling （G65） is different with the subprogram calling（G98）, which is introduced as below：
·The independent argument variable (the data are transmitted into the macro program) can be specified through G65 while G98 is lack of the function.
·When M98 block includes the other NC command (such as, G01 X100.0 M98 Pp), the subprogram is called after the commands are executed. Contrarily, the macro program is called by G65 without any conditions.
·M98 block includes the other NC command (such as, G01 X100.0 M98 Pp), the machine stops in the single block mode. Contrarily, if it includes G65, the machine doesn't stop.
·用G65, If G65 commands, the level of the part local variable is changed.However,if M98 commands, the level of the part variable isn't changed.

3.7.6.2 Non-mode Calling（G65）

When G65 is specified, the user macro program specified by address P is called. The data (the independent argument variable) can be sent to the user macro program.
（1） Specifying the independent argument variable
The independent variable can be specified in two types. The independent variable I uses the letters except for G, L, O, N and P, and each letter specifies for one time. Ⅱ uses A, B, C and I_i, J and

K_i(i is 1～10). Based on the used letter, the independent variable type can be auto set.

A) I Independent variable I

The corresponding relation between the addresses in the independent variable assignment I and the variable number in the macro program is shown as below:

ADDRESSES OF THE INDEPENDENT VARIABLE	VARIABLES IN THE MACRO PROGRAM
A	#1
B	#2
C	#3
D	#7
E	#8
F	#9
H	#11
I	#4
J	#5
K	#6
M	#13
Q	#17
R	#18
S	#19
T	#20
U	#21
V	#22
W	#23
X	#24
Y	#25
Z	#26

For example:

B_A_D_…J_K_ Correct

B_A_D_…J_I_ Not correct

B) II Independent variable assignment II

A__ B__ C__ I__J__K__ I__J__K__ _ _ _ _

The independent value can assign the values to A, B and C; moreover, the independent variable of the maximum ten groups can be specified by addresses I, J and K. The several numbers can be assigned with the same address in order and the addresses which are not required can be omitted. The corresponding relation between the addresses distributed by the independent variable II and the variable number of the macro program is shown as below:

ADDRESSES OF THE INDEPENDENT VARIABLE ASSIGNMENT II	VARIABLES IN THE MACRO PROGRAM
A	#1
B	#2
C	#3
I1	#4
J1	#5
K1	#6
I2	#7
J2	#8
K2	#9
I3	#10
J3	#11
K3	#12
I4	#13
J4	#14
K4	#15
I5	#16
J5	#17
K5	#18
I6	#19
J6	#20
K6	#21
I7	＃22
J7	＃23
K7	＃24
I8	＃25
J8	＃26
K8	＃27
I9	＃28
J9	＃29
K9	＃30
I10	＃31
J10	＃32
K10	＃33

The suffix of I, J and K is the order of the assigned group.

C) Mixed using the independent variable I and I

It doesn't alarm even the independent variables of the assignment I and II are in the block with command G65. If the independent variables I and II correspond to the same variables, the later specified one is valid.

Example 1:

G 65　A1.0　B2.0　I－3.0　I4.0　D5.0　P1000;

< The variable>

＃1:1.0

＃2:2.0

＃3:

＃4:－3.0

＃5:

＃6:

＃7 → 5.0

n the example, although the independent variables I 4.0 and D 5.0 all are specified in ＃7 variable, the later specified D 5.0 is avid.

Example 2: Setting the datum reference/base　position

Before the processing holes is commanded, the datum position of the holes must be set.

X coordinate value of X_0 hole datum position

Y coordinate value of X_0 hole datum position

The calling commands of the macro program:

G 65　P9200　X_x　Y_y;

The following variables should be used:

Counting the holes of ＃100.

＃100 is for X coordinate value of the datum position of the macro program of the holes.

＃102 is for Y coordinate value of the datum position of the macro program of the holes.

#24 uses the macro program calling commands for assigning the values to X coordinate of the datum position.

#25 uses the macro program calling commands for assigning the values to Y coordinate of the datum position.

The macro program is edited as below:

09200;

Send the datum position into the macro program of the holes.

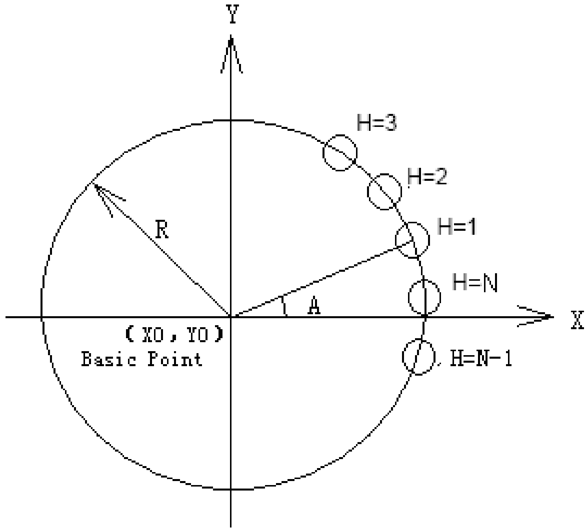
＃102＝＃25;

＃100＝0;　: The hole counter is reset.

M99;

Example 3: The tap ring：

The reference position set by the macro program which is set by the datum position is taken as the center of a circle, h holes to be processed is distributed on the ring on the equal interval. The 1st hole is on the straight line of the angle a, refer to the following figure：



XO, YO The coordinate values of XO and YO tap ring reference positions.

R radius

A starting angle

H Number of H holes

The macro program calling commands:

G65 P9207 Rr Aa Hh;

When h＜0, the work piece is processed based on –h counting in CW direction.

The following variables should be used:

#100 Counting the holes
#101 The datum position of X coordinate value
#102 The datum position of X coordinate value
#18 The radius value r
#1 The starting angle a
#11 The number of holes h
#30 Saving X coordinate value of the datum position
#31 Saving Y coordinate value of the datum position
#32 Counting means the 1st hole is being processed
#33 The angle of the 1st hole

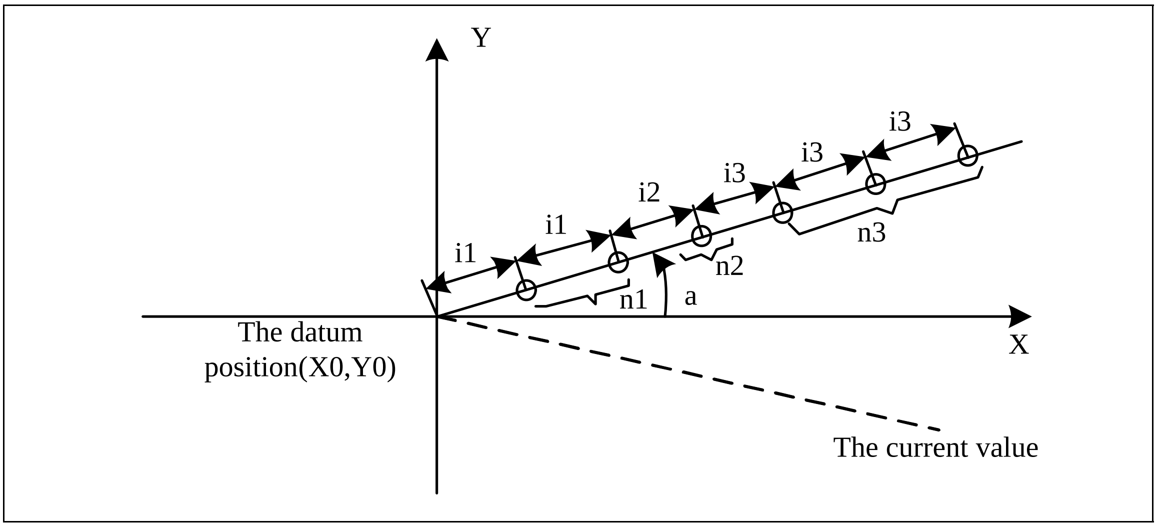
The macro program is edited as below during the absolute programming:

```
09207;
#30=#101;          : Saving the datum position
#31=#102;
#32=1;
WHILE[#32LEABS[#11]]DO1;       : Repeat based on the number of the holes
#33=#1+360*[#32-1]/#11;
#101=#30+#18*COS[#33];          : The hole position
#102=#31+#18*SIN[#33];
X#101    Y#102;
#100=#100+1;                    : 1 The number of the processing holes adds one
```

```
#32=#32+1;
END1;
#101=#30;                    : Return to the datum position
#102=#31;
M99;
```

Example 4 : Unequal interval oblique line

The position set by the macro program which is set by the datum position, is taken as the datum position, and it is arranged in the unequal intervals（1, 12……）in the direction of angle a which is formed by the hole edge and X axis,



The coordinate values of X0 and Y0 reference positions

A angle

I The interval of the holes

K The number of the holes is continuously set by the equal interval and it must be assigned value through the decimal position.

The macro program calling commands

G65 P9203 Aa, I1, Kn1, I12, Kn2…………;

When n=1, Kn can't be rewritten.

The following variable can be used:

#100 : The counter for counting holes

#101 : X coordinate value of the datum position

#102 : Y coordinate value of the datum position

#1 : Angle a

#4 : The 1^{st} interval 1_1

#6 : The 1^{st} one to space the hole numbers n_1 of group 1_1

#7 : The 2^{nd} interval 1_2

#9 : The 2^{nd} one to space the hole numbers n_2 of group 1_2

┆

#2 : Saving X coordinate value of the datum position

#3 : Saving X coordinate value of the datum position

#5 : Taking the counting number of holes interval I_1

#8 : The distance from the datum position to the current hole

The macro program is edited as below during the absolute programming:

```
09203;
#2=#101;    : Saving the datum position
#2=#102;
#5=4;
#8=0;
WHILE[#5 LE 31]D01;            : The hole interval assignment I is limited in 10
IF[#[#5]EQ#0]GOTO 9001;        : If the assignment is I, ＜＞ ends.
D02;
#8=#8+#[#5];
#101=#2+#8*COS[#1];            : The hole position
#102=#3+#8*SIN[#1];
X#101   Y#102;
#100=#100+1;                   : The number of holes adds 1.
#[#5+2]=#[#5+2]-1;
IF[#[#5+2]LEO]GOTO 9002;       : Repeat for K times
END2;
N9002 #5=#5+3;                 : Move to the next assignment I
END1;
N9001 #101=#2;                 : Return to the datum reference position
#102=#3;
M99;
```

3.7.6.3 Modal Call (G66)

Modal call can be specified when the following command is executed. Call specified macro program each time to execute a movement command when macro call mode is performed.

G66 P（program no.）L（repetition count）＜argument designation＞;

＜argument designation＞is identical with its function in simple call.

G67 Macro call cancellation

Note:

·Program number of modal call is specified by address P after G66.

·When a number of repetition is required, a number from 1to 9999 can be specified at address L.

·Identical with non-modal call (G65), data specified by argument is passed to macro program.

·Calls can be nested to a depth of 4 levels including non-modal call（G65）and modal call（G66）. This does not include subprogram call（M98）.

·Modal calls can be nested by specifying another G66 code during a modal call.

·In G66 block, macro program can not be called.

·G66 needs to be specified before arguments.

·No macro program can be called in a block which contains a function such as miscellaneous function that does not involve movement command.

·Local variables (arguments) can only be set in G66 blocks. Note that local variables are not set each time a modal call is performed.

Sample 1 Drilling cycle

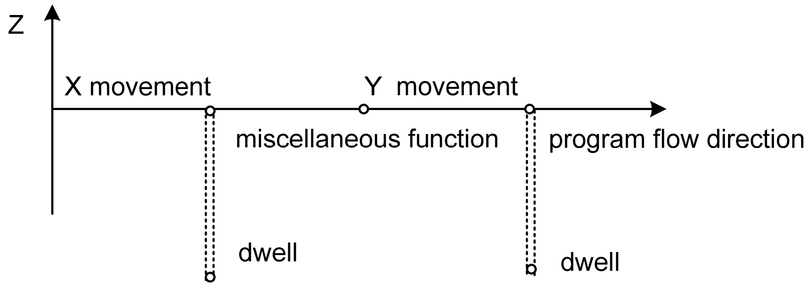


Fig.3-31

Drilling cycle is performed at each poisoning point.

G66 P9082 R（point R）X（point Z）X（dwell time）;

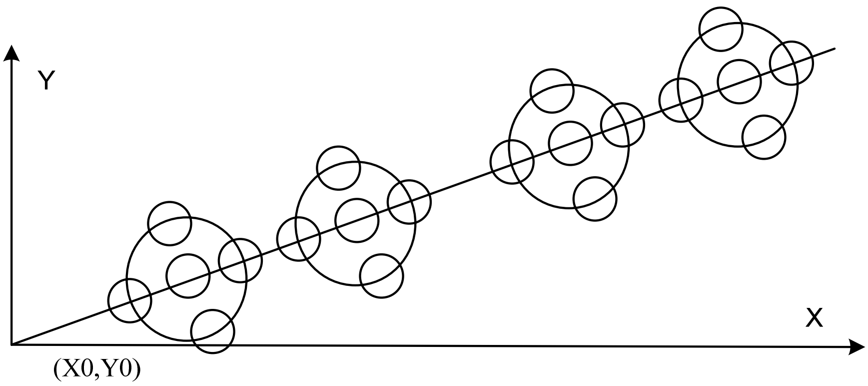
X ;
M ;
Y
┆
G67 ;

Drilling cycle is performed by a certain move program in this area.

Macro format is as follows (in incremental programming):

G9082;
G00 Z#18;
G01 Z#26;
G04 X#24;
G00 Z-[ROUND[#18]+ROUND[#26]];
M99;

Sample 2 combined holes



For [3.7.7.2 non-modal call G65], the drilling programs of holes which are composed of the hole rings described in Example 2 and the holes described in Example 3 and which are arranged in an unequal interval on the oblique line must be executed by macro program and fixed (canned)cycle as follows:

G81…………;
G65 P9200 X（coordinates of reference point）Y（coordinates of reference point）;
G66 P9207 R（radius）A（starting angle）K（hole number）;

G65 P9203 A（degree of angle）I（interval）K（number）I（interval）;
G67;

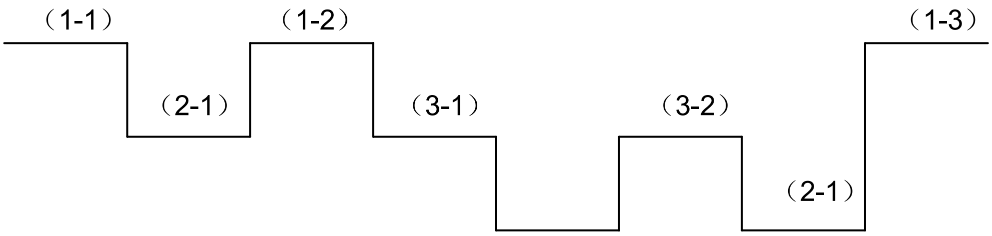
3.7.6.4 Multiple Call

Like calling another one from a subprogram, another macro program can be called from a macro program. Multiple calling includes single and modal call, which repetition times are less than or equal to 4 times.

3.7.6.5 Multiple Modal Call

In modal call, movement command can be performed once when the specified macro program is called once. If several modal macro programs are specified, movement command in last macro program can be executed once when the following macro program is called once. Macro program is called continuously by the following specified command.

Sample 1
G66 P9100;
Z10000; （1-1）
G66 P9200;
Z15000; （1-2）
G67; :P9200 cancelled
G67; :P9100 cancelled
Z-25000; （1-3）
09100;
X5000; （2-1）
M99;
09200;
Z6000; （3-1）
Z7000; （3-2）
M99;
Perform sequence (blocks without movement command are omitted)



(Note) :Because it is not macro call after (1-3), a modal macro program is not called.

3.7.6.6 Macro call using G code

G code used to call macro program is set by parameter. It can be replaced by N__G65 P△△△△<argument assignment>. Following command has the same function:

N__G××< argument assignment >

Correspondence between macro call×× and macro call number △△△△is set by parameter.

Called G code ×× and called program number △△△△ are set in parameter.

In addition to G00, at most 10 can be selected between G01 and G255 to call macro program. These G codes can not be specified in MDI mode in the same way as with G65. These G code can not be specified in macro call program using G code, which can not be used in subprogram call command with M code.

Set Following Parameters:

06050	G code used in macro call:9010
06051	G code used in macro call:9011
⋮	
06059	G code used in macro call:9019

Sample 1 CW arc machining using G02

G02 I（Radius）D（offset number）;

（1）Set following Parameters

Macro program:9010call G code =12

（2）Record the following macro program.

```
09010;
#1-ABS[#4]-#[2000+#7];
IF[#1 LEO]GOTO 1;
#2=#1/2;
#3003=3;
G01 X[#1-ROUND[#2]]Y#2;
G17 G02 X#2 Y-#2R-#2;
I-#1;
X-#2 Y-#2 R#2;
G01 X[#-ROUND[#2]]Y#2];
#3003=0;
N1 M99;
```

3.7.6.7 Call Subprogram by M Code

By setting an M code used to call a subprogram in parameter. The command of N__G__X Y__……M98P△△△△ can be replaced by following simple command.

N__G__X__Y__……M××;

In M98 execution, MF and M code are not sent to PLC.

Correspondence between M×× code call subprogram and program numbers △△△△ is set by parameter.

Set the Following Parameters

06071	M code for subprogram call:9001
06072	M code for subprogram call:9002
⋮	
06073	M code for subprogram call:9003
06086	M code for subprogram call:9026

Sample 1 ATC canned cycle by M06

（1） Set the following parameters

Subprogram:M code called by 9001=06.

（2）Record the following macro program

```
09001;
#1=#4001;
#3=#4003;
G28  G91  Z0  M20;
G28  Y0;
M21;
G00  Z10000;
M22;
G28  Z0;
M23;
G#1  G#3  M99;
```

3.7.6.8 Macro Call Using an M Code

By setting an M code used to call macro program in parameter. Namely, N__ G65 P△△△△<defined variable>

Following command has the same function.

N__ M××< defined variable >

The number of called macro program is set by corresponding parameters.

In addition to part specified M codes, 10 called macro program can be used in M06～M255. However, the same as G65, these M codes can not be input with MDI mode. Namely, these M commands can not be used in subprogram call with G code, M code and T code.

Parameters are set as follows:

06087	user macro program:9027 calls M code
⁓	⁓
06089	user macro program:9027 calls M code

3.7.6.9 Subprogram Call Using a T Code

Subprogram can be called with a T code in a parameter.

N__G__X__Y__……………………Tt;

The following 2 programs have the same functions as T code.

#149 = t;

N__G__X__Y__………………………M98 P90000;

TF and T code are not sent to PLC when macro program is called by T code.

In a macro called with a G code or in subprogram called with an M or T code, no subprogram can be called using a T code. T code in such a macro or subprogram is treated as an ordinary T code.

3.7.6.10 Differences between M98 (Subprogram call) and G65 (macro program call)

（1）G65 may contains arguments, which is not contained in M98.

（2）In M98, after a command different from M, P and L is performed, it is transferred to subprogram by M98. M65 is only for transmission.

（3）When M98 contains other addresses except O, N, P and L, single block perform is stopped, while it is continuous in G65.

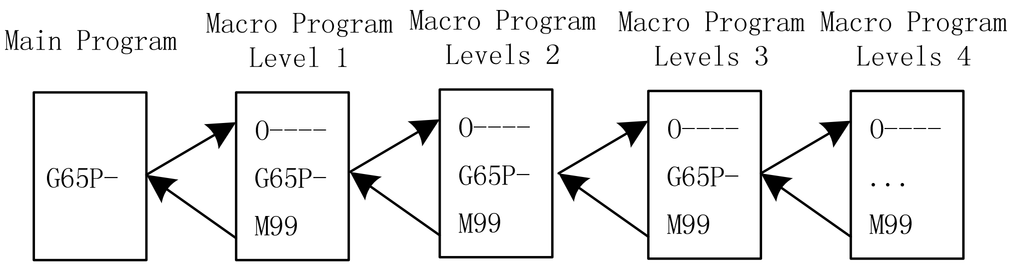
（4）Level of local argument can be changed in G65. M98 does not have this capability. Namely, #1 specified before G65 has one meaning, and it has the other meaning in macro program. #1 specified before M98 is identical with #1 in the subprogram call.

（5）When G65 and G66 used together, nest time is up to 4, which is the same as M98.

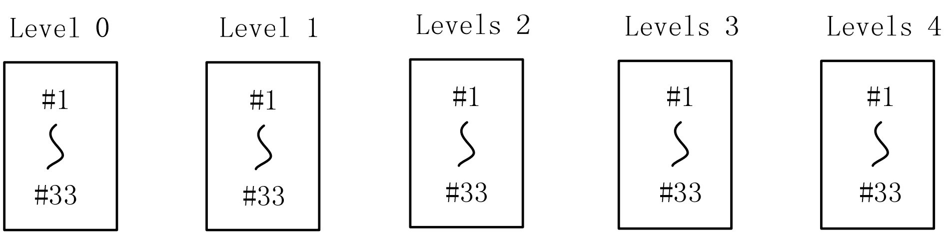
3.7.6.11 Correspondence between the Nest of Customer Macro Program and Local Variables

Its nest level increases 1 when macro program is called by G65, G66 or G code. At the same time, the level of local variable increases 1.

Correspondence between macro program call and local variables is as follows:



Local variables



① Note: #1～#33 local variables are provided in main programs.

② When macro program (level1) is called by G65, local variables of main program (level 0) is saved, and local variables #1～#33 (level1) of new macro program is prepared, replacement of argument is possible(the same as ③)

③ Once macro program (levels 2, 3, 4) is called, each local variable series (levels 1, 2, 3) is saved, and new local variables (levels 2, 3, 4) are prepared.

④ When M99 return from each macro program, local variables (levels 0, 1,2, 3,)saved in ②,③ are resumed as it is saved.

3.7.7 Limitations

（1）Available variables

#0, #1～#33, #100～#199, #500～#999, system variables.

（2）Available variable values

Max.: $\pm10^{47}$, Min.: $\pm10^{-39}$

（3）< expression formula>using rating data

Max.:±99999999, Min.: ±0.0000001

decimal:available

（4）Calculation precision decimal system 8 digits

（5）Macro program call nest degree Max. 4 levels

（6）Repetition identification sign 1～3

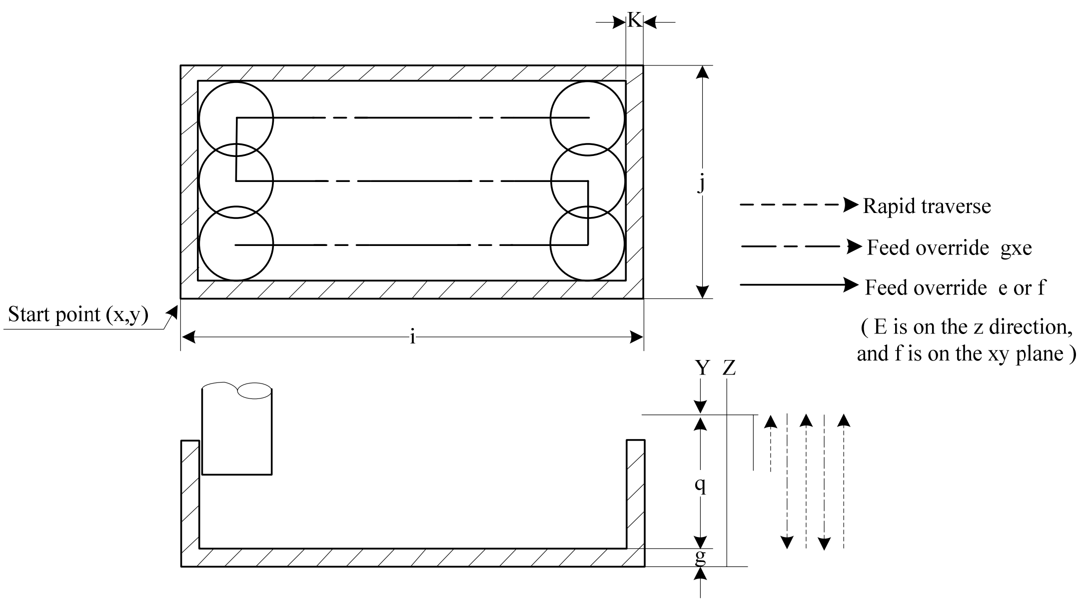
（7）[ ] nest Max. 5 levels

（8）subprogram call nest degree Max. 4 levels

3.7.8 Sample of Customer Macro Call

3.7.8.1 Grooving

Groove canned cycle is performed by customer macro call in the following drawing. Z is a machining dimension with certain depth, which is cut-in depth of machining dimension.



（1）Command called by customer macro program

G65 P9802 X x Y y Z z R r Q q I I J j K k T t D d F f E e *

meaning of each address

xy:Start point (bottom -left of groove) XY axes absolute coordinates.

zr:Absolute coordinates of point Z and R（Reference drawing）

g: Cut-in depth of one time (positive number)

ij: X direction of machining area, length of Y direction（positive number）（Reference drawing）（when i>j, work efficiency is higher）

k:End allowable value

t:Machining width is not exceed to xt％ of tool diameter.

d:Tool radius compensation number （01～99）

f:Feed rate on xy plane.

e:Feed rate in cut-in, feed with 8×e feed rate at 1mm before cut-in.

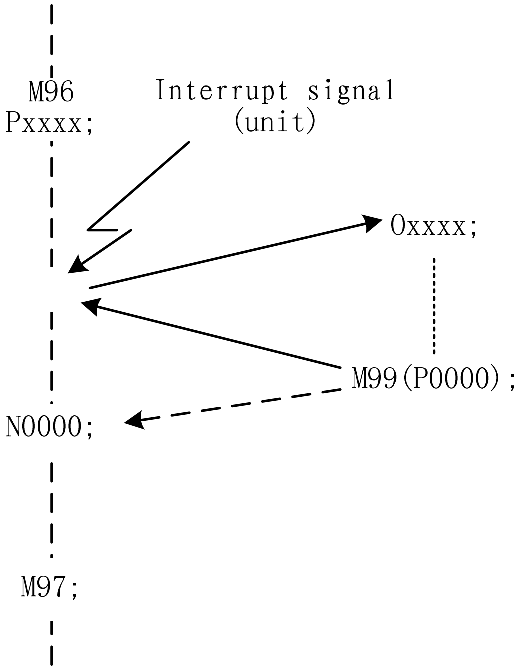
（2）Customer macro program

```
0 9802;
＃27＝＃[2000＋＃7];
#28=#6+#27;
#29=#5-2*#28;
#30=2*#27*#23/100;
#31=FUP[#29/#30];
#32=#29/#31;
#10=#24+#28;
#11=#25+#28;
#12=#24+#4-#28;
#13=#26+#26+#6;
G00 X#10 Y#11;
Z#18;
#14=18;
D01;
#14=#14－＃17;
IF[#14GE13]GOTO 1;
#14=#13;
N1 G01 Z#14 F#8;
X#12 F#9;
#15=1;
WHILE[#15 LE #31] D02;
Y[#11+#15*#32];
IF[#15 AND 1 EQO]GOTO02;
X#10;
GOTO 3;
N2 X#12;
N3 #15=#15+1;
END2;
G00 Z #18;
X#10 Y#11;
IF[#14 LE#13]GOTO 4;
```

G01 Z[#14+1F[8*#8];
END1;
N4 M99;

3.7.9 Interruption Function of Macro Program

When M96 PX X X X is performed in a program, macro program interruption is valid, and execution on blocks, input an interrupt signal （UINT） in NC to execute the program specified by PX X X X.



Set M99, program returns back from macro program to original program. Sequence number of returning is set by address P.

Note: Please refer to the operator's manual from machine manufacturer when this function is used.

3.8 Feed G Code

3.8.1 Feed Mode G64/G61/G63

Feed From:

Exact stop modeG61
Tapping mode G63
Cutting mode G64

Functions:

Once G61 specified, this function is valid until G62, G63 or G64 is specified. Tool is decelerated at the end point of a block, and then an in-position check is made. Then the next block is executed.
Once G63 specified, this function is valid until G61, G632 or G64 is specified. Tool is not decelerated at the end point of a block, but the next block is executed. When G63 is specified, feedrate override and feed hold are invalid.
Once G64 specified, this function is valid until G61, G632 or G63 is specified. Tool is not decelerated at the end point of a block, but the next block is executed.

Explanations:
No parameter format
G64 is default mode of the system, program is not decelerated at the end point of a block, but the next block is executed directly.
The purpose of in-position check in exact stop mode is to check that the servo motor has reached within a specified range.
In exact stop mode, move path of cutting mode and tapping mode are different.
For details, please refer to following figure 3-38

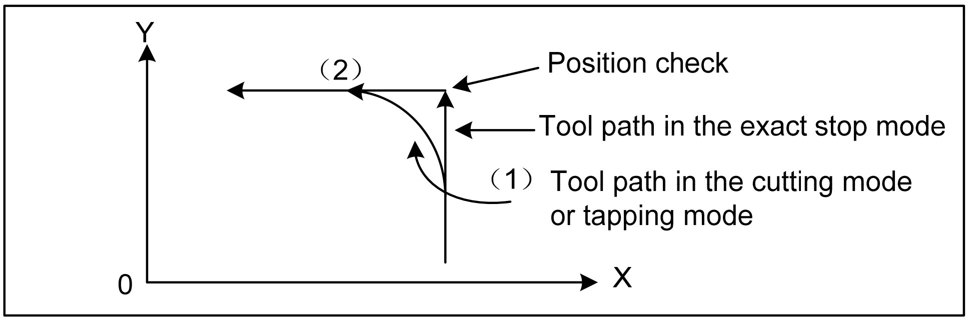


Figure 3-38 Tool paths from block 1 to block 2

3.8.2 Automatic Corner override （G62）

Command format: G62

Function: Once G62 specified, this function is valid until G61, G63 or G64 is specified. When tool radius compensation is performed, the movement of the tool is decelerated at an inner corner. This reduce cutting amount at unit time, and produces a smoothly machined surface.

Explanations:

1. When tool radius compensation is performed, the movement of the tool is automatically decelerated at an inner corner and internal circular area. This reduces the load on cutter and produces a smoothly machined surface.

2. When G62 is specified, and the tool radius compensation applied forms an inner corner, the feedrate is automatically adjusted at both ends of the corner. There are 4 types of inner corners (Fig.3-39). In figure: $2° \leq \theta \leq \theta p \leq 178°$. Θp is set by parameter P144.

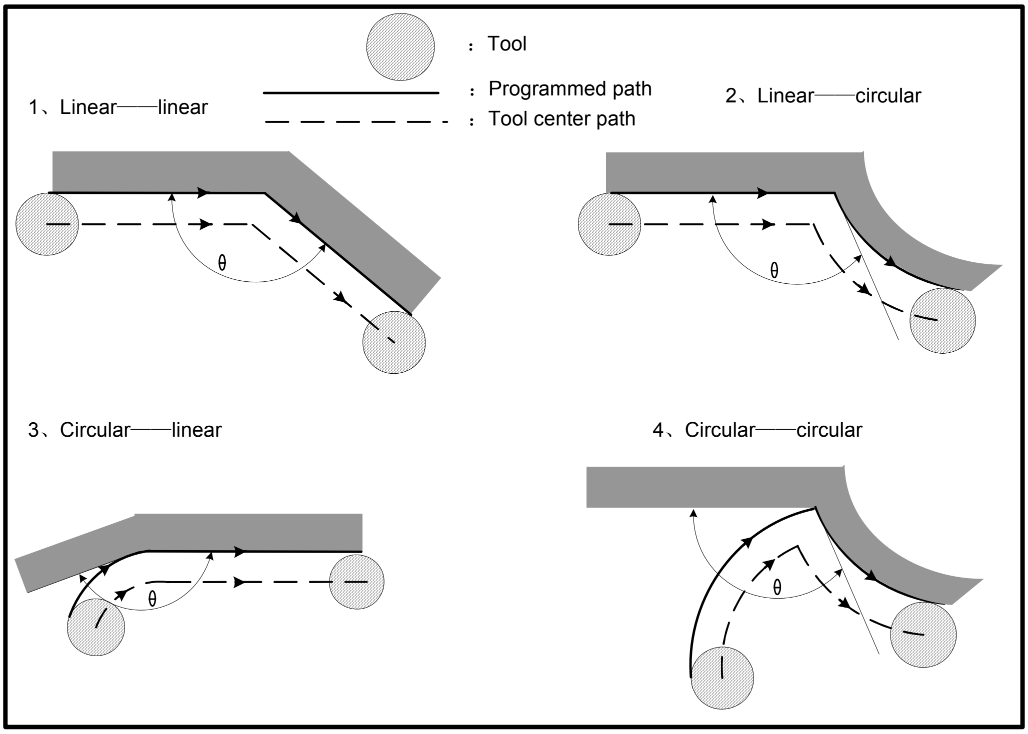


Fig.3-39 4 types of inner corners

3. When a corner is determined to be an inner corner, the feedrate is overridden before and after the inner corner. The distances Ls and Le, where the feedrate is overridden, are distance from points on cutter center path to the corner. Fig. 3-40 Ls+Le≤2mm

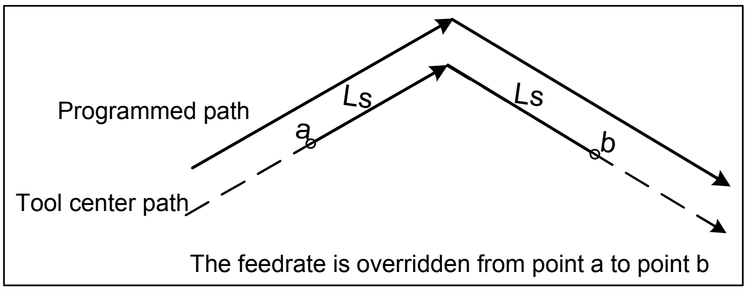


Fig.3-40 straight to straight line

4. 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. 3-41)

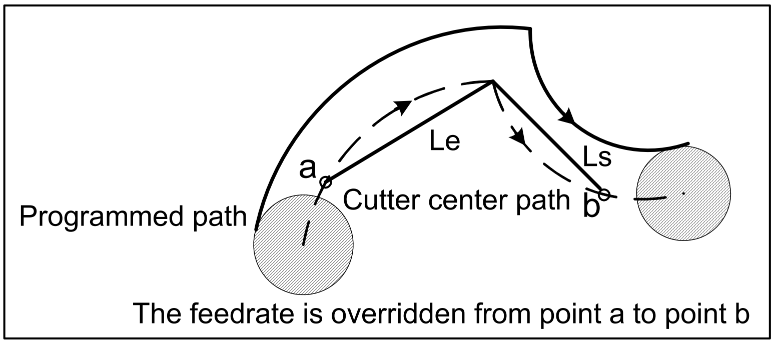


Fig.3-41 arc-arc

5. Regarding program types are straight line-straight line and arc-arc, the feedrate is overridden from point a to point b and from point c to point d. (fig.3-42)

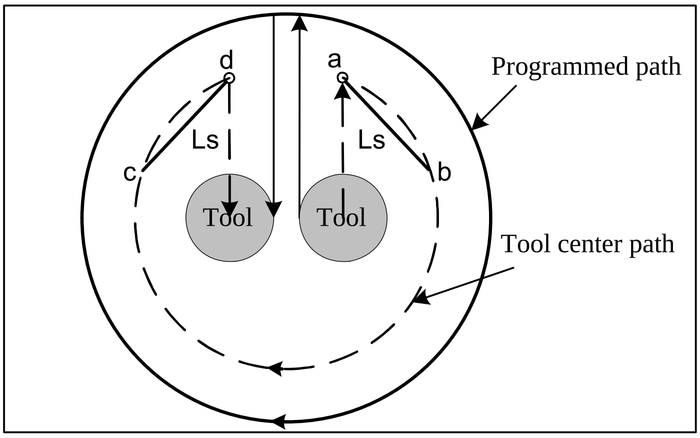


Fig.3-42 straight line-arc, arc- straight line

Limitations

1. Override for inner corners is invalid during acceleration/deceleration before interpolation.
2. Override for inner corners is invalid 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.

3.9 Introduction of Five Axes Control

3.9.1 Tool Center Point (TCP) Control

TCP control format:
G43.4 IP_α_β_ H **;**
IP_α_β_ ;
TCP control cancel format:
G49 IP_α_β_;

IP :In absolute command mode, the end point coordinates
In incremental command mode, the move distance of the TCP
α,β :In absolute command mode, the end point coordinates of rotary axes

In incremental command, the amount of movement of the rotary axes.

H :Tool offset number

When the CNC executes rotation interpolation, it controls the control point so that the TCP moves linearly toward the worktable (workpiece). The end point of TCP path is the coordinate in the program coordinate system.

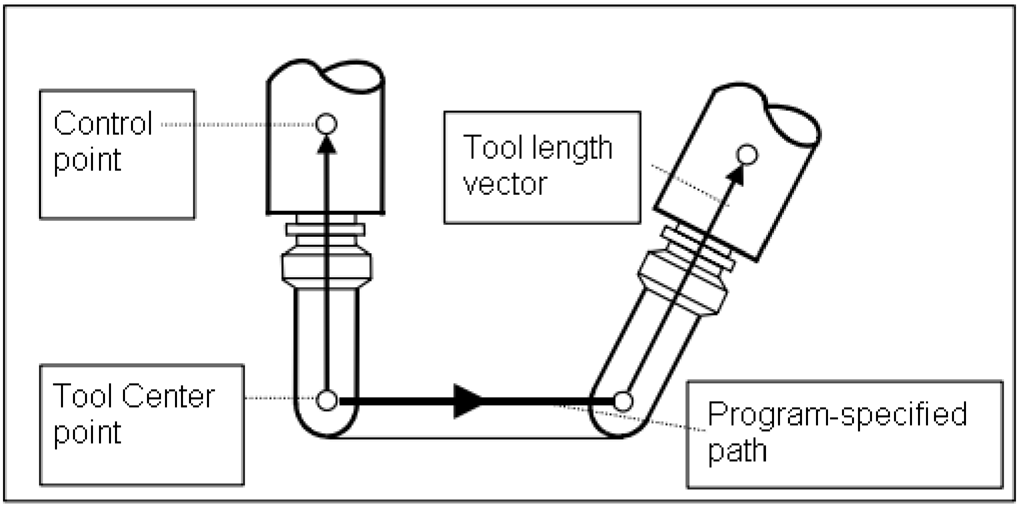
Function:

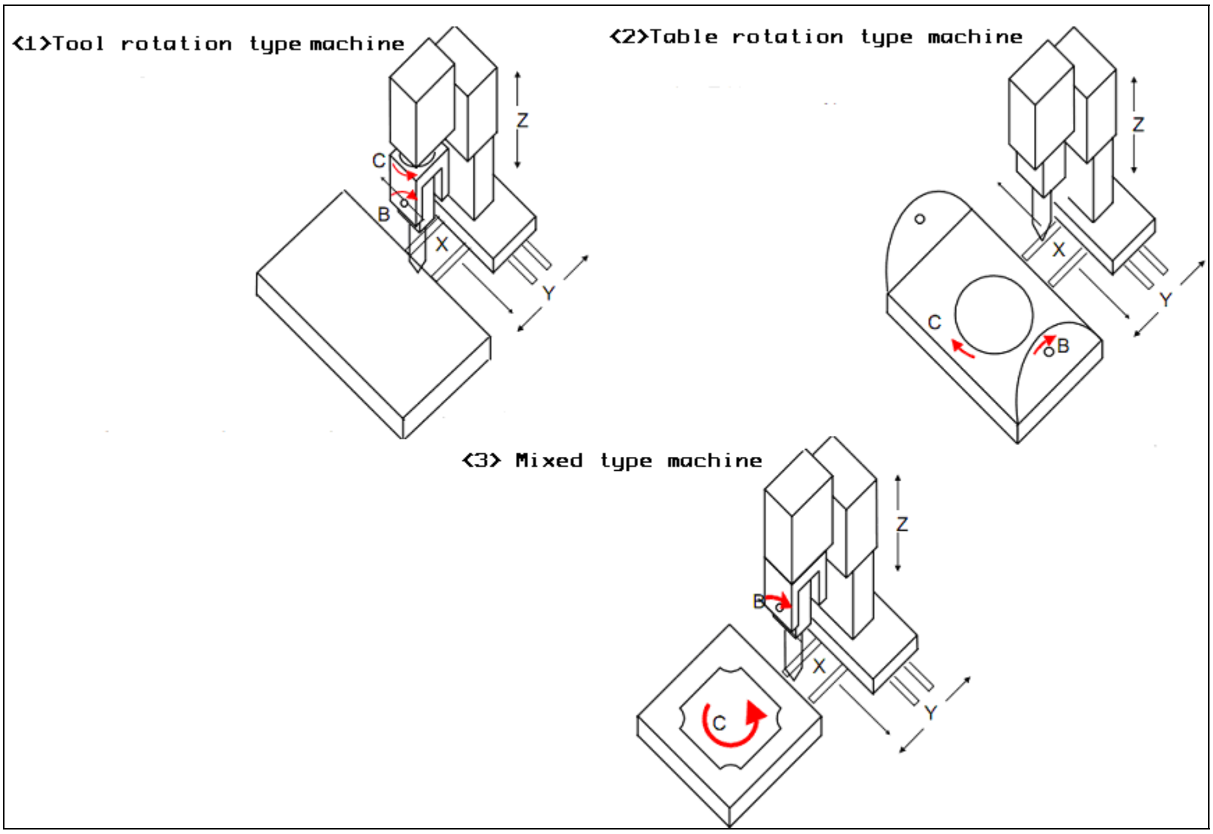
This function is intended to perform machining on such 5-axis machines which have rotary axes that turn a tool or table as well as three orthogonal axes (X, Y, and Z axes) by accomplishing tool length compensation while changing the attitude of the tool. Even when the direction (cutter to workpiece) is changed, the TCP still moves along the specified path.

A coordinate system used for programming the TCP control is called the programming coordinate system. The coordinate system that fixed on the worktable is used as programming coordinate system, which makes CAM programming easy.

There are three types of 5-axis machine tool: ①the one that rotates the tool only; ②the one that rotates the table only; ③the one that rotates both the tool and table.

This function is applied in the 5-axis machine tool including X,Y, Z three ortho-axes and cutter rotary axis and worktable rotary axis.





When the coordinate system fixed on the worktable is taken as the programming coordinate system, a program can be run without considering the rotation worktable, because as the worktable rotates, the position and direction of workpieces are changed at the same time, i.e. when a straight line is specified, the TCP moves along a straight path with respect to the workpiece as instructed.

Example:

Machine configuration: The A-axis is the rotation for controlling the tool.

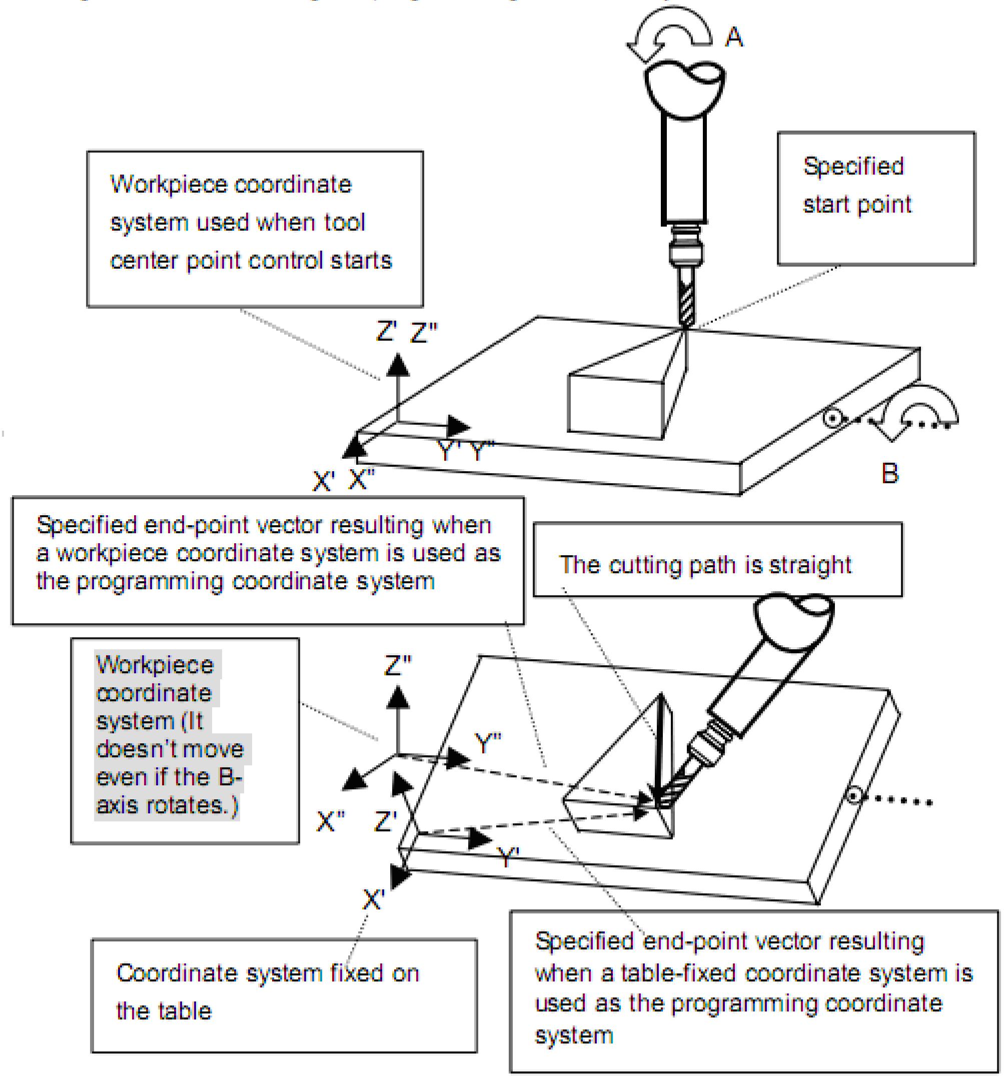
The B-axis is the rotation axis for controlling the table.

Program: Creating using the programming coordinate system.

Example)

Machine configuration: The A-axis is the rotation axis for controlling the tool.
The B-axis is the rotation axis for controlling the table.

Program: Created using the programming coordinate system.



Explanation:

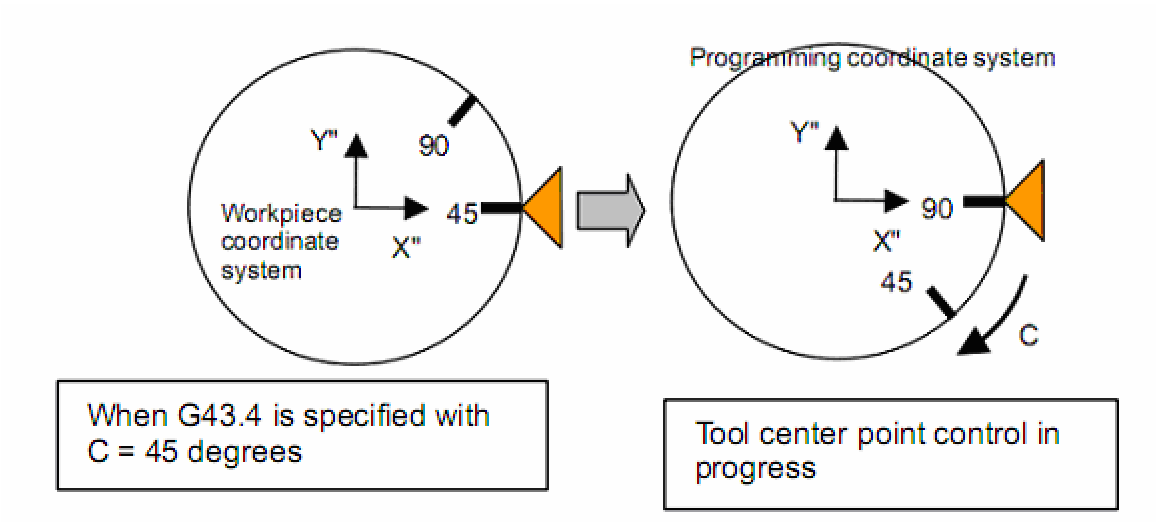
Program coordinate system

Program coordinate system is the coordinate system performs TCP control. Command G43.4 specifies the coordinate system fixed on the worktable as the programming coordinate system. When the coordinate value of rotary axis is zero, the program coordinate system and workpiece coordinate system are coincident. Thereafter, the program coordinate system rotates with the worktable rather than the tool.

The X, Y, Z commands behind G43.4 are regarded as coordinates within the programming coordinate system.

When G43.4 is commanded, the standard status of programming coordinate system of the worktable rotary axis is set by offset commands (G54-G59).

In the following instruction, X' Y' Z' represent the coordinate system fixed on worktable.



Display the current position during TCP control

The current position of control point in the workpiece coordinate system is displayed in the machine coordinate system during TCP control.

TCP control command

When TCP control is in use, it specifies the position of TCP at the end of blocks which can be seen in the program coordinate system.

For rotary axis, it specifies coordinate value of blocks ends.

Besides, for feedrate, the F command specify the tangential speed with respect to workpiece (the relative speed between workpiece and tool).

.

Commands available during TCP control

Commands available during TCP control are linear interpolation (G01), positioning (G00).

When specifying linear interpolation (G01), a specified speed is performed on TCP through speed control function.

Reset operation during TCP control

Resetting in G43.4 modal status will disable the status. (the same effect as G49 is executed).

Mode switching

After the TCP control is enabled, switching the modes will disable the modal status (the same effect as G49 is executed). If TCP control is activated in AUTO and MDI mode, the switching operation after entering into G43.4 status is to cancel the TCP control. If it is needed to switch back to AUTO and MDI mode, the operation can only be done after re-entering into G43.4 status, otherwise, machining error and danger will occur.

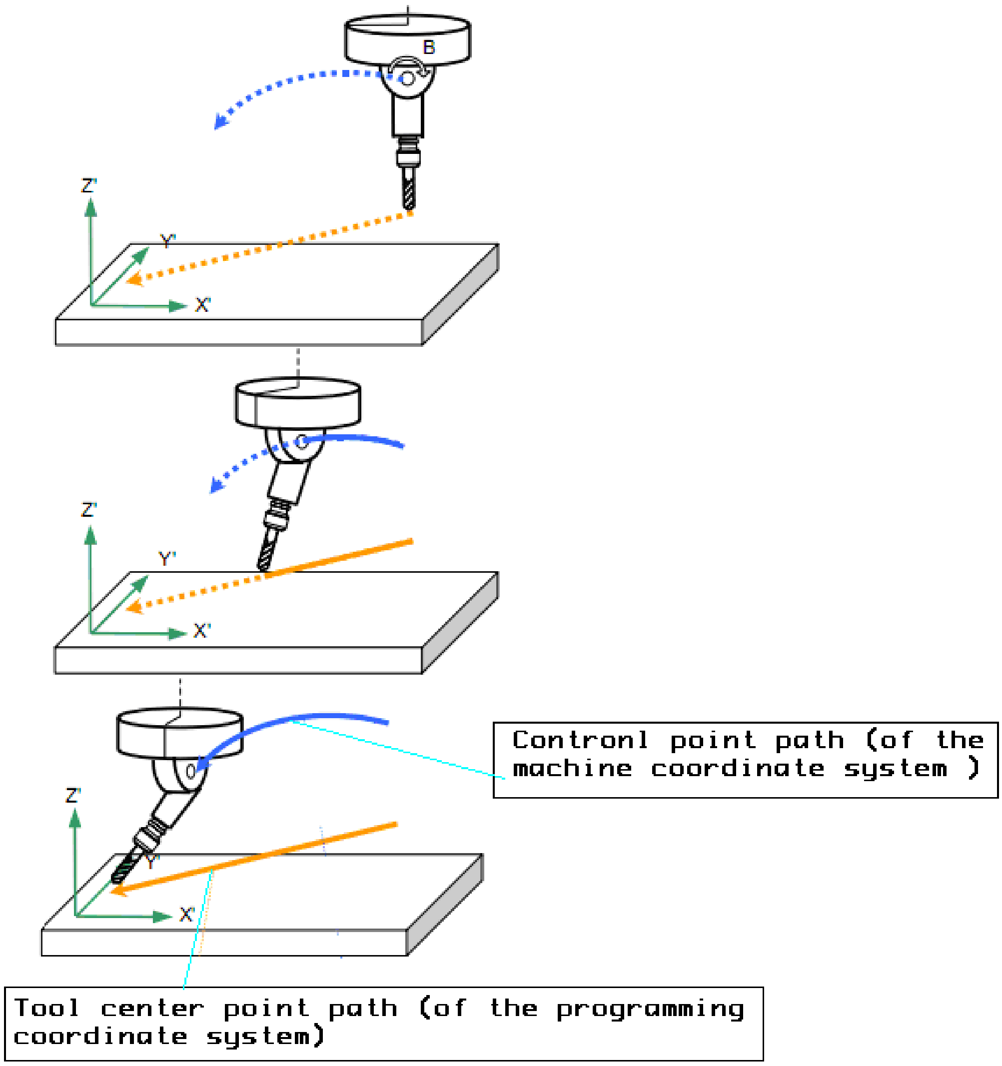
Example:

Tool rotation type machine

When the workpiece coordinate system is taken as programming coordinate system and linear interpolation is specified on X, Y, Z axes, the CNC can control the TCP to move linearly towards the worktable (workpiece) while tool rotation being performed. Through speed control, the TCP moves towards the worktable (workpiece) at a specified speed.

For this type of machine, when the tool rotation axis rotates, the worktable does not rotate with respect to the workpiece coordinate system, so the programming coordinate system always coincides with the workpiece coordinate system.

N1 G00 G90 B0 C0 ;
N2 G54 ; prepare program coordinate system
N3 G43.4 H01 ; TCP control starts, H01 is the tool compensation number
N4 G00 X200.0 Y150.0 Z20.0 ; move towards end point
N5 G01 X5.0 Y5.0 Z5.0 C60.0 B45.0 F500 ; linear interpolation
N6 G49; cancel TCP control
N7 M30;



Worktable rotation type machine

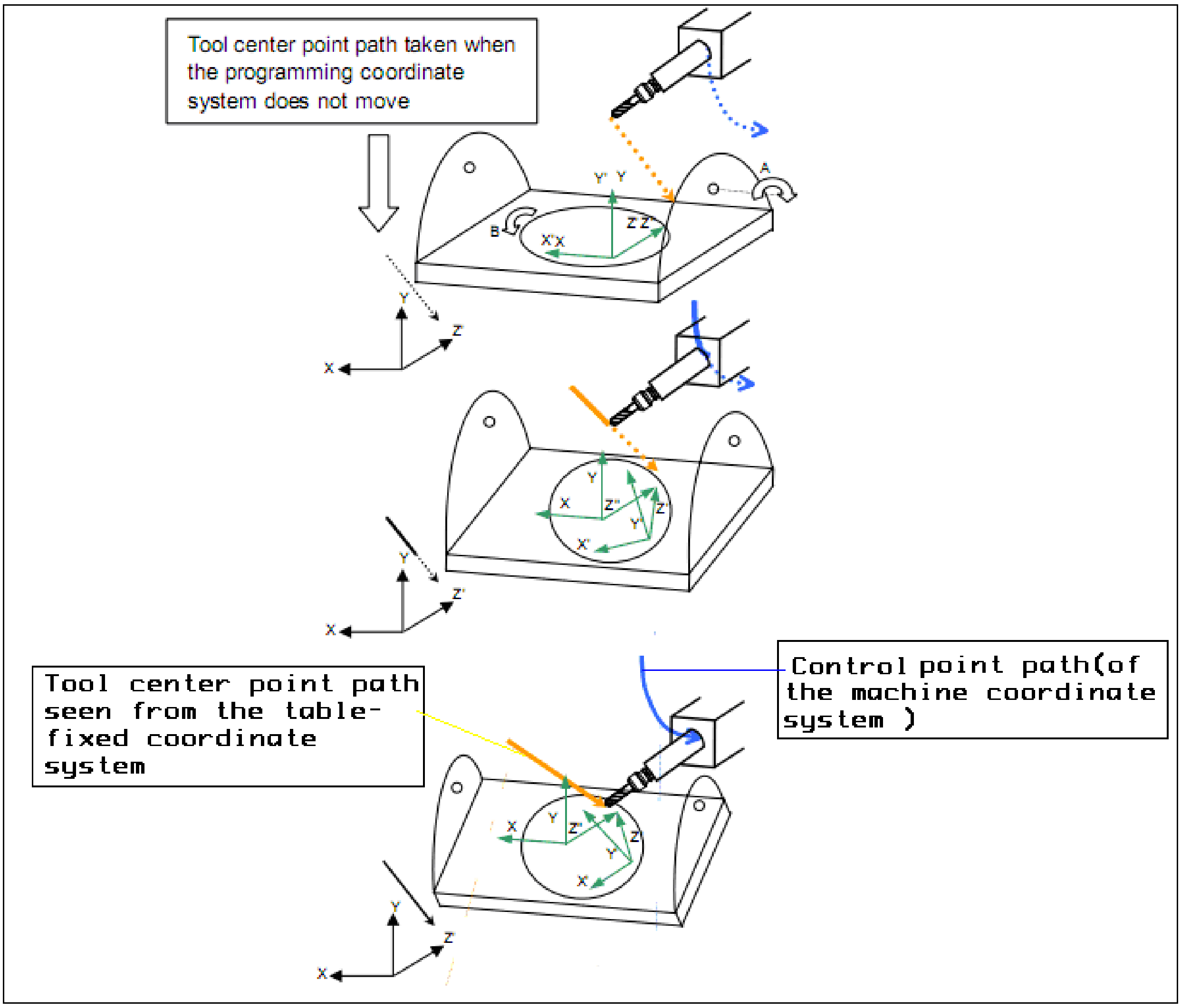
Specify linear interpolation on X, Y, Z axes in the programming coordinate system, the CNC can control the TCP to move linearly towards the worktable (workpiece) while worktable rotation being performed.

The TCP moves towards the worktable (workpiece) at a specified speed.

For this type machine, the rotation of any rotary axis enables the worktable rotation, meanwhile, the workpiece coordinate system does not change, but the programming coordinate system which is fixed on the worktable rotates with it.

N1 G00 G90 A0 B0 ;
N2 G54 ; prepare program coordinate system

N3 G43.4 H01 ; TCP control starts, H01 is the tool compensation number
N4 G00 X20.0 Y100.0 Z0 ; move towards start point
N5 G01 X10.0 Y20.0 Z30.0 A60.0 B45.0 F500 ; linear interpolation
N6 G49; cancel TCP control
N7 M30;



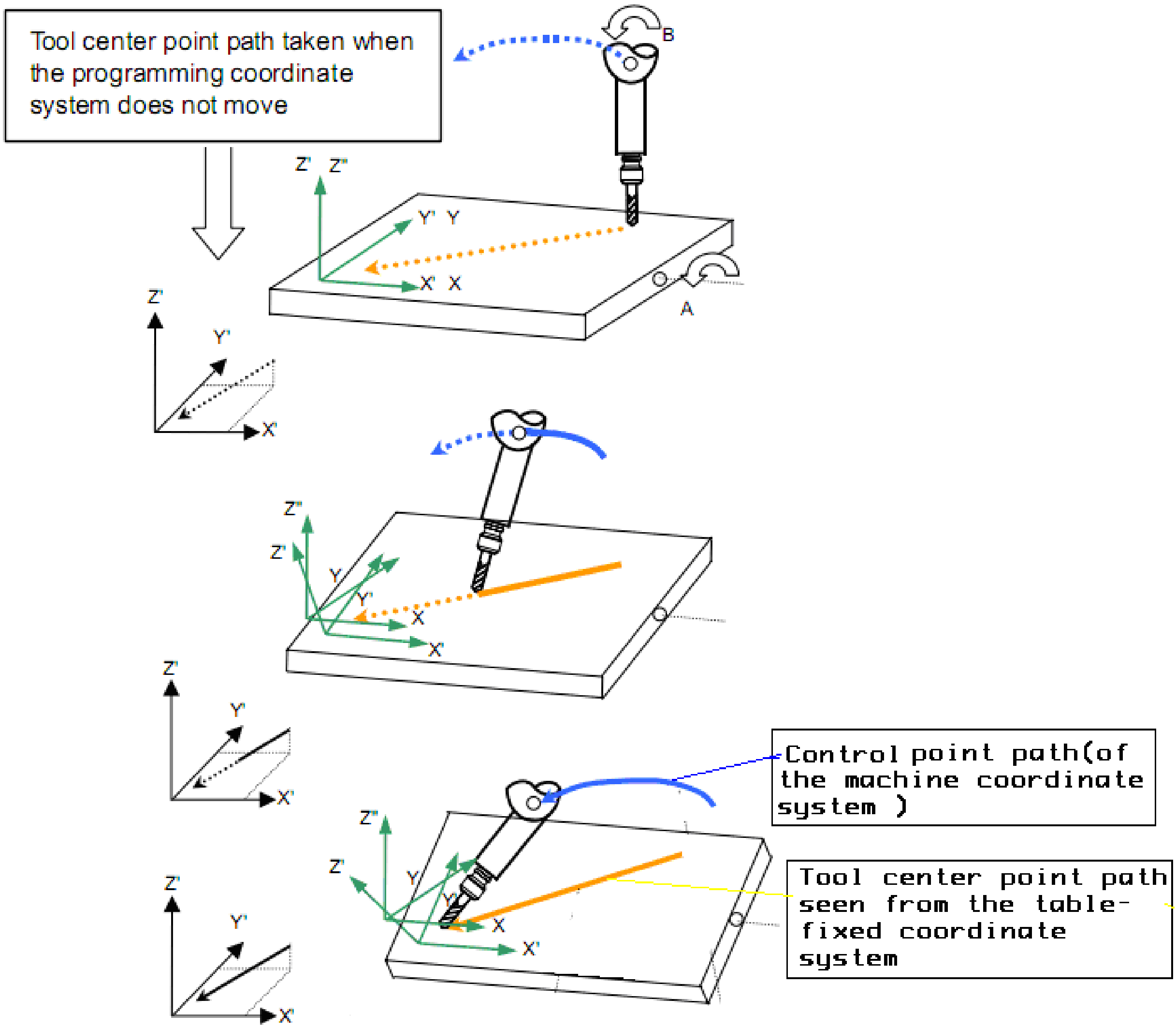
Mixed-type machine

Specify linear interpolation on X, Y, Z axes in the programming coordinate system, the CNC can control the TCP to move linearly towards the worktable (workpiece) while cutter rotation and worktable rotation being performed.

The TCP moves towards the worktable (workpiece) at a specified speed.

For this type of machine, the rotation of worktable rotary axis instead of tool rotary axis enables the rotation of worktable, meanwhile, the workpiece coordinate system does not change, and the programming coordinate system which is fixed on the worktable rotates along with it.

N1 G00 G90 A0 B0 ;
N2 G54 ; prepare program coordinate system
N3 G43.4 H01 ; TCP control starts, H01 is the tool compensation number
N4 G00 X20.0 Y100.0 Z0 ; move towards start point
N5 G01 X10.0 Y20.0 Z30.0 A60.0 B45.0 F500 ; linear interpolation
N6 G49; cancel TCP control
N7 M30;



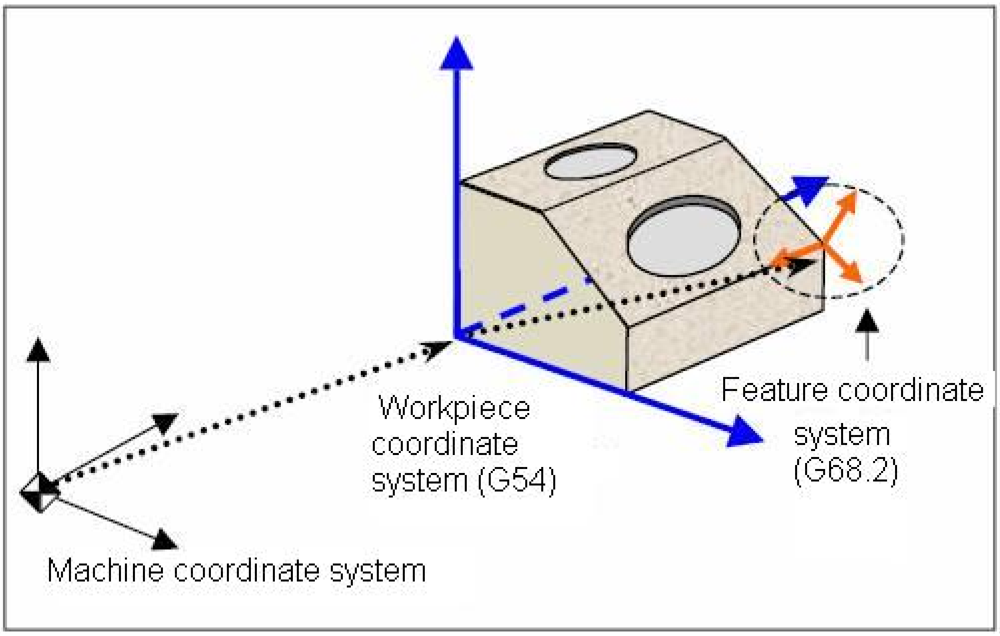
Limitations:

Manual interference

Please do not intervene manually during TCP control.

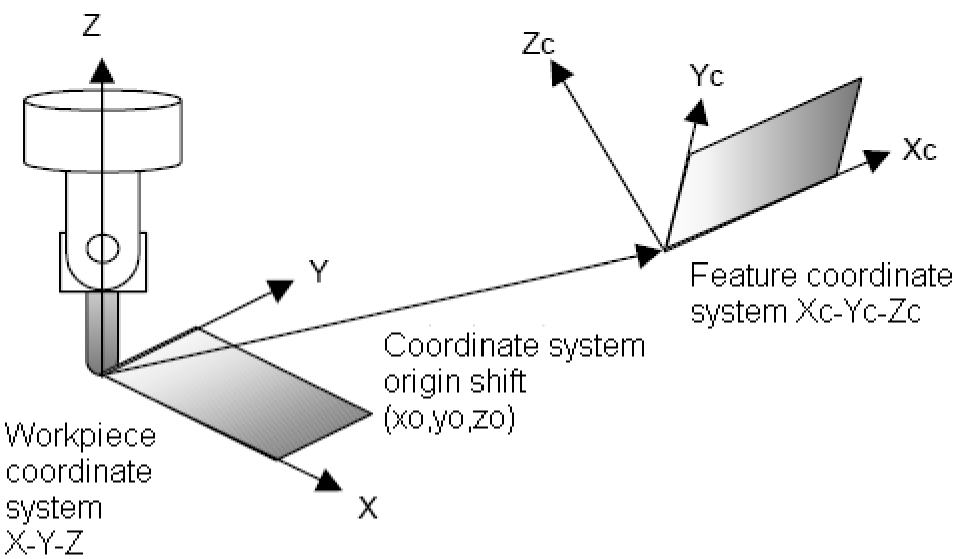
3.9.2 Tilted Working Plane Command

Programming for creating holes, pockets, and other figures in a datum plane tilted with respect to the workpiece would be easy if commands can be specified in a coordinate system fixed to this plane (called a feature coordinate system). This function enables commands to be specified in the feature coordinate system. the feature coordinate system is defined in the workpiece coordinate system.

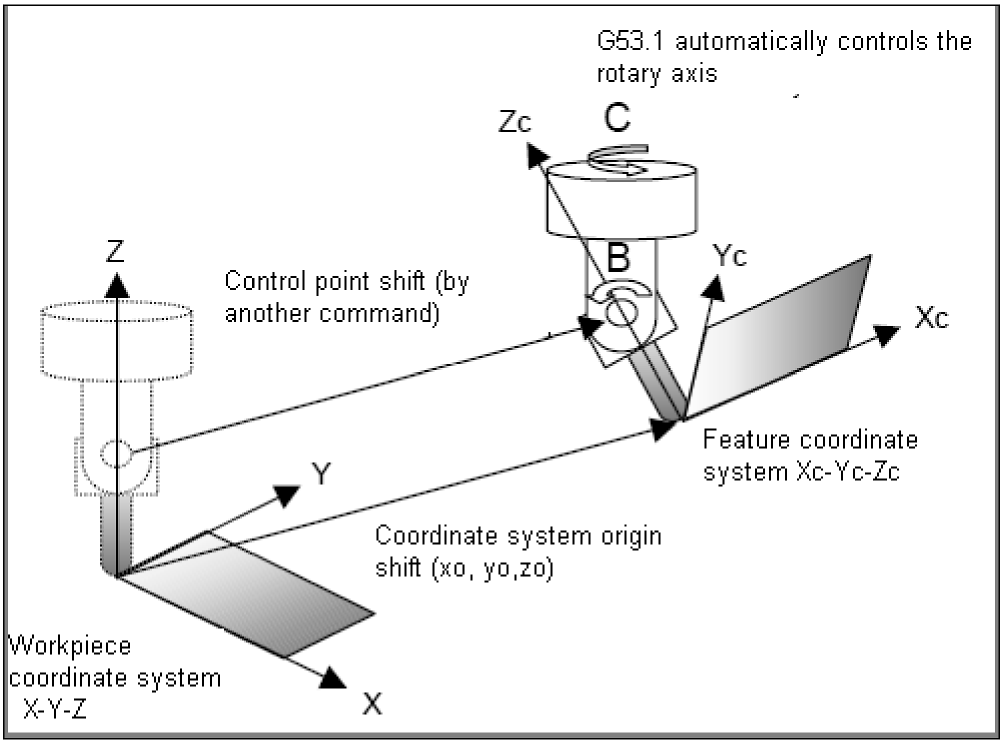


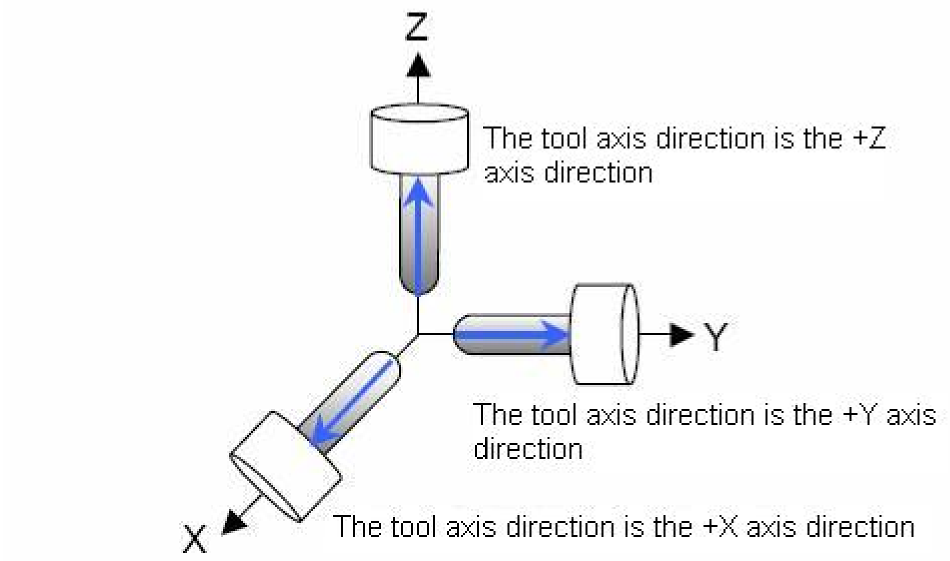
Command G68.2 transfers the part machining coordinate system to “feature coordinate system”, and the coordinates in blocks subsequent are regarded as specified in the feature coordinate system, until G69 is commanded. When G68.2 specifies the relationship between feature coordinate system and workpiece coordinate system in advance and does not specify the an angle for rotary axis, command G53.1 will automatically specify the +Z direction of feature coordinate system as direction of the tool axis. The relationship is shown as follows:

G68.2 Command



The vector of tool axis is directed from tool nose to tool hilt. Shown as follows:





This function can be applied in three types of 5-axis machine tools:

1). Tool rotation type machine—the two rotary axes control the tool.

2). Worktable rotation type machine—the two rotary axes control the worktable.

3). Mixed-type machine—one rotary axis controls the tool while the other rotary axis controls the worktable.

Format:

Set feature coordinate system

G68.2 X <u>x0</u> Y <u>y0</u> Z <u>z0</u> I <u>α</u> J <u>β</u> K <u>γ</u>; (feature coordinate system setting)

Machining commands

G69; (cancel feature coordinate system setting) x0,y0,z0 are the origin points, (absolute coordinates in workpiece coordinate system), I, J, K are Euler angle, used to specify the direction of feature coordinate system.

Tool axis direction control

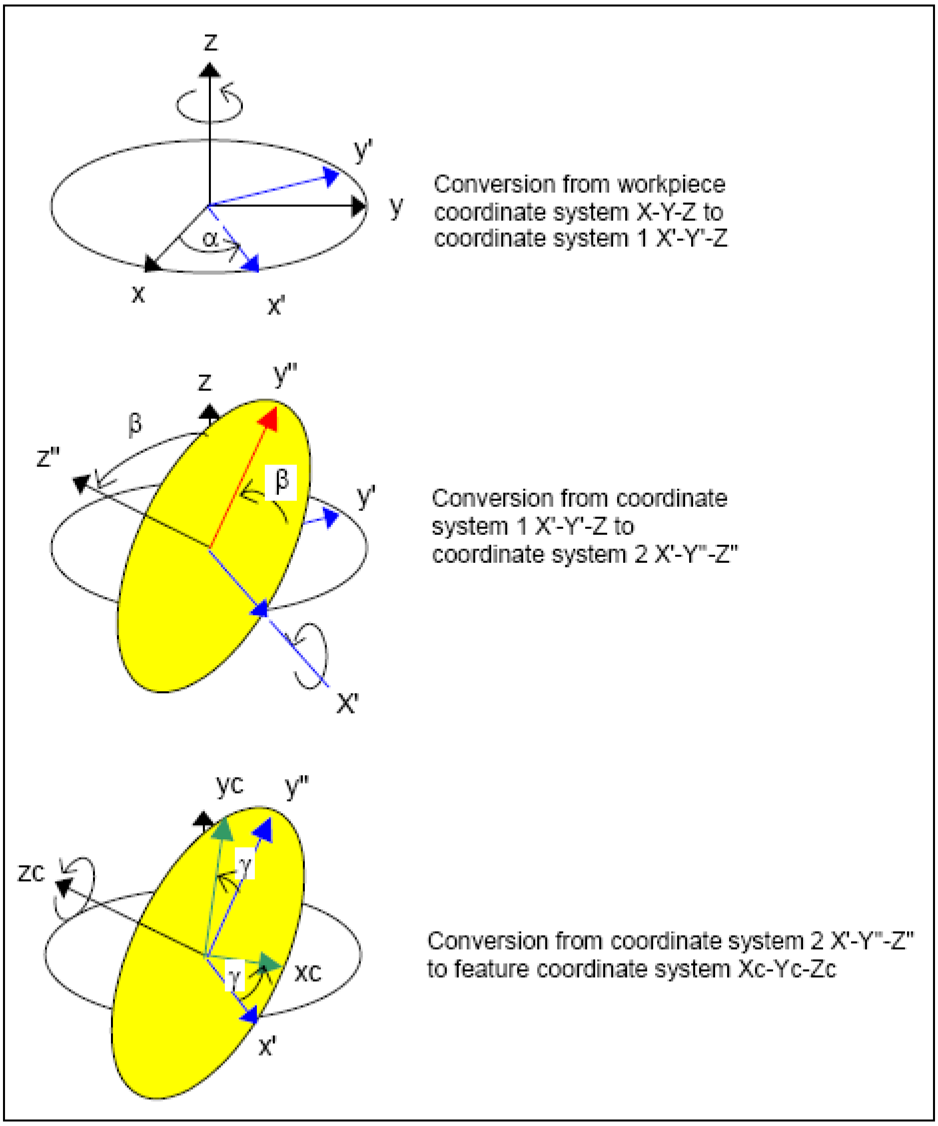
G 53.1 ; (tool axis direction control)

Note:

1. G53.1 should be commanded in a block after the block that contains G68.2. Otherwise, an alarm is generated if G53.1 is specified without G68.2 being specified in a preceding block.
2. G53.1 should be commanded independently.
3. Usually, when G53.1 is commanded, rotary axis moves at the specified cutting feedrate (when cutting feed) or maximum positioning speed (when positioning).

Coordinate conversion using an Euler angle

The conversion of feature coordinate system is performed as it rotates around the origin point of workpiece coordinate system. A rotation of α degree around the Z axis converts the “workpiece coordinate system” to “coordinate system 1”; a rotation of β degree around the X axis converts the “coordinate system 1” to “coordinate system 2”; a rotation (starts from “coordinate system 2”) of γ degree around the Z axis converts the origin point of workpiece coordinate system to (Xo,Yo,Zo). This coordinate system is called “feature coordinate system”. The relationship between “workpiece coordinate system” and “feature coordinate system” is as follows:

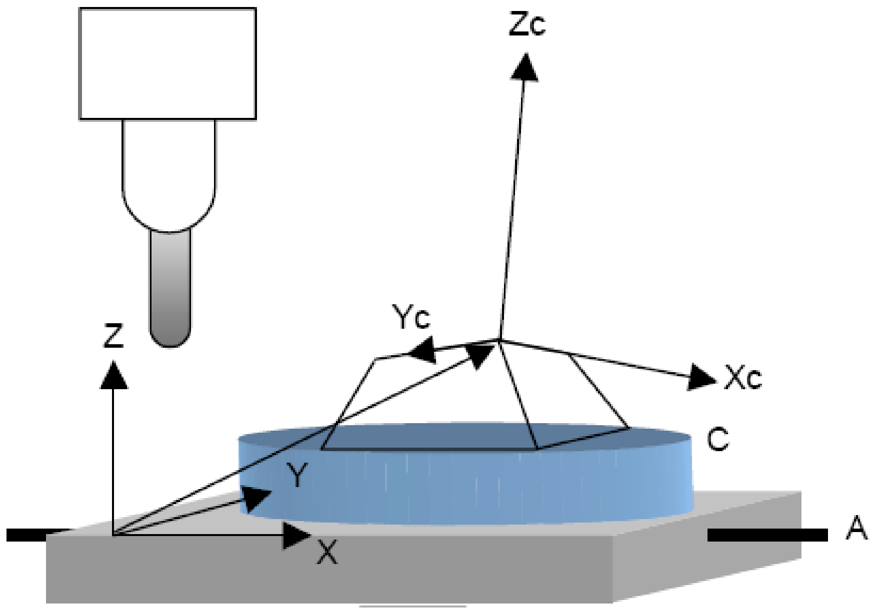


Explanations:

Worktable rotation type

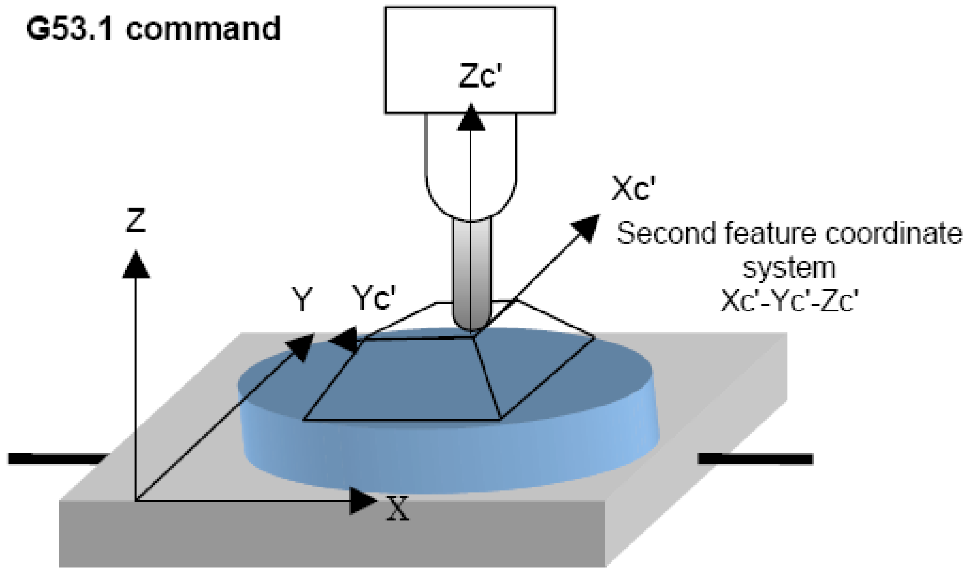
When this function is applied in a worktable rotation type machine, the feature coordinate system Xc-Yc-Zc is set in the workpiece coordinate system based on the coordinate system origin point (x0,y0,z0) and the Euler angle. Command G53.1 calculates and controls the motion of the rotary axis, which converts the direction of tool axis to the +Z direction of the feature coordinate system.

Take the A, C-type worktable rotation machine for example, the feature coordinate system set by G68.2 is shown as follows (only setting, no motion occurs):



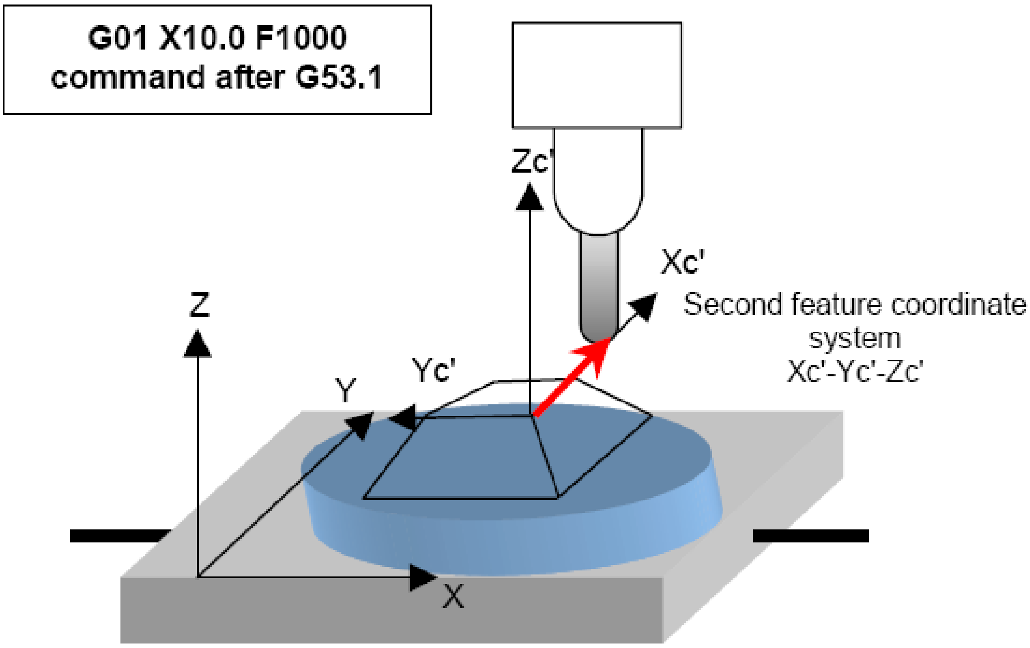
After the feature coordinate system (called the first feature coordinate system) is set by command G68.2, when the table rotates by the G53.1 command, the CNC will control the motion of two rotary axis and convert the tool axis direction to the +Z direction of “feature coordinate system”. The feature coordinate system that has rotated is called the second feature coordinate system. Once G53.1 is specified, the subsequent machining commands are assumed to be specified in the second feature coordinate system. Shown as follows:

- **G53.1 command**



Tool rotation type machine

When this function is applied in the tool rotation type machine, the command G68.2 sets the feature coordinate system, G53.1 controls the tool rotary axis in a such a way that the tool axis will be oriented in the +Z direction of feature coordinate system. Tool length compensation can be performed by specifying G43 after the tool rotates, and the control point will be shifted to the tool center point.



Example: (when the tool axis does not cross the rotary axis)

O100

N1 G54;

N2 G90 G01 X0 Y0 Z100.0 F1000;

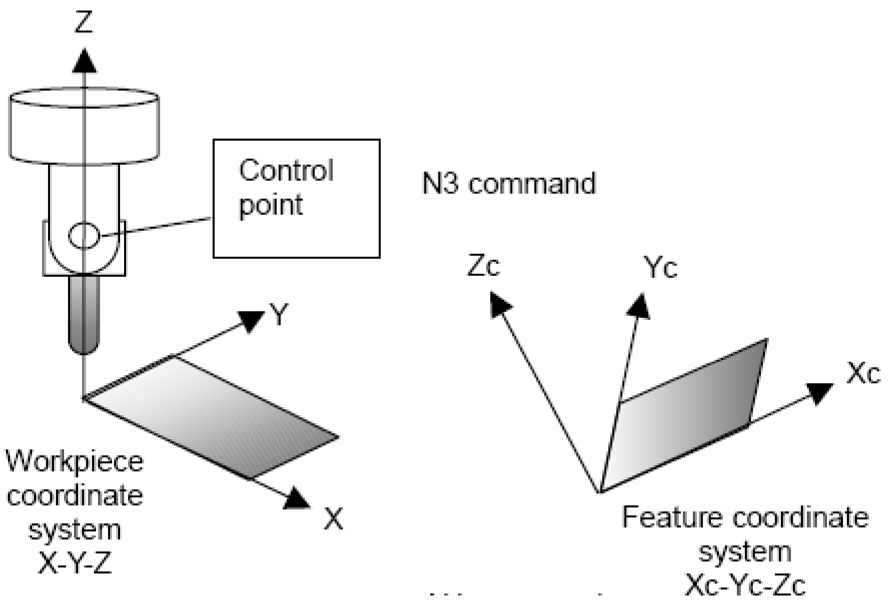
N3 G68.2 X100.0 Y100.0 Z50.0 I30J15.0 K20.0;

N4 G01 X0 Y0 Z30.0 F1000;

N5 G53.1;

N6 G43 H01

N7 G01 X0 Y0 Z0



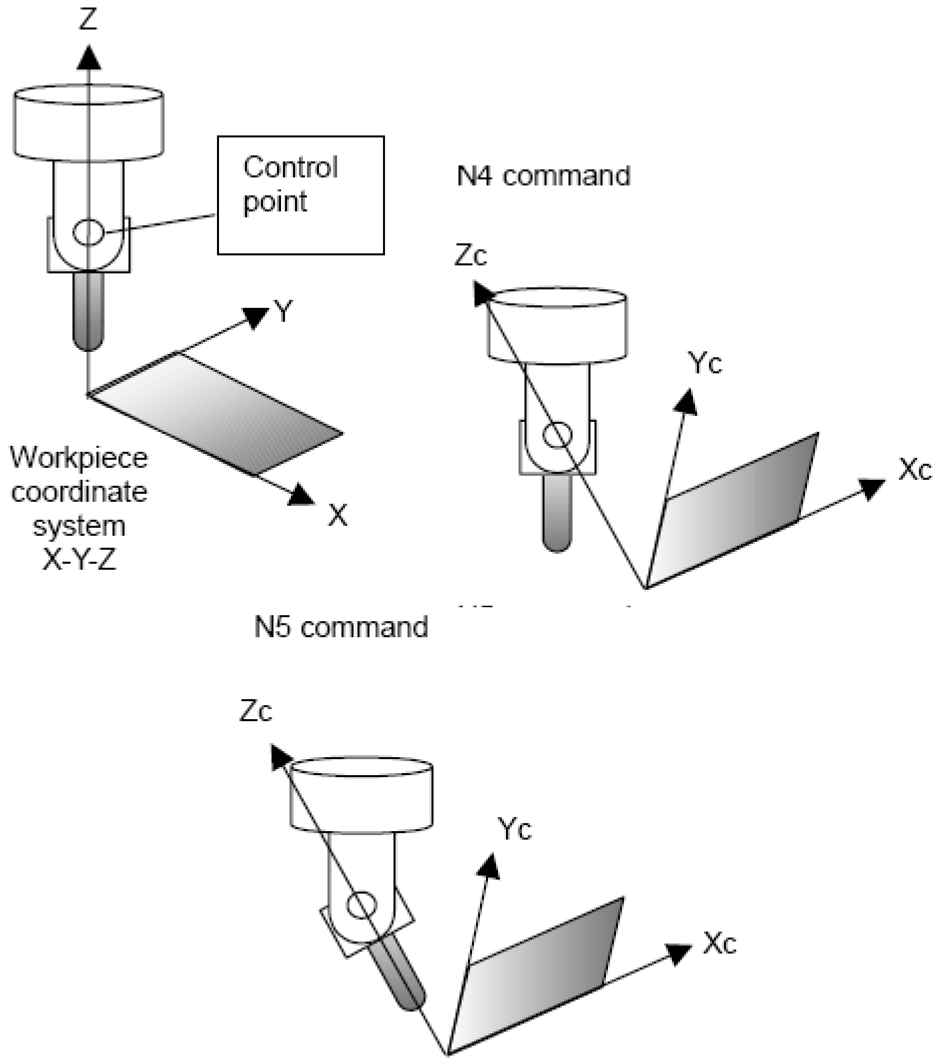
1. Set the feature coordinate system by G68.2. No machine motion occurs.
2. The commands after G68.2 are specified in the feature coordinate system, therefore, the motions are within the feature coordinate system.
3. G53.1 controls the tool rotary axis in a such a way that the tool axis will be oriented in the +Z direction of feature coordinate system
4. G43 performs the tool length compensation after considering the tool length and cross offset vector between tool axis and rotary axis. G43 alone does not produce motion. The tool length compensation is valid only when motion commands are specified after G43.

The motion when only G53.1 is commanded

When G43 is not commanded, the tool length compensation is not performed. The coordinates in the program are the actual coordinates of the control point.

For example:

```
O200;
N1 G54;
N2 G90 G01 X0 Y0 Z30.0 F1000;
N3 G68.2 X100.0 Y100.0 Z50.0 I30.0 J15.0 K20.0;
N4 G01 X0 Y0 Z0 F1000;
N5 G53.1;
```

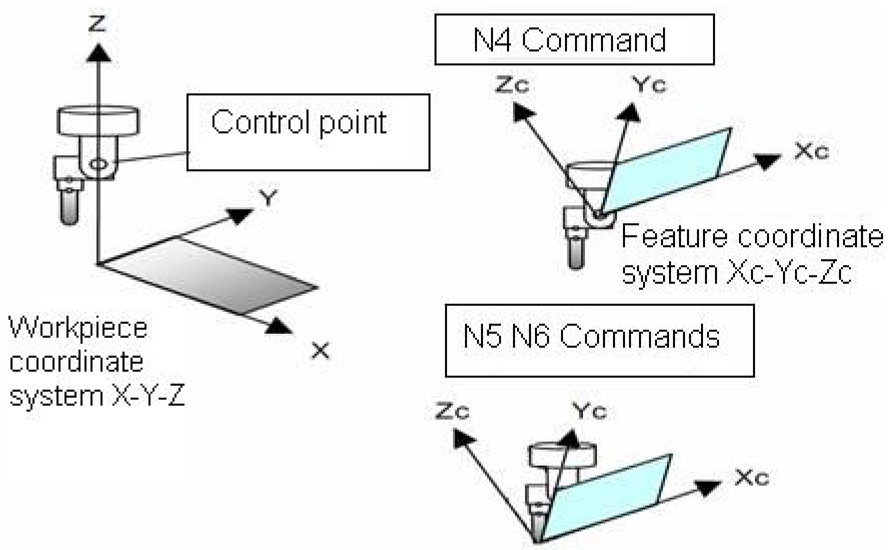


The motion when only G43 is commanded

When only G43 is commanded, the tool axis does not rotate. The tool length compensation is performed in the feature coordinate axis after considering the tool length and the cross offset vector between tool and rotary axis. The G43 does not produce motion. The compensation is performed in the motion command after G43.

For example:

N1 G54;
N2 G90 G01 X0 Y0 Z30.0 F1000;
N3 G68.2 X100.0 Y100.0 Z50.0 I30.0 J15.0 K20.0;
N4 G01 X0 Y0 Z0 F1000;
N5 G43 H01;
N6 G01 X0 Y0 Z0 F1000;



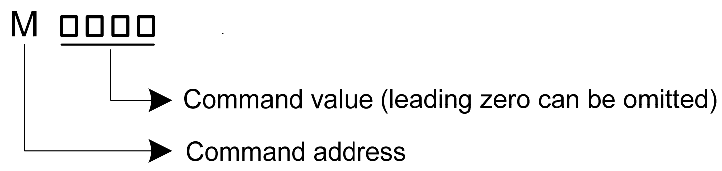
Mixed-type machine

This function is also available for a mixed-type machine in which the tool head rotates on the tool rotary axis and the table rotates on the table rotary axis. The feature coordinate system Xc-Yc-Zc is set in the workpiece coordinate system based on the coordinate system origin shift (xo, yo, zo) and the Euler's angle. G53.1 controls the tool rotary axis in such a way that tool axis will be oriented in the +Z direction of the feature coordinate system. The worktable rotation will convert the feature coordinate system (called the first feature coordinate system) to a new feature coordinate system (called the second feature coordinate system). The tool axis direction is actually the +Z direction of the “second feature coordinate system). By using G43 after G8.2, the tool length compensation is performed in the feature coordinate axis after considering the tool length and the cross offset vector between tool and rotary axis.

4 Auxiliary Function M Function

M command consists of an M command address and its following digits, which is a non-axis movement command in machining, used for controlling the program flow or auxiliary functions output to PLC, such as spindle CW reversion/CCW reversion, cooling on/off, tool exchange, table exchange and so on.

Command format:



There is only one M command in one block.

M command is used for calling macro program (parameter No.: **6071~6089 specified**) and subprogram (M98, M99). M command of macro program, M98 and M99 is called without execution of PLC.

Correspondence between M command value and function are defined by PLC program which is edited by tool machine builder. Please refer to user manual provided by tool machine builder.

4.1 M command for Program Flow Controlling

4.1.1 M00 (Program Stop)

Automatic operation is stopped after M00 is executed. When the program is stopped, all existing modal information remains unchanged. The automatic operation can be restarted by pressing the cycle starting button.

4.1.2 M01 (Optional Stop)

After M01 is executed and optional stop switch on machine operator’s panel is pressed, automatic operation is stopped. It realizes the same functions as above mentioned M00. No operation is executed when optional stop switch is off.

4.1.3 End of Program（M30,M02）

This indicates the end of the main program. In automatic operation mode, main program ends and automatic operation stops when M30 and M02 are executed. The system is on a status of reset, Returning to the start of the program is depended on parameters.

Position parameter **1803 #5** can be used for M30 returning to the start of program and **1803#4** is for M02.

M02, M30 are commands for part counting, part number adds 1 when it is performing.

4.1.4 Subprogram Call（M98）

This code id used for calling a subprogram. It’s format is M98 Pnnnnoooo (nnnn is program used time, oooo is program name).

M98 is internal process of NC. M code and strobe signals are not set to PLC.

4.1.5 End of Subprogram or Cycle（M99）

It is used for called subprogram or macro program return controlling to main program or cycle execution program.

M98 is internal process of NC. M code and strobe signals are not set to PLC.

4.2 M Commands Defined by Standard PLC

When machine tool manufacturer uses GSK standard PLC program, meanings of M commands are as follows:

4.2.1 Spindle CW/CCW Rotation and Stop Commands (M03, M04, and M05)

M03: Spindle CW rotation (positive rotation)
M04: Spindle CCW rotation (negative rotation)
M05: Spindle stop

4.2.2 Cooling on/off Commands （M08,M09）

M08: cooling on
M09: cooling off

4.2.3 Spindle Directional Command (M19)

M19:Spindle orientation is to stop the spindle at a specified angle position.

4.2.4 Rigid Tapping Commands (M29)

M29:The system and the spindle servo are changed into rigid tapping state by this command.

5 FEED FUNCTION

Feed function is to control the traverse speed of the tool, which includes rapid traverse and cutting feed.

5.1 Rapid Feed (Rapid Traverse)

Rapid traverse is defined by G00,G27,G28,G29,G30,G60, which is used for tool rapid poisoning.

Rapid traverse speed is not defined in programming and is set by N1226, and each separately sets the rapid traverse speed.

Allowable feedrate range is depended on the specification of machine, which maximum value is limited appropriately.

Range adjustment keys on panel can be operated as follows:
F0,F25％,F50％,F100%

F0: Set by parameter **N1231**

Note: In G00 block, it is valid though feedrate F is defined, G0 is used to position.

5.2 Cutting Feed

In linear interpolation (G01) and circular interpolation (G02, G03) , the numbers following F code are used to commend the feedrate of tool. The tool moves at the cutting feedrate complied in programming. Cutting feedrate can be adjusted (range: 0％～200%) by override switch on machine operation panel.

1. Feed per minute (G94): Tool feed per minute is specified by setting a number after F.
2. Feed per revolution (G95): Tool feed per revolution is specified by setting a number after F.

5.2.1 Feed per Minute（G94）

Command format:G94 F_

Function: Tool feed per minute. Unit: mm/min or inch/min

Explanations:

1. When G94 (mode of feed per minute) is specified, tool feed per minute is specified by setting a number after F
2. G94 is a modal command, and it is valid until G95 is specified. Feed per minute is default of the system starting.
3. An override from 0% to 200% can be applied with override switch on panel.

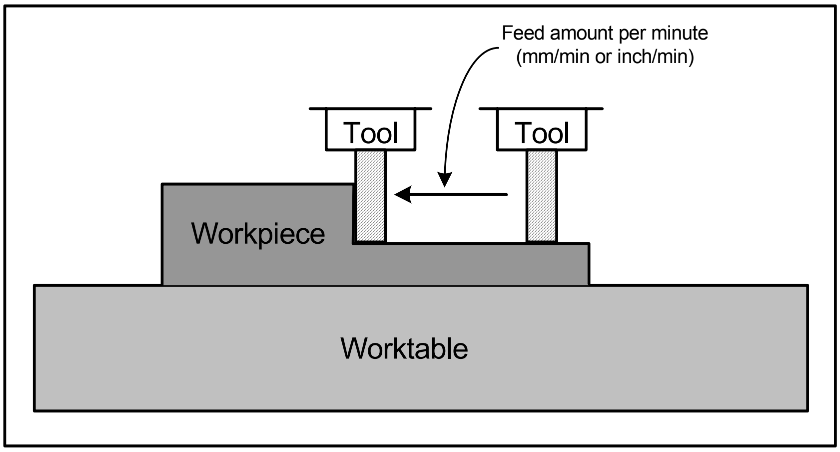


Fig.5-1 Feed per minute

Limitations: some command cannot use the feed per minute, such as thread cutting.

5.2.2 Feed per Revolution（G95）

Command format::G95　F_

Function: Tool feed per minute. Unit: mm/min or inch/min

Explanations:

1. When G95 (mode of feed per revolution) is specified, tool feed per minute is specified by setting a number after F.
2. G94 is a modal command, and it is valid until G95 is specified.
3. An override from 0% to 200% can be applied with override switch on panel.
4. Upper limit of feedrate is set by per minute, and the feed per rev is also limited by federate. And their conversion is as follows:

Fm=Fr×N

Fm: feed per minute

Fr: feed per revolution

N:　Spindle speed

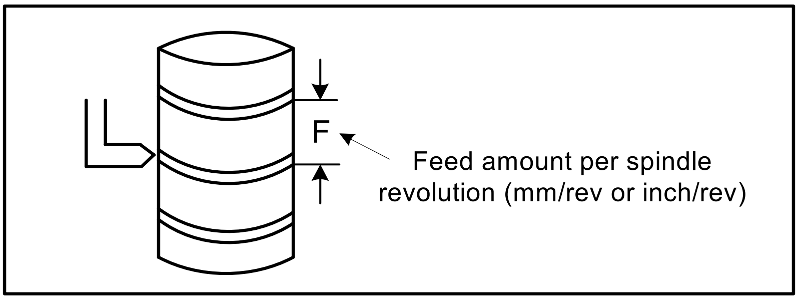
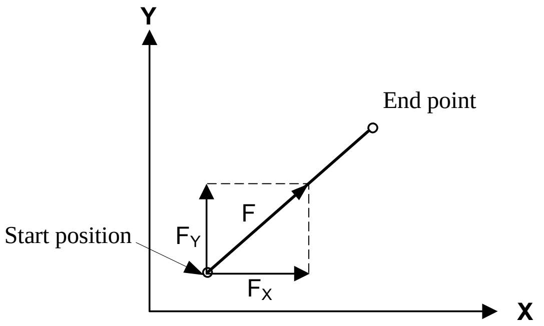


Fig.5-2 Feed per revolution

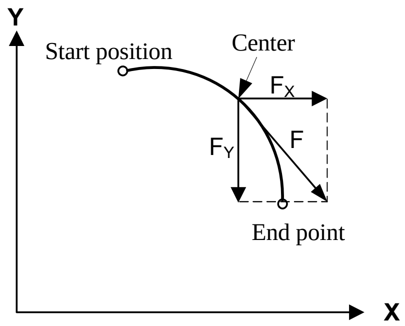
Note: When the speed of spindle is low, feedrate fluctuation may occur. The slower the spindle rotates, the more frequently feedrate fluctuation occurs.

5.3　Tangential Speed Control

Cutting feed controls tangential speed of contour path to reach the feedrate specified by command.



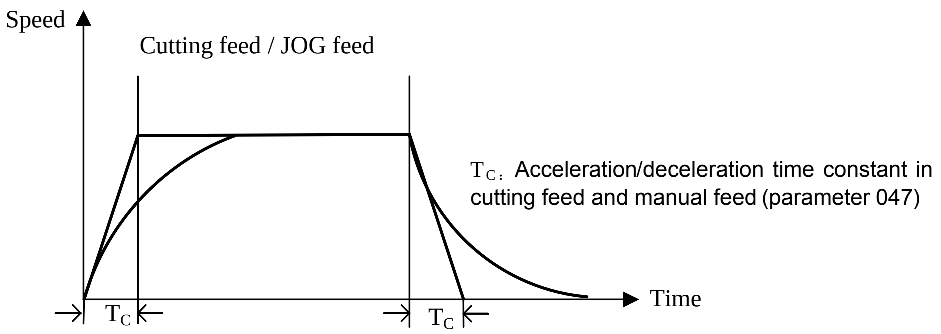
Linear interpolation



circular interpolation

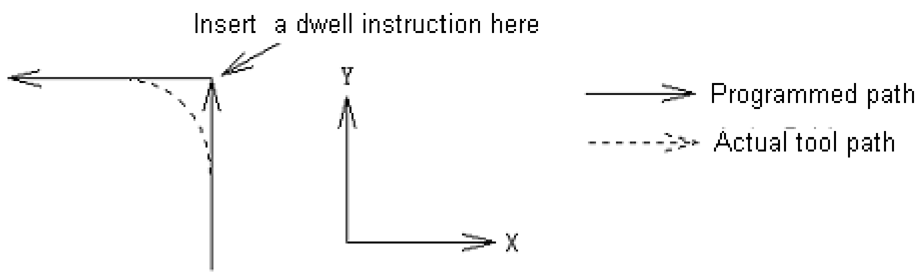
F: feedrate on tangent direction $F = \sqrt{F_x^{\ 2} + F_y^{\ 2} + F_z^{\ 2}}$

F_x: feedrate on X axis direction F_y:feedrate on Y axis direction F_z:feedrate on Z axis direction



5.4 Acceleration/Deceleration Process on the Corner of Program

Example: Only Y moves in the last block, X moves in the next block, X accelerates when Y decelerates, at the moment, the tool path is as follows:



If exact stop command is inserted, tool will be move according to full line on above figure. Otherwise, the bigger the cutting federate is, or the longer the acceleration/ deceleration time constant is, the bigger the arc at the corner is. In arc command, arc radius of actual tool path is smaller than the one specified by the program. On allowable range of mechanical system, reduce acceleration/deceleration time constant should be reduced at the corner.

6 SPINDLE FUNCTION

6.1 Spindle Control

Speed is specified by S code and the number following. After code signal is transferred into analog signal, which is sent to machine used for spindle control.

S □□□□

→ Command value (leading zero can be omitted)

→ Command address

Explanations:

1. The spindle speed can be specified directly by address S followed by a max. five-digit value. Its unit is r/min. For example: M3 S300 indicates that spindle revolution speed is 300 rotations per minute.

2. S is an analog value, which is not cleared in reset, but cleared when power off.

3. Necessary requirements of analog spindle rotation are specified rotation directions of M3 and M4 besides speed specified by S code.

4. When movement command and S command are in the same block, both of them are executed at the same time.

7 TOOL FUNCTION (T FUNCTION)

7.1 Tool Selection Function

Tool selection function consists of address T and the following number, which is used for selecting tools on the machine.

Command format:

T □□□□

Command value (leading zero can be omitted)

Command address

Only one T code can be commanded in a block. For number digits specified the address T and the machine operations corresponding to T, please refer to machine tool builder's manual.

When a movement command and a T code are specified in the same block, the commands are executed in one of the following two ways:

1. Execute the movement command and T code simultaneously.

2. Execute T code after completion of movement command execution.

The selection of either 1 or 2 refers to the machine tool builder's operation manual.

Ⅲ OPERATION

1 OPERATION PANEL

1.1 Panel Division

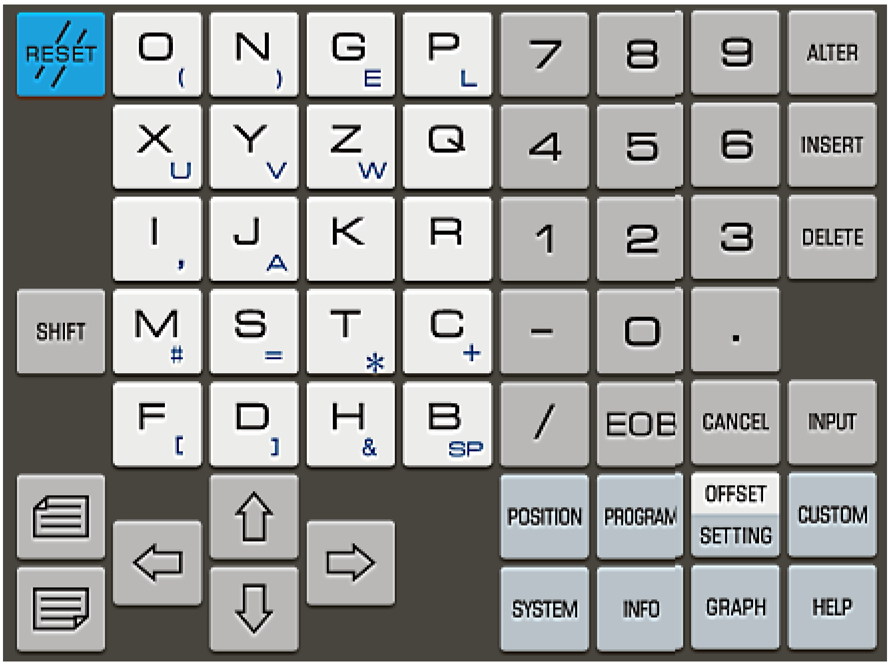
GSK25i is employed an aluminum alloy solid operator panel, which includes LCD (liquid crystal display), edit keypad, menu display and machine operation panel etc..

1.2 Panel Functions

1.2.1 LCD (Liquid Crystal Display)

The system employed color 10.4 inch liquid crystal display, with resolution ratio 800×600.

1.2.2 Edit Keypad



Edit keypad is divided into 10 small areas, function of each area is as follows:

No.	Key name	Function Exlanation
1	Reset key	System reset, feed,output stop
2	Address key	Address input
3	Numerical key	Digt input
4	Cancel key	Delete input characters（unsaved in buffer）
5	Input key	Press this key after number key or address key is pressed,and data is saved in buffer
6	Help	Enter help manual and PMC ladder diagram information
7	Screen operation key	Press any key to enter corresponding interface (details are shown as follows)
8	Page key	For page turning in an interface
9	Cursor move key	Cursor moving up, down, left or right
10	Edit key	For insertion,alteration,deletion of program and word in editing

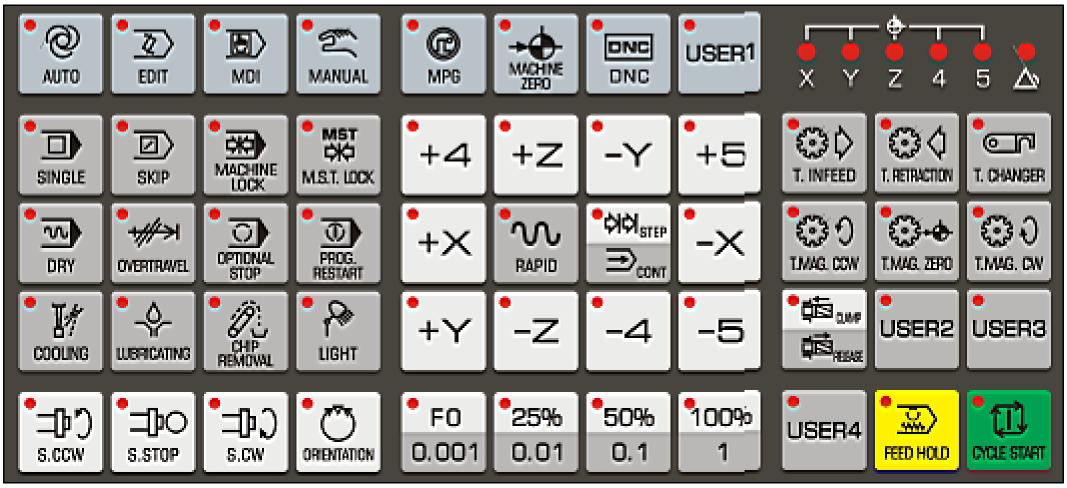
1.2.3 Introduction of Screen Operation Keys

There are 7 page manual display keys on operation panel, see following figure:

Key Name	Function Exlanation	Remark
Position	To enter position interface	Display curren relative pos., absolute pos., inpegrated pos., monitor display pages through soft keys switching.
program	To enter program interface	Display program, MDI, detection, data, file list display pages. Program list is switched by page turining keys.
system	To enter system interface	Switch parameters, diagnosis, PLC through soft key. Check or alter parameters, edit PLC etc.
alarm	To enter alarm interface	Check each alarm information pages through soft keys.
graph	To enter graph interface	Display reference graph, graph pages through soft keys. Graph center, size, proportion and display interface are set here.
help	To enter help interface	Check corresponding information of the system through soft keys.
Offset/ Page set	To enter Offset/ Page set interface	Set tool length compensation, radius compensation and screw-pitch error compensation of each feed axis through soft key switching display. Set coordinate system of work part, macro variables and log-in etc.

Note: By pressing corresponding function keys continuously, above soft key interfaces can be viewed. Please refer to chapter three of this manual for detailed explanation of each page.

1.2.4 Machine Control Panel



Key name	Selection mode	Function explanation	Operation and remarks
Edit	edit mode key	To enter edit mode	Switch to edit mode in automatic operation. The system stops after present block is performed.
Auto mode	Auto mode key	To enter auto mode	The system selects internal memory program in auto mode
MDI	MDI mode key	To enter MDI mode	The switch from auto mode into MID mode is completed after the system has run the current to stop
Machine zero mode key	Machine zero mode	To enter machine zero mode	Being switched to machine zero in auto mode, the system decelerates and stops immediately
Step/continous	Manual step mode key	To enter step mode	This mode is valid only in manual mode
Manual	Manual mode key	To enter manual mode	After being switched from manual mode into auto operation, the system immediately decelerates to stop
MPG	MPG mode	To enter MPG mode	Being switched to MPG mode from auto operation, the system immediately decelerates to stop
DNC	DNC mode	To enter DNC mode	Being switched to DNC

			mode in auto operation, the system immediately decelerates to stop
Spindle CCW Spindle stop Spindle CW Spindle exact stop	Spindle control key	Spindle CCW Spindle stop Spindle CW Spindle exact stop	MPG, step, manual
Spindle override	Spindle override key	For spindle speed adjustment (spindle analog control)	Any mode
Tool magazine zero Tool magazine CCW Tool magazine CW Tool magazine forward Tool magzine backward	Tool magzine key	Tool magzine on/off	Manual mode
clamp/release tool	Key of clamp/releas e tool	clamp/release tool manually	Manual mode
Manual tool change	Manual tool change	Manual tool change	Auto mode
Block Skip key	Block Skip key	For skipping of block headed with“/”sign, if its switch is set for ON, the indicator lights up	Auto mode, MDI, DNC
Single block	Single block switch	Single block/ continues execution switching. if its switch is set for ON, the indicator lights up	Auto mode, MDI, DNC
Dry run	Dry run switch	If dry run is valid, the Block Skip indicator lights up	Auto mode, MDI, DNC
M.S.T. lock key	M.S.T. lock key	If its M.S.T. lock is set for ON, the indicator lights up. M,S,T function is invalid	Auto mode, MDI, DNC
Machione lock key	Machione lock key	If its machine lock is set for ON, the indicator lights up. Axis operation output is invalid	Auto mode, MDI, machine zero, MPG, step, manual mode, DNC
Work light	Work light switch	Work light switch on/off	Any mode
Lubricating	Lubricating switch	Lubricating switch on/off	Any mode

Cooling switch	Cooling switch	Cooling switch on/off	Any mode
Chip removal	Chip removal key	Chip removal key on/off	Any mode
Feedrate override key	Feedrate override key	Feedrate override adjustment	Auto mode, MDI, Edit mode, machine zero, MPG, step, manual mode, DNC
Rapid traverse	Rapid traverse key	Rapid traverse on/off	Any mode
F0 （0．001,0．01 ,0．1,1）	Selection of rapid override, manual step, MPG override.	Rapid override, manual, step, MPG override selection keys.	Auto mode, MDI, machine zero, MPG, step, manual mode, DNC
+X/-X/+Y/-Y/+Z/-Z/+4/-4/+5/-5	Manual feed key	For positive/negative moving of X, Y, Z, 4, 5 in manual/step operation mode. The positive is the MPG selection axes	Machine zero, step, manual mode, MPG
Overtravel release	Key of overtravel release	After the machine moves to press down the hard limit, it alarms, the overtravel release key is pressed, its indicator lights, and the machine reversely moves till the indicator is OFF.	Manual mode
Optional stop	Optional stop ON/OFF	Whether the program with M01 is stopped	Auto mode, MDI, DNC
Feed hold	Feed hold key	Auto operation of the system is stopped by pressing this key.	Auto mode, MDI, DNC
Cycle start	Cycle start key	The system is performed automatically by pressing this key	Auto mode, MDI, DNC

2 SYSTEM POWER ON/OFF AND PROTECTION

2.1 System Power on

Before this GSK980MDa is powered on, confirm that:

1. The machine is in a normal state.
2. The power voltage conforms to the requirement of the machine.
3. The connection is correct and secure.

The following page is displayed on liquid crystal display after GSK980MDa is powered on:

The current position (Absolute POS) is displayed after the self-check and initiation are finished. The parameter N1600#7 can set whether the GSK logo is displayed or not when the system is started.

2.2 Power off

Before power is off, confirm that:

1 X,Y,Z,4,5 axes of the CNC are stopped;
2 Miscellaneous functions (spindle, pump etc.) are off;
3 The CNC power is cut off prior to machine power is cut off.

Before power is off, check that:

1 LED indicating cycle start of the panel is in a halted state;
2 All movable parts on CNC machine are in a halted state;
3 CNC power is cut off by pressing POWER OFF button.

Emergency Power Off

Under the emergency situations during the machine operation, the machine power should be cut off immediately to avoid the incidents. But it should be noted that there may be an error between the system displayed coordinate and the actual position. So the machine zero and toolsetting operation should be performed again.

Note: Please see the machine builder's manual for the machine power cut-off operation.

2.3 Safety Operation

2.3.1 Reset

Press 

key and the system is in the status of reset:

1. All exes movement stops
2. M, S function is invalid
3. By modifying parameter ＃1031, it is able to set whether local coordinate system is cancelled or not after reset. 0: not cancel 1: cancel
4. By modifying parameter ＃1604, it is able to set whether display of DNC operation program is deleted or not after reset. 0: not delete 1: delete
5. By modifying parameter ＃1611, it is able to set whether edited program in MDI mode is deleted or not after reset. 0: not delete 1: delete
6. Processing mode is set by altering parameter ＃1801 after reset. 0: system reset 1: system deletion
7. By modifying parameter ＃1971, it is able to set whether macro common variable #100 - #199 and local variable #1 - #33 are deleted or not after reset. 0: not delete 1: delete
8. Set tool length offset reset by altering parameter ＃2601. 0: not delete 1: delete

2.3.2 Emergency Stop

Under the emergency situations, all axes movement of machine (spindle rotation, cooling) are stopped immediately by pressing emergency stop button. The button is holding on stop position.

Button release modes are different for their different machine tool builders, usually are released by pressing down to turn CW.

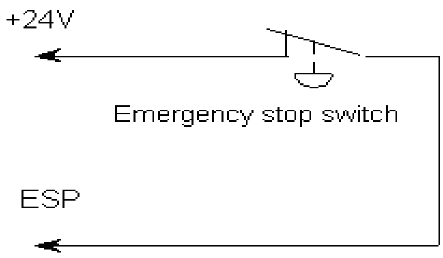
Note 1:Cut off machine power by pressing this button.

Note 2:Control unit is on a status of reset.

Note 3:Troubles are removed prior to button is released.

Note 4:After the button is released, return to reference point with manual operation or G28 command.

General emergency stop signal is normally-closed contact signal. When the contact is disconnected, the system enters emergency stop state, making the machine stop. Circuit connection of emergency stop signal is as follows:



2.3.3 Feed Hold

key（or button）can be pressed to make the running pause when the machine is running. Feed hold indicator lights up simultaneously. It calls for special notice that the running pauses after current command is finished when rigid tapping, cycle command or single block command is executing.

2.4 Cycle Start and Feed Hold

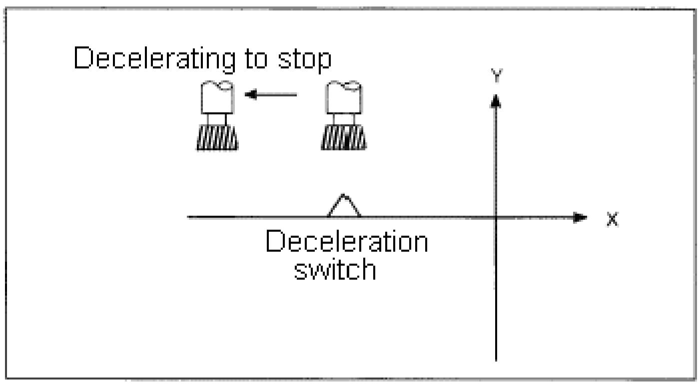
Start and feed hold keys on panel are used for program start and pause operation in auto, MDI and DNC mode.

2.5 Overtravel Protection

Overtravel protection measures should be taken to prevent machine damage due to the overtravel of X, Y, Z axis.

2.5.1 Hardware Overtravel Protection

The stroke limit switches are fixed at the positive and negative maximum stroke position of X, Y and Z axes respectively, If the overtravel occurs, running axis decelerates to eventually stop when it contact with limit switch, and the emergency alarm is issued.



Detailed Explanation

(1) Overtravel in auto mode

In auto mode, all existed axes operation decelerate to stop eventually when tool moves along one axis and contacts with limit switch. Overtravel alarm displays simultaneously. Program stops on current block that overtravel occurs.

(2) Overtravel in manual mode

In manual mode, once any axis of machine contacts with limit switch, all operation of axes decelerate to stop.

2.5.2 Software Overtravel Protection

Software stroke range is set by parameter **NO1880** on data parameter manual. Set coordinate value of machine as reference value. If machine position (machine coordinate) exceeds software stroke range, overtravel alarm will occur. When software stroke range is set by bit parameter N01070 ＃7, alarm occurs (0:before 1: after) overtravel. In manual mode, move axis reversely until the alarm is eliminated.

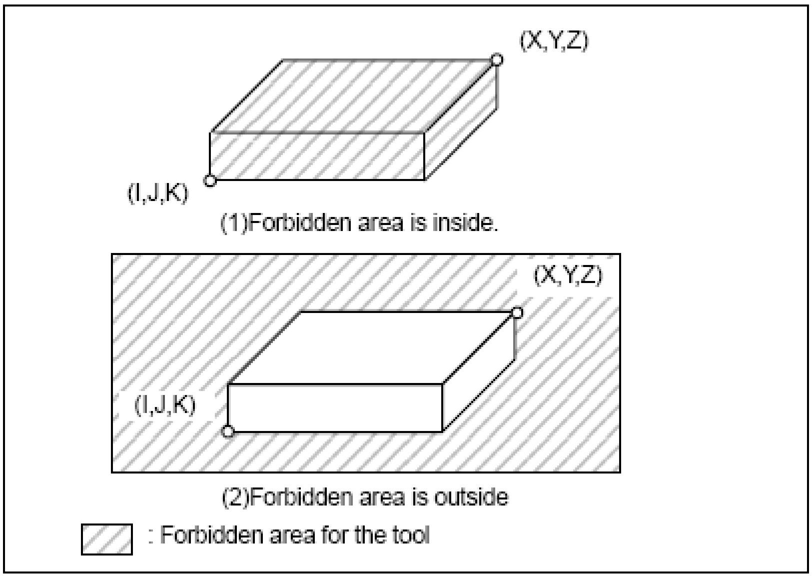
2.5.3 Eliminate Overtravel Alarm

The way to eliminate alarm is press 

key on panel in manual mode, then move axis reversely (move out negatively for positive overtravel, , vice versa)

2.5.4 Stored Stroke Check（G22-G23）

Three areas which the tool cannot enter can be specified with stored stroke check 1, stored stroke check 2, and stored stroke check 3.



(1)Forbidden area is inside.

(2)Forbidden area is outside

When the tool moves into the forbidden area, an alarm is displayed and the tool is decelerated and stopped. When the tool enters a forbidden area and an alarm is generated, the tool can be moved in the reverse direction from which the tool came.

Explanation

1. Stored stroke check 1

Parameters (No. 1080, 1081) set boundary. Outside the area of the set limits is a forbidden area. The machine tool builder usually sets this area as the maximum stroke.

2. Stored stroke check 2

Parameters (No. 1082, 1083) or commands set these boundaries. Inside or outside the area of the limit can be set as the forbidden area. Parameter OUT (No. 1070#0) selects either inside or outside as the forbidden area.

In case of program command, a G22 command forbids the tool to enter the forbidden are, and a G23 command permits the tool to enter the forbidden area. Each of G22 and G23 should be commanded independently of another command in a block.

The command below creates or changes the forbidden area:

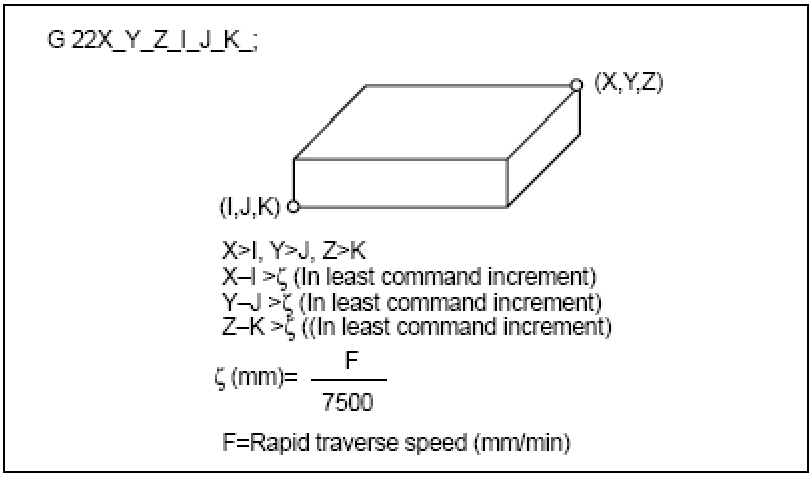


Fig. 2-6 creating or changing the forbidden area using a program

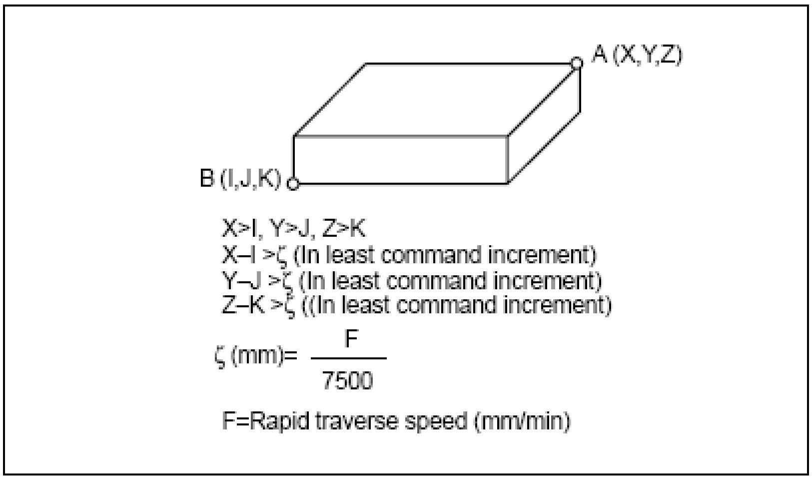


Fig. 2-7 creating or changing the forbidden area using a program

In stored stroke check 2, even the sequence of coordinate value of the two points, a rectangular, with the two points being the apexes, will be set as the area.

When the forbidden area is set by parameter Nos. 1082, 1083, the data should be specified by the distance from the machine coordinate system in the least command increment. (output increment).

If it is set by a G22 command, specify the data by the distance from the machine coordinate system in the least increment (input increment.)

The programmed data are then converted into the numerical values in the least command increment, and the values are set as the parameters.

3. Checkpoint for the forbidden area

Confirm the checking position (the top of the tool or the tool chuck) before programming the forbidden area.

If point A(the top of the tool) is checked in the following figure, the distance "a" should be set as the data for the stored stroke limit function. If point B (the tool chuck) is checked, the distance "b" must be set. When checking the tool tip (like point A), and if the tool length varies for each tool, setting the forbidden area for the longest tool requires no re-setting and results in safe operation.

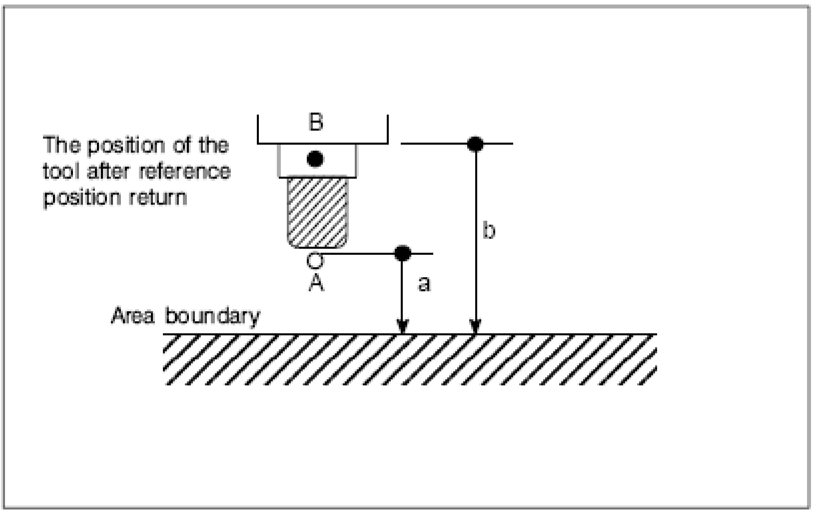


Fig. 2-8 setting the forbidden area

4. Forbidden area overlapping

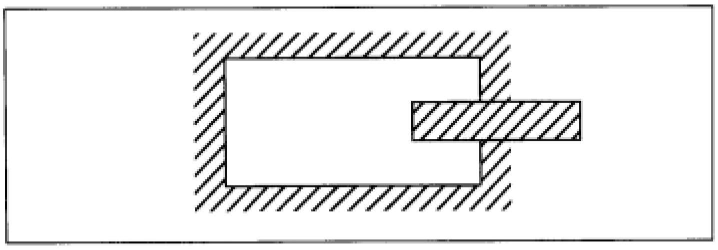


Fig. 2-9 setting the forbidden area overlapping

Unecessary limits should be set beyond the machine stroke.

L(mm)=F/7500

5. overrun amount of stored stroke limit

If the maximum rapid traverse rate if F(mm/min), the maximum overrun amount, L(mm), of the stored stroke limit is obtained from the following expressing:

L(mm)=F/7500

The tool enters the specified inhibited area by up to L (mm).

6. Effective time for a forbidden area

Each limit becomes effective after the power is turned on and manual reference point return or automatic reference point return by G28 has been performed. after the power is turned on, if the reference points in the forbidden area of each limit, an alarm is generated immediately. (Only in G22mode for stored stroke limit 2).

7. Releasing the alarms

If the enters a forbidden area and an alarm is generated, the tool can be moved only in the backward direction. To cancel the alarm, move the tool backward until it is outside the forbidden area and reset the system. When the alarm is canceled, the tool can be moved both backward and forward.

8. Changing from sG23 to G22 in forbidden area

When G23 is switched to G22 in the forbidden area, the following results.

(1). When the forbidden area is inside, an alarm is informed in the next move.

(2). When the forbidden area is outside, an alarm is informed immediately.

Alarms

Alarm No.	Message	contents
500	OVER TRAVEL:+n	Exceeded the nth axis (1-4) + side stored stroke limit 1.
501	OVER TRAVEL:-n	Exceeded the nth axis (1-4) - side stored stroke limit 1.
502	OVER TRAVEL:+n	Exceeded the nth axis (1-4) + side stored stroke limit 2.
503	OVER TRAVEL:-n	Exceeded the nth axis (1-4) - side stored stroke limit 2.

3 INTERFACE DISPLAY AND OPERATION

3.1 Position Interface

3.1.1 Five Ways for Interface Display

Press  to enter Position Interface, there are five modes in this interface such as 【ABS】,【REL.】,【COM.】,【POSC】,【AUTO】. They can be viewed by corresponding soft keys or pressing  key continuously. Detailed information of each interface is as follows:

1) Absolute POS.: Press 【ABS】 key to display current tool position on current coordinate system. It is called "absolute coordinate" hereafter (see fig. 3-1)

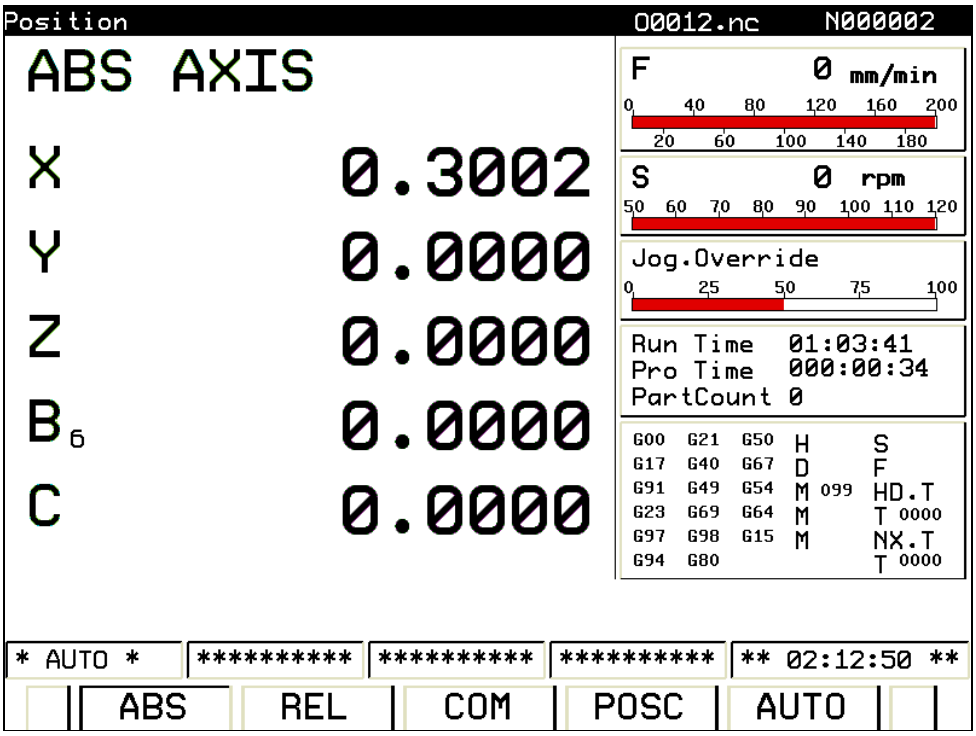


Fig.3-1 Absolute coordinate

Fig.3-1, figure on the left are absolute value on coordinate system. The first progress on the right is federate (F), which can be adjusted by feed override button. The progress of the following S and rapid override are also adjusted by override button with different override values.

2) Relative POS.: relative interface displays current tool position on relative coordinate system. It will be called "Relative coordinate" hereafter. Press soft key【REL】to enter sub-interface of relative interface (see Fig.3-2).

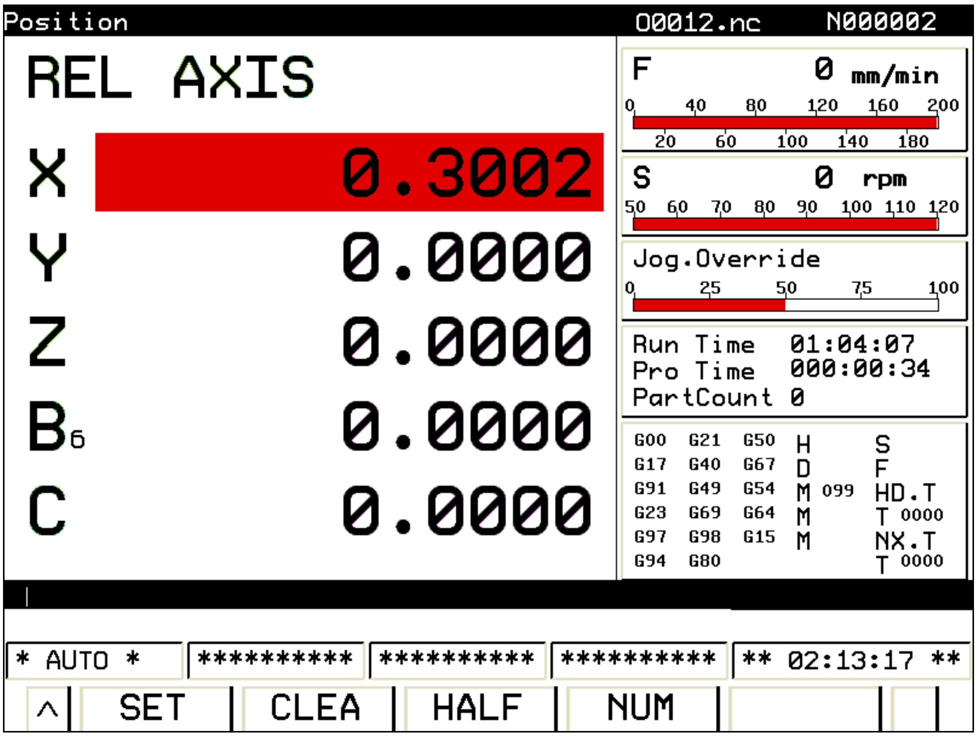


Fig.3-2 Relative coordinate

Preset steps of relative coordinate: chose the axis needs to be altered by up and down direction keys,selected position turns yellow. Input data need to be set to corresponding coordinate by pressing 【SET】key, and cursor will skip to next line.

The clearing steps of relative coordinates system: Select the axis by up and down direction keys, then press soft key 【CLEA】to clear X coordinates.

Centring steps of the relative coordinates system: centring operations are similar to the above ones.

3) Integrated interface

Press【COM 】soft key to enter this interface. The following coordinate position values are displayed:

(A) Position on relative coordinate system;

(B) Position on absolute coordinate system;

(C) Position on machine coordinate system;

(D) Range-to-go

There is other information including speed, operation time, parts counting, current mode and so on. Detailed display page is as follows (Fig.3-3):

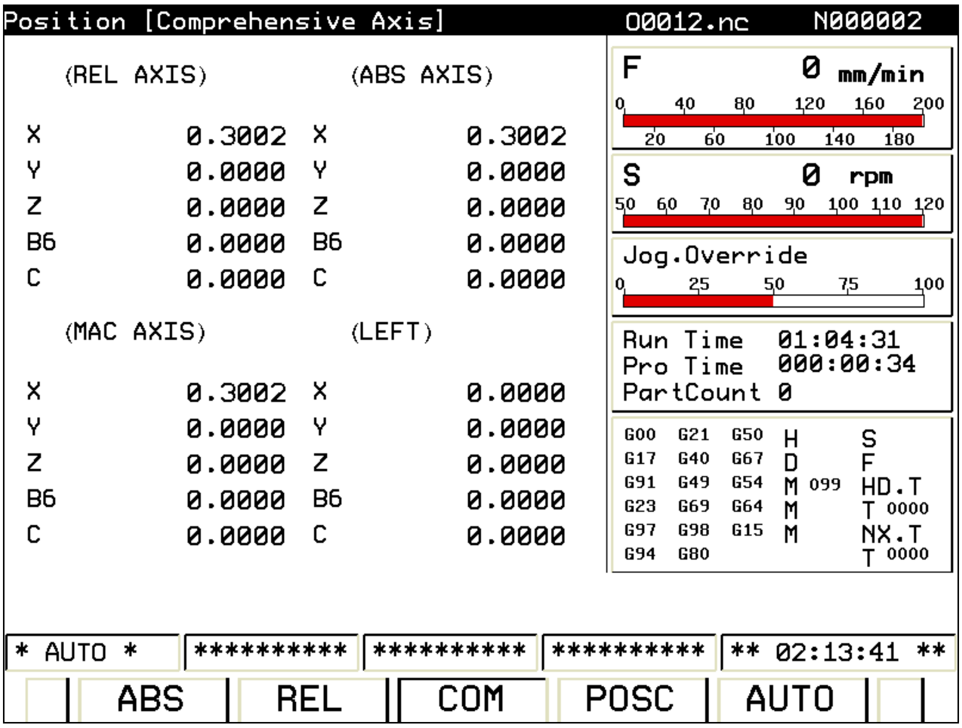


Fig.3-3 detailed display page

4）Monitoring Mode

Press 【POSC】soft key to enter【POSC】interface（see fig. **3-4**）

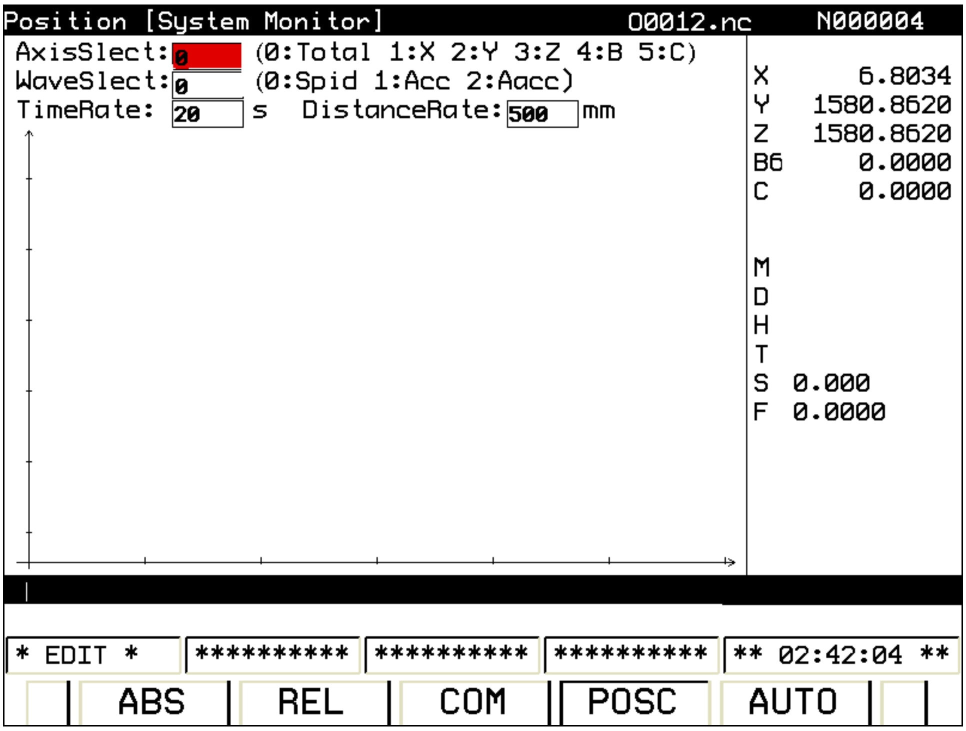


Fig.3-4 Monitoring interface

Each axis perform state can be viewed by altering axis selection parameters, and wave speed, acceleration, jerk are viewed by altering wave shape selection. Wave shape display proportion is changed by altering the proportion of two axes. Among them, cross axis indicates time proportion axis. Each lattice indicates input time block. Vertical axis for distance proportion axis, and each lattice is input distance.

Absolute coordinate value of current operation and some simple modal information are displayed in monitoring interface.

5) Auto-check

Press 【AUTO】soft key to enter this interface (see fig.3-5).

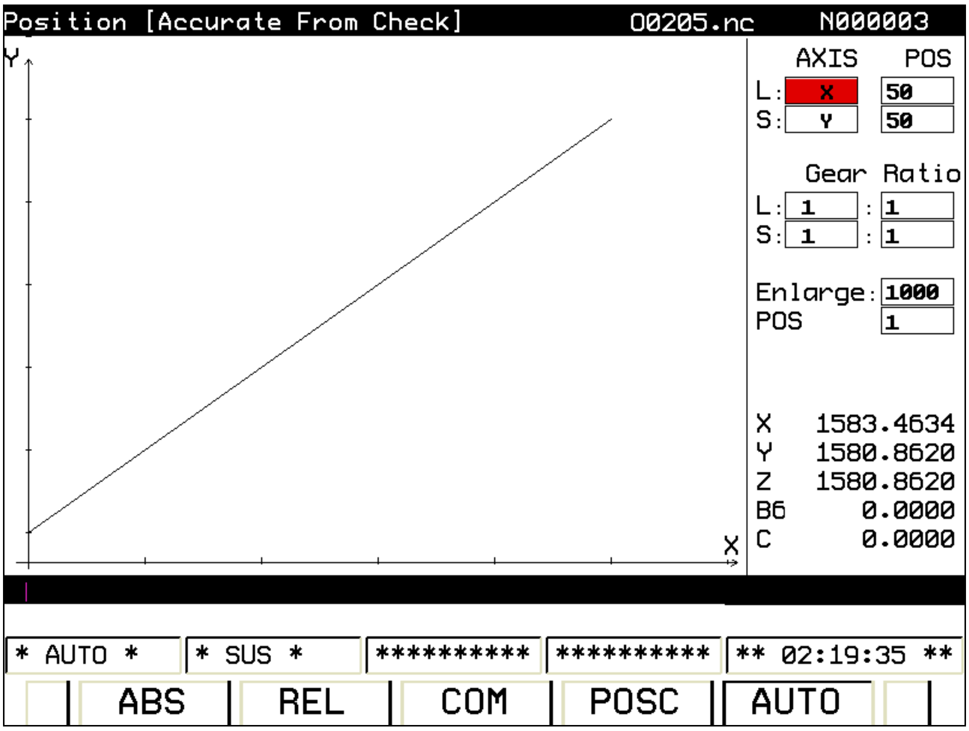


Fig.3-5 Auto-check interface

3.2 Program Interface

Press 

key on panel to enter program page. This page includes five modes such as 【PROG】,【DETECT】,【DATA】,【FILE】and【OPT】. When operation mode is MDI, 【DETECT】 interface changes into【MDI】, and each interface can be viewed and modified by corresponding soft key. Detailed information is as follows:

3.2.1 Program Display

Press 【PROG】key to enter program display interface. Current program block in memory unit is displayed in this interface (see Fig. 3-6)

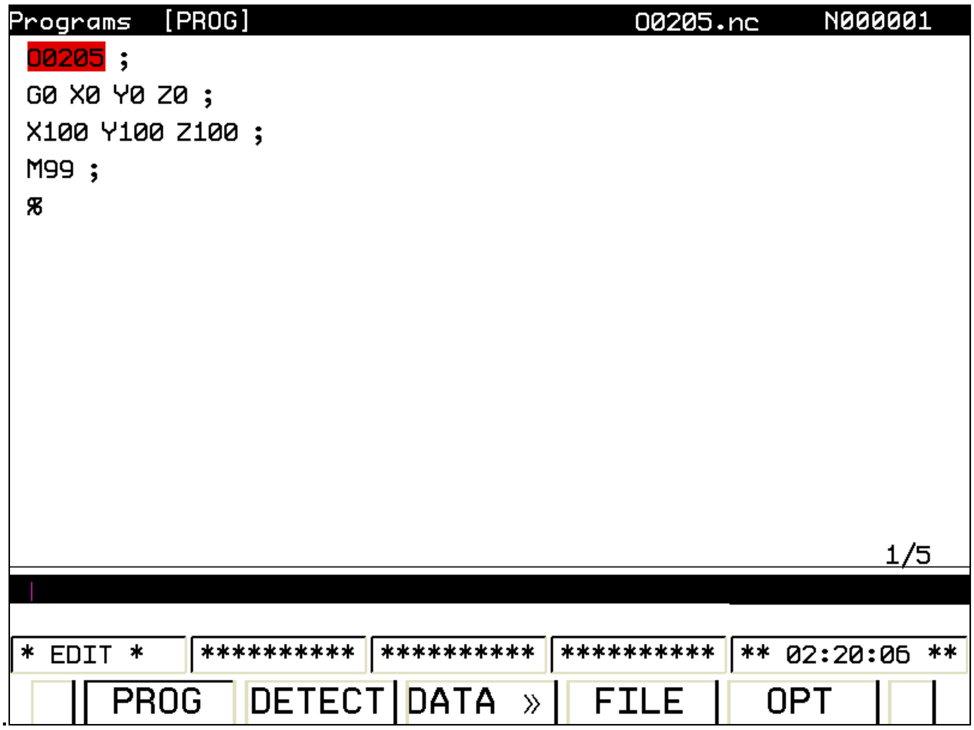


Fig.3-6

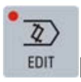
Note: 1）R25.nc indicates program name （This is refer to the program name on file list）
N indicates line number of actual executing codes for current executing codes.
2）1/37336, 1is line number of current execution, 37336 is total lines.

3.2.2 Set up a Program

The maximum program file includes 200,000 lines, occupying 12M of space capacity. If a program capacity is bigger than 8M, this program can be loaded by SD card.

Set “auto number” to 1 by following methods introduced in section 3.3.1. System will automatically insert order number between blocks when a program is edited. Number increment of a specific order number is set by parameter ＃No1621. User can set parameters as required.

Steps of program setting-up:

1. Press  key to enter edit mode.
2. Press  key to enter program interface and 【FILE】soft key to enter file list interface (Fig. 3-7)

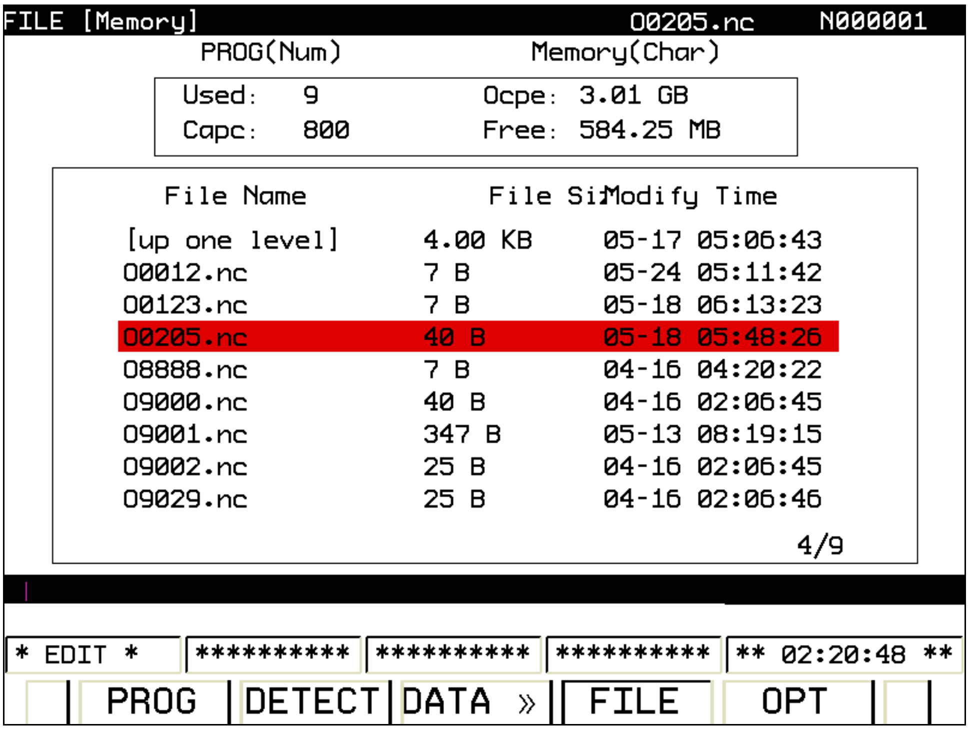


Fig.3-7 file list interface

Input new program name by pressing【OPT】soft key, for example: Input O0001 (generally, input O1, system will edit its name as O0001 automatically after pressing 【NEWP】button, then press this button again (see fig.3-8). The name entered should start with O. Otherwise, flashing alarm will occur: invalid input, capacity or length entered exceed limit. (NC code named by other name, load only by U disk. For detailed operation, refer to file list display 3.2.8)

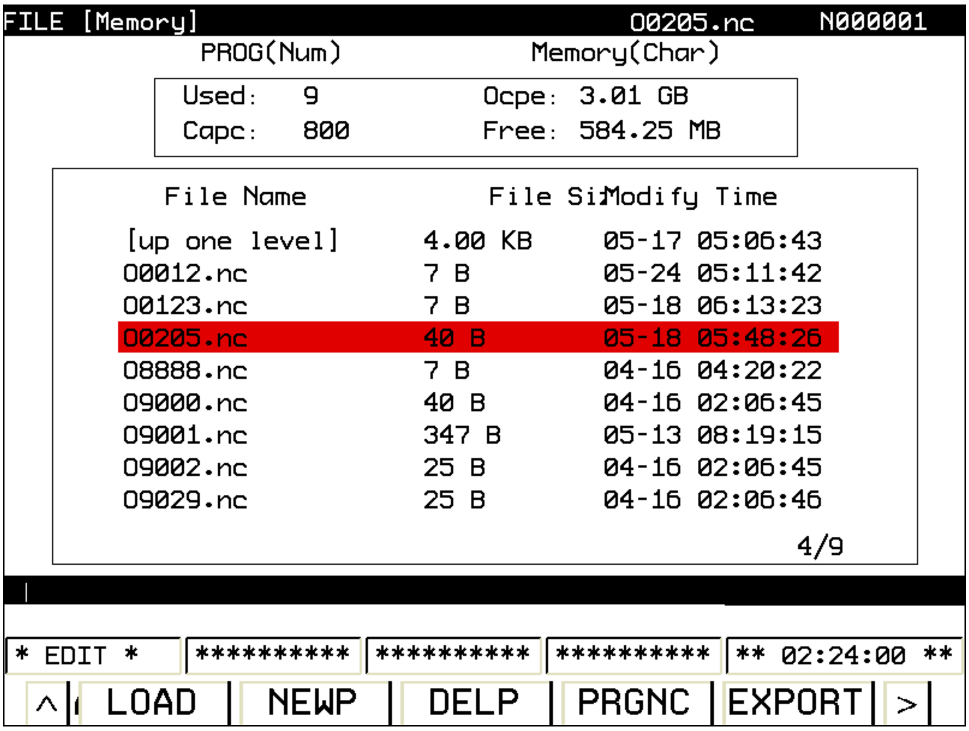


Fig. 3-8

It can be seen from graph, the existed code is needed to load, the cursor should be moved to the code, then press 【LOAD】soft key or change program name directly. When an unwanted name is deleting, the system will prompt to confirm this operation. If the code needed to load in the contents, the cursor is moved to the content, and【OPEN】is selected to open the contents to load the codes as the above operations as Fig. 3-9.

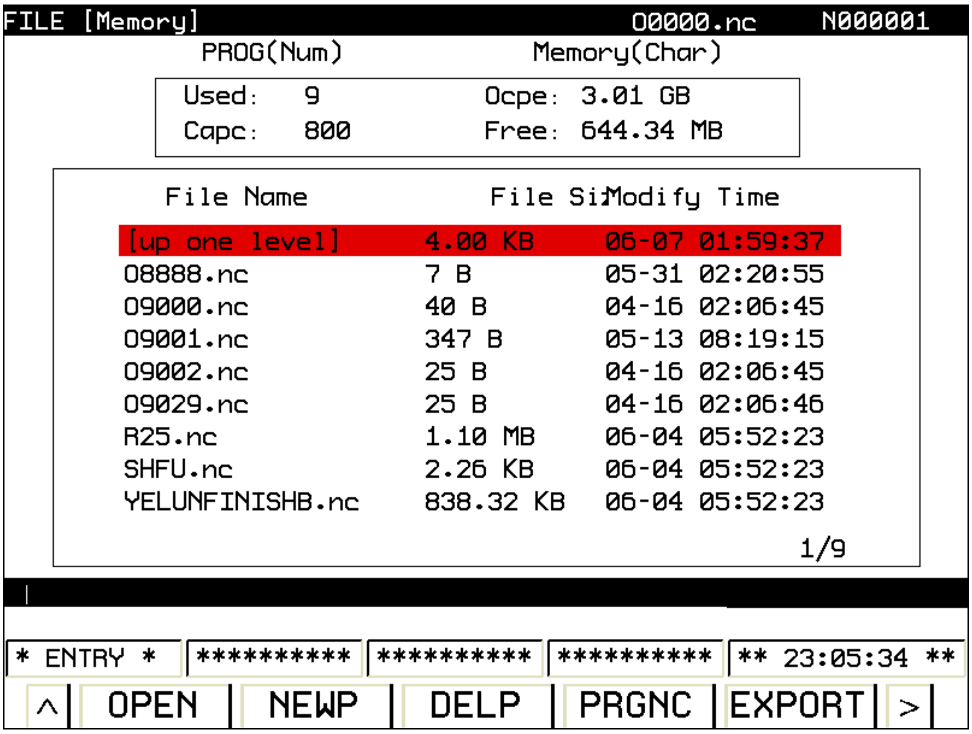


Fig. 3-9

【up one level】in figure indicates entering the last contents.

4. Press 【LOAD】soft key to enter edit interface of program. Code can be written in 【EDIT】 mode.

3.2.3 Edit Program

Setting-up, selection,edit and deletion of part program can be done by editing panel operation.

Part program editing should be done in edit mode. Press  key to enter edit mode. By pressing soft key 【PROG】->【OPT】,the interface turns to edit and modify page.

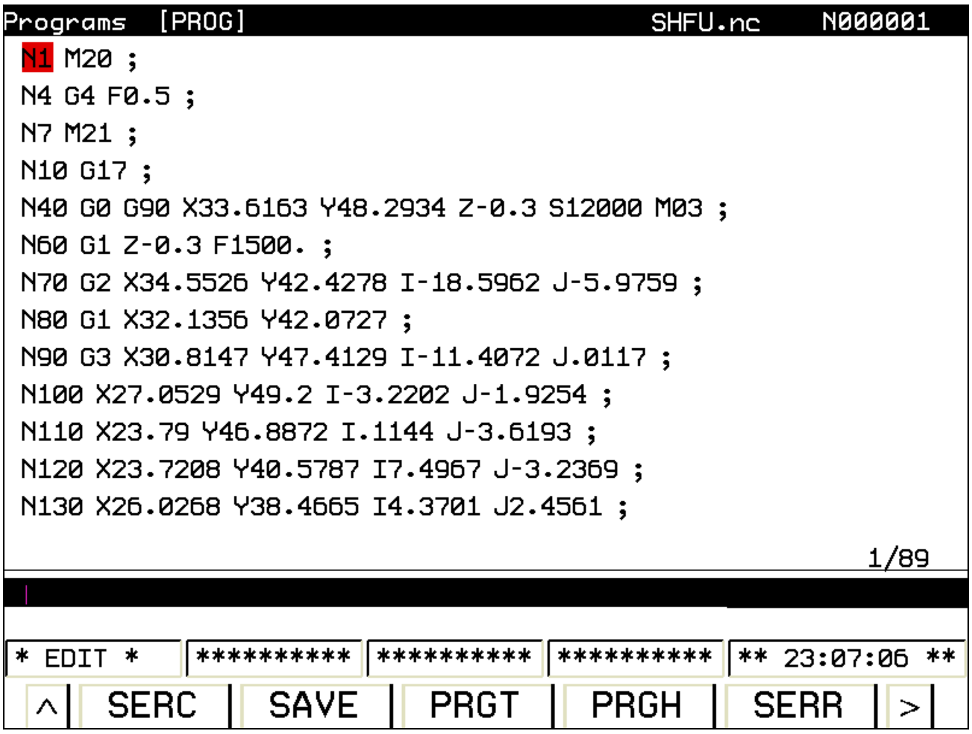


Fig.3-10

Code and line number can be searched here. It can skip to the front/ end of program directly. The system can not save modified program automatically. It is necessary for user to press【SAVE】and 【ENTER】keys to save the modified program. User can press keys to do corresponding operation as required.

Note:

1）In code search, press soft key 【SERC】to do cycle search or press up and down keys to do one-direction searching. Input corresponding code, press ⇧ to search it upwards to the front of program, or press ⇩ to search it downwards to the front of program. After single direction is finished, flashing alarm will occur: search contents does not exist.

Press【▶】to enter next page to edit code (see fig. 3-11).

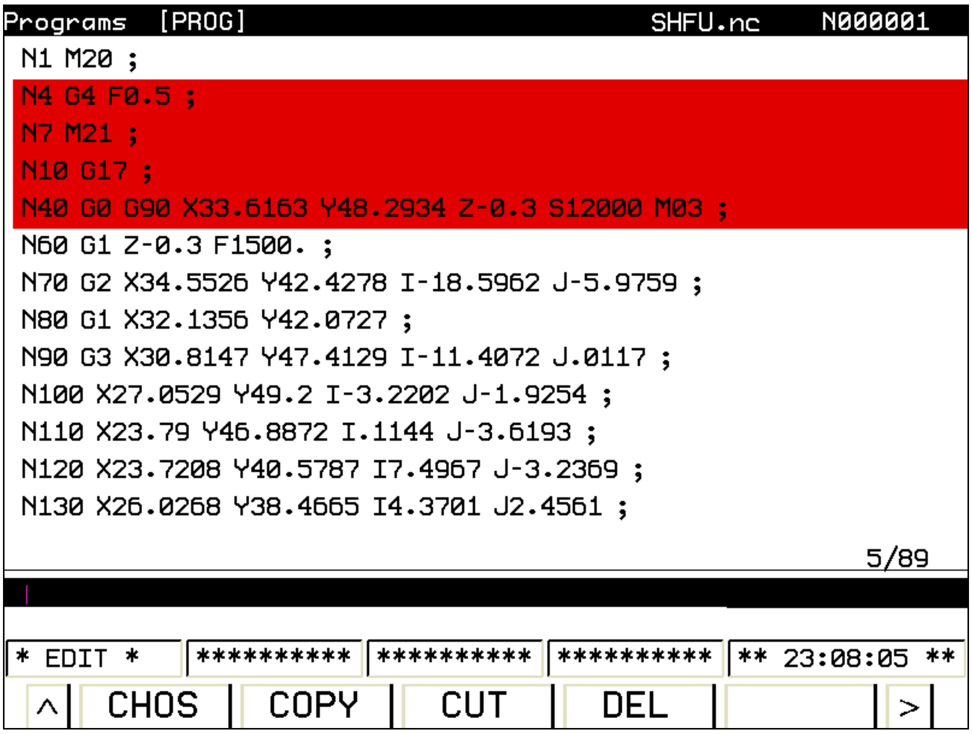


Fig. 3-11

Duplication, shearing, deletion can be done by clicking 【CHOS】key (using up and down direction keys, mutiple-chioce is available). Press 【CHOS】soft key again to cancel selection function.【PAST】soft key will present on blank soft key on the right side for user paste operation. The system supports paste operation for program switching, namely, it is available for user to duplicate on current file and paste to other one. 【PAST】key will be effective and then be cancelled automatically until the user switches the operation mode. For example: Current paste state will be cancelled by shifting EDIT mode to MANUAL mode.

Note:

1）In edit mode, single letter input is available.

2）【DEL】soft key can be used to delete Characters one by one when there is an error.

3）Press SHIFT key, and then press character key to input characters bellow the button.

4）If there is an error, input correct code block and move cursor to wrong code, press ALTER to alter it.

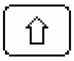
5）If a new code is needed to insert behind a code, move cursor to this code block, input new code and then press INSERT soft key.

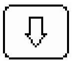
For example: G00 X100.Y100., insert code Z100.behind Y100., move cursor on Y100.and input Z100.. press INSERT key, the code will be G00 X100.Y100. Z100..

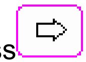
6）One line of code can be selected by 【CHOS】key. Direction key ⇩ can be used for multiple-choice in a single line, and ⇧ is used to cancel the selection of a single line.

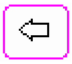
3.2.4 Cursor Positioning

In edit mode, select  key to enter program page.

Press  key to move up the cursor to the last line. When the line which the cursor is in is bigger than the end row of the last line, the cursor can be moved up to the end of the last line.

Press  key to move down the cursor to the next line. When the line which the cursor is in is bigger than the end row of the last line, the cursor can be moved down to the end of the next line longer.

Press  to move right the cursor to one row. When the cursor is in the end of the line, it can be moved to the home of the next line.

Press  to move left the cursor to one row. When the cursor is in the home of the line, it can be moved to the end of the last line.

Press  to Page up the screen and the cursor moves to the first row of the first line in the last screen.

Press  to Page down the screen and the cursor moves to the first row of the first line in the next screen.

3.2.5 MDI Input Display

In MANUAL mode , select  key and press 【MDI】soft key to enter MDI display interface. In this interface, muti-block program can be edited and performed, program format is the same as edit program . MDI is used for simple program test operation (see fig.3-12).

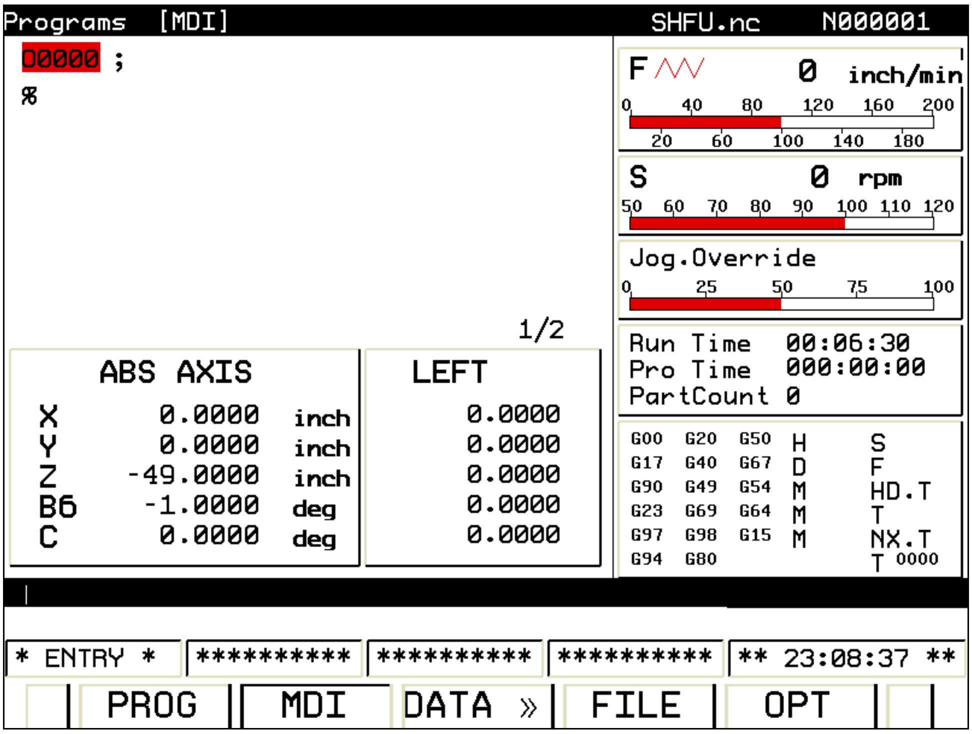


Fig.3-12

Main operation points of MDI are as follows:

1) Press MDI mode switch.

2) Press 

function key on panel to select program screen. Program number O0000 will be added automatically.

3) Use a general program edit operation to compile a program to be execute. Insertion,alteration, deletion and search line number can be used for programming in MDI mode. (refer to introduction of edit interface for detailed operation).

4）To delete program edited in MDI mode, refer to operation in edit interface.

5) Move the cursor to the line to be executed, press cycle start button on panel. Program starts operation until it reaches the end of statement (02 or M30).

6）To stop or end MDI operation in the middle of operation, please follow the following steps:

a. Stop MDI mode

Press feed pause switch on the operation panel. Feed pause indicator lights up and cycle start indicator turns off. Responses of machine are as follows:

（i） When machine is running , the feed operation decelerates to stop.

（ii） Tool stop state is interrupted when machine on this state.

(iii) When M, S or T command is executing, operation stops after M, S, T execution is finished.

When cycle start button on operation panel is pressed again, machine is restarted.

b. End MDI mode.

By pressing 

key on MDI panel, auto operation is stopped. The system enters reset state. When reset command is executed on machine operation, operation decelerates to stop.

Note: Edited program in MDI mode can not be saved.

3.2.6 Data Display

Press 【DATA】soft key to enter data display interface. Command value and modal value are displayed on current executed block (fig. 3-13). Modal state of current executed program is displayed in MDI mode.

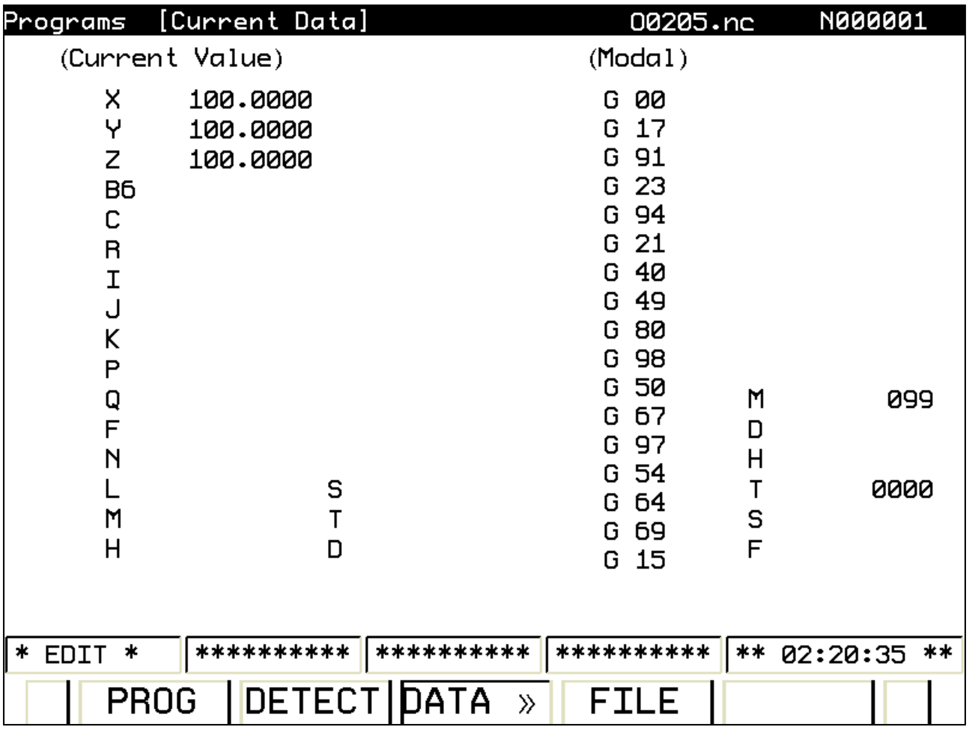


Fig.3-13

Press【DATA】key again to enter the data interface in the next block and the system displays the command value and modal value of next block following the block which is being executed.

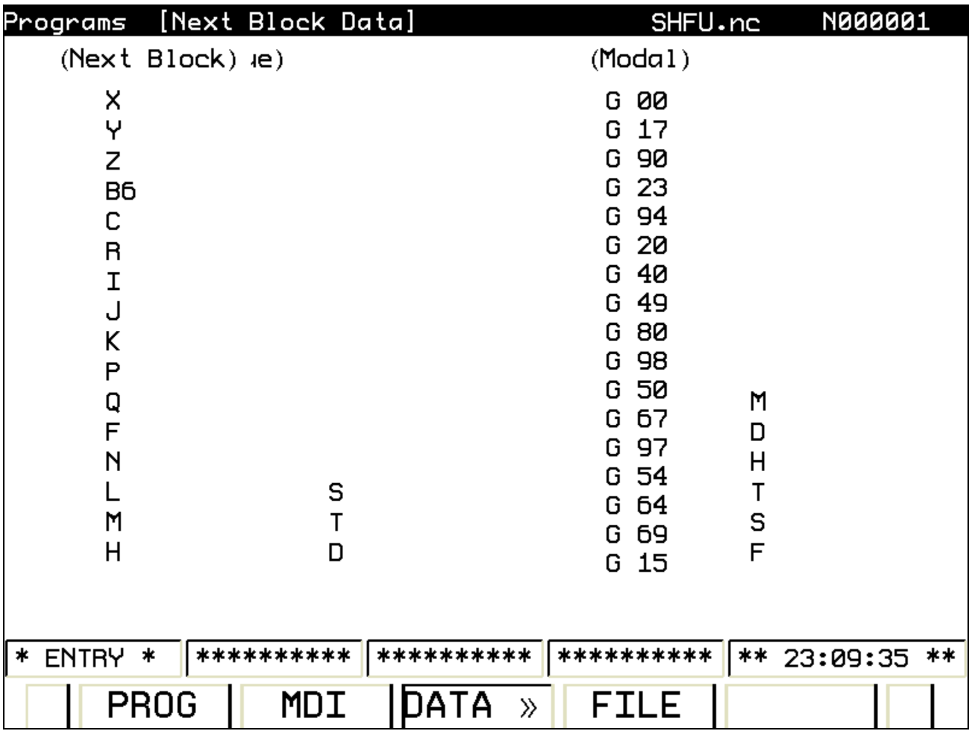


Fig.3-14

3.2.7 Detection Interface

In EDIT mode, press 【DEL】key to enter detection interface. The whole code execution procedure, coordinates of absolute position and remain momentum, spindle speed, feedrate, tool number, and modal can be viewed in detection interface at real time. See fig. 3-15. In this interface, each override can be altered by corresponding button on operation panel.

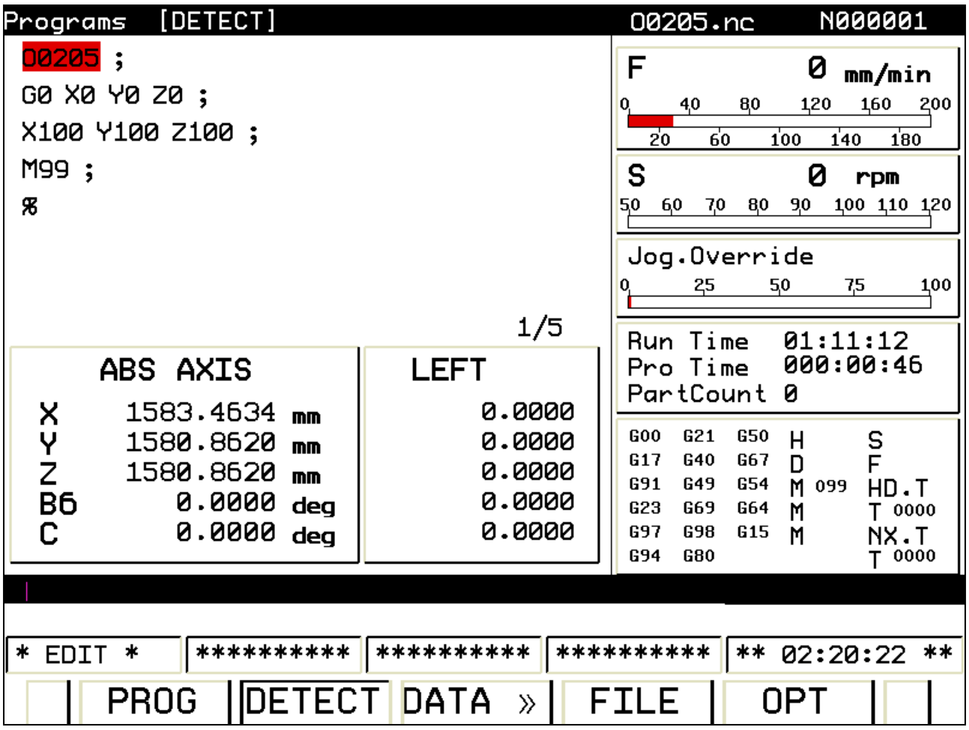


Fig.3-15

3.2.8 File List Display

Press 【FILE】 soft key to enter file list display interface. Following contents can be seen here (see fig.3-16):

(a) Used capacity: number of saved program (including contents)

Capacity: Total number of programs that can be saved in the system. The number of program that can be saved is 200 at present.

(b) Used capacity: Capacity that has been occupied by saved program (displayed by character number).

Available space: Capacity that can be used. Present maximum capacity is 30 M.

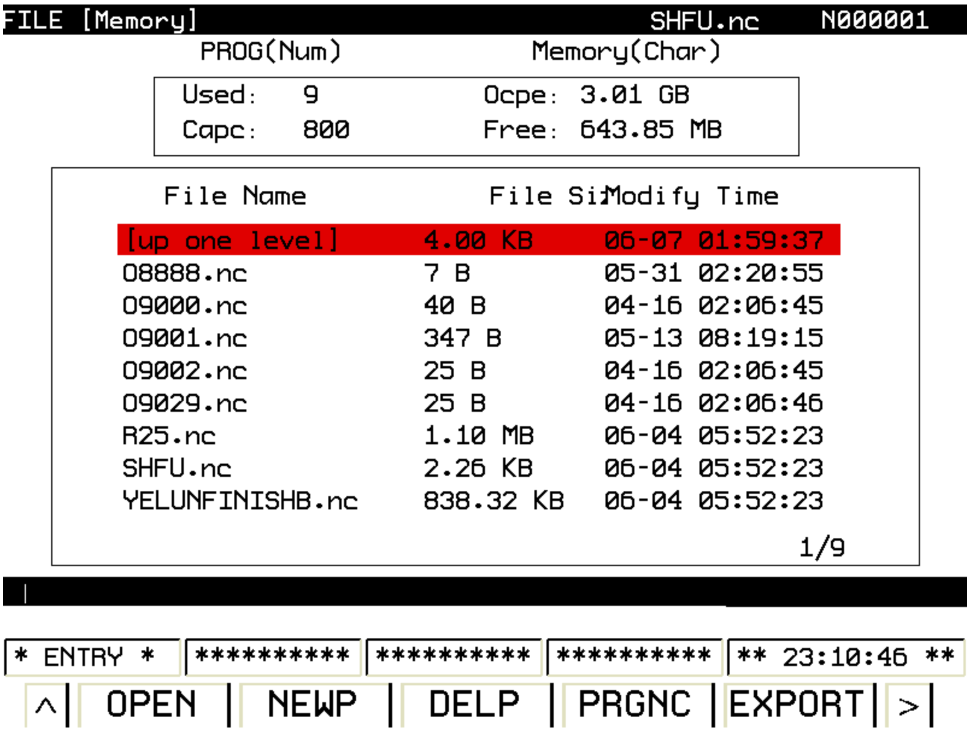
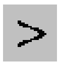


Fig. 3-16

Press 【OPT】soft key to enter file operation interface (refer to 3.2.1 for detailed introduction)

10 CNC program names can be displayed once on program contents display page. If CNC programs are more than 10, it can not be displayed completely on one page. Press page turning key on this page. LCD displays CNC program name on the next page, and all CNC program name can be displayed again by pressing page turning key repeatedly.

Press soft key 

on following fig.3-16 to enter the next interface (see Fig. 3-17):

Press 【REFL】soft key to return current uploaded program name. For example: If R25.nc is uploaded in current program edit interface, current list interface will automatically skip to the page with R25.nc by pressing 【REFL】soft key, and R25.nc is highlighted and selected.

【BK_EDT】soft key. When current code is executed in auto mode or MDI mode, 【BK_EDT】 soft key can be used to upload the code that user will edit simultaneously. In background edit mode, editing and saving altered information simultaneously are available. In operation state, press 【Background EXIT】soft key to withdraw from background edit.

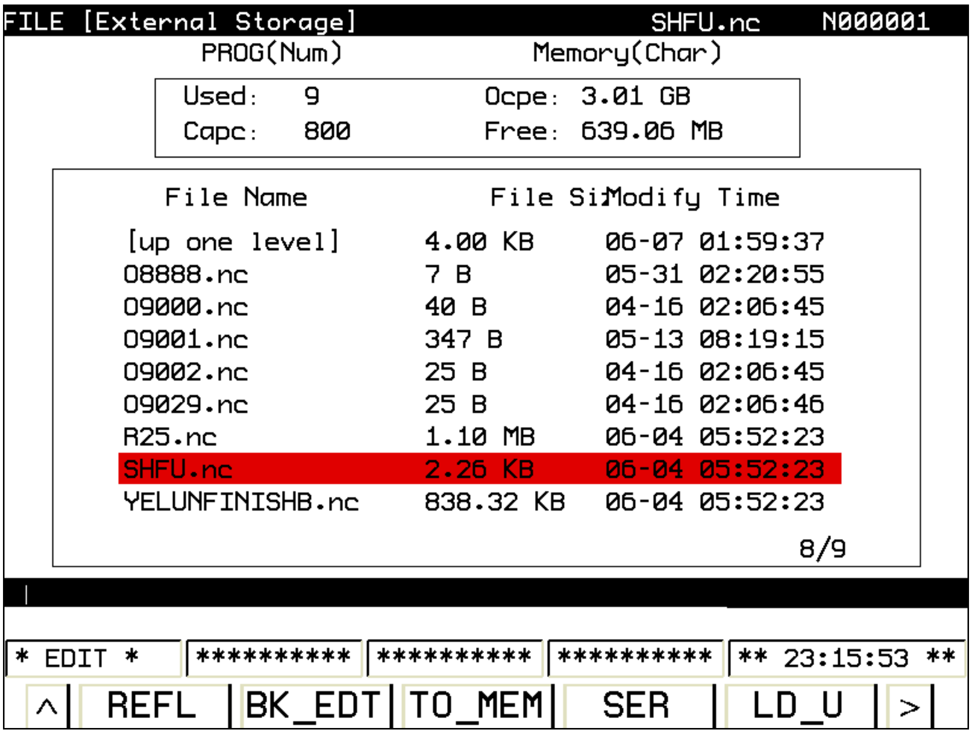


Fig. 3-17

This system supports U disk operation. Press 【LD_U】soft key after plugging in U disk, there will be prompting tips displayed in interface immediately: SD card has been successfully loaded. In current interface, press 【COPY】soft key to duplicate nc file. The duplicated program will be directly copied into internal memory to store. Press 【TO_MEM】soft key to enter internal memory, see Fig. 3-16. User can copy nc file from internal memory to U disk by 【COPY】soft key. Press 【TO_MEM】soft key to enter external memory from internal memory.

3.3 Display Setting

3.3.1 Page Setting

2. Enter the page

Enter the offset and information display setting interface by pressing 

. There are six interfaces such as **[Offset]**, **[Set]**, **[Workpiece]**, **[Macro variable]**, **[Pitch]** and **[Logging in Log-in]**, which can be checked and modified by the corresponding softkeys or each interface can be shifted by pressing 

. Refer to the following figure for details (Fig.**3-18**):

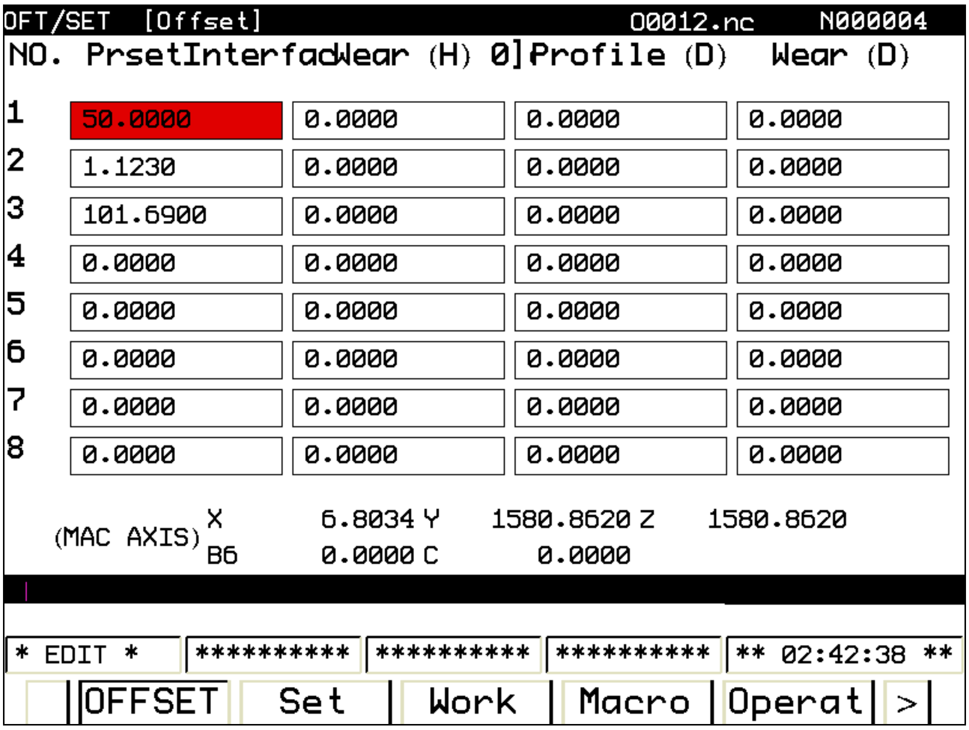


Fig. 3-18

Enter the next page by pressing【▶】.

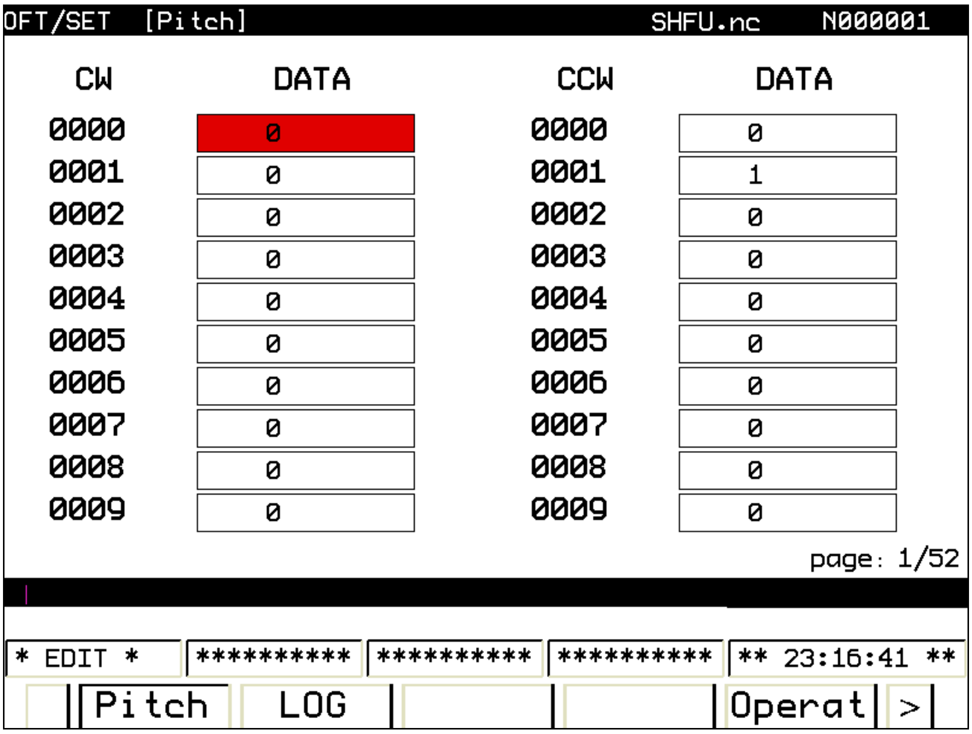


Fig. 3-19

Notice: The pitch error compensation setting can only be shifted between +7—7. If it exceeds its range, the system flashes with an alarm: Invalid data.

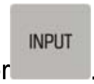
Procedure for setting and displaying the tool offset value

Refer to the figure 3-18

1) Press the function key.

2) The tool compensation screen is displayed by pressing the softkey [**Offset**]. The screen varies according to the type of tool offset memory.
3) Move the cursor to the compensation value to be set or changed using page keys and cursor keys, or enter the compensation number in this case, the compensation number can be searched by controlling the soft key **[Search].**

4) Set the compensation value. A value is input before pressing the softkey **[Input]** or 

, refer to the figure 3-20. The tool compensation automation acceleration automatic adding function can be achieved by pressing **[Input +]**. For example, D1 must be changed into 2 from 5. In this case, there are two methods can be performed: a. To write the number 2 directly before controlling the softkey **[input]** or 

. b. To write－3 firstly, and then the softkey **[Input＋]** is pressed. The softkey **[C.Input]** can be directly read a machine coordinate system of Z axis at its outline (H)(tool length compensation number), and the machine coordinate, relative coordinate and absolute coordinate can be shifted directly by the **[Coordinate shifting]** so that the user can easily check them. The machine coordinate system of Z axis can be directly read in the figure（H）which is the tool length compensation number through pressing **[C.Input].** The machine coordinate, the relative coordinate, the absolute coordinate can be directly switched through pressing **[Switching the coordinates]**, so that the user can check them in time.

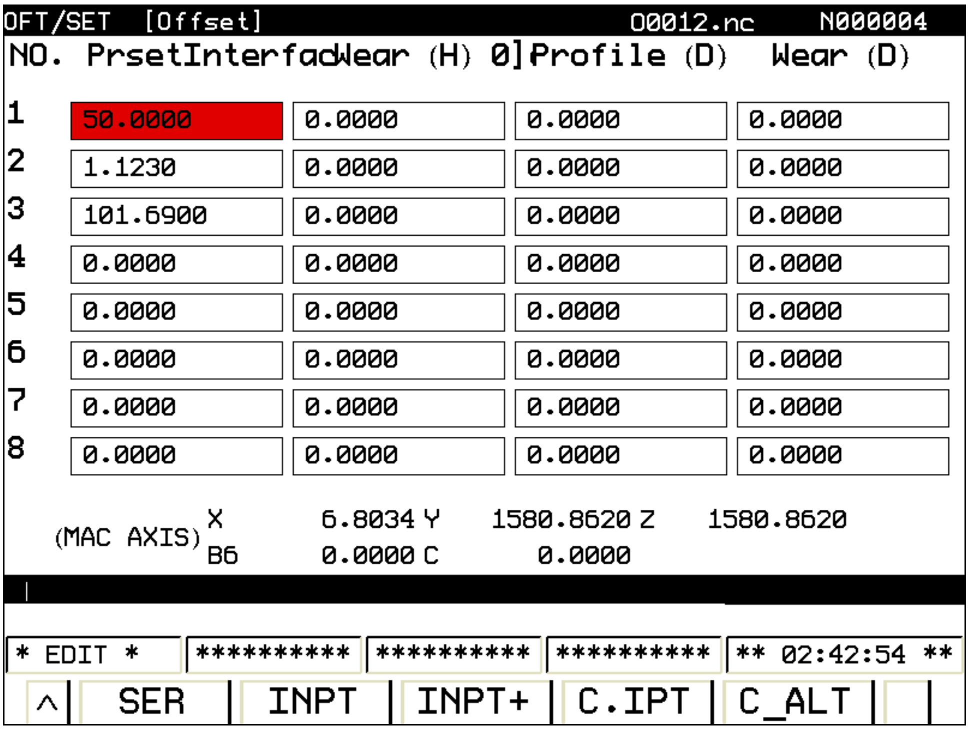
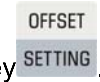


Fig. 3-20

2. The description of **[Set]** interface operation
Procedure for setting the setting data
1) Select the Edit/MDI mode.

2) Press the function key .

3) Press the softkey **[Setting]** to display the setting data screen.

4) Move the cursor to the item to be changed by pressing the cursor keys.

5) Enter a new value and press either  or the softkey **[Input]**.

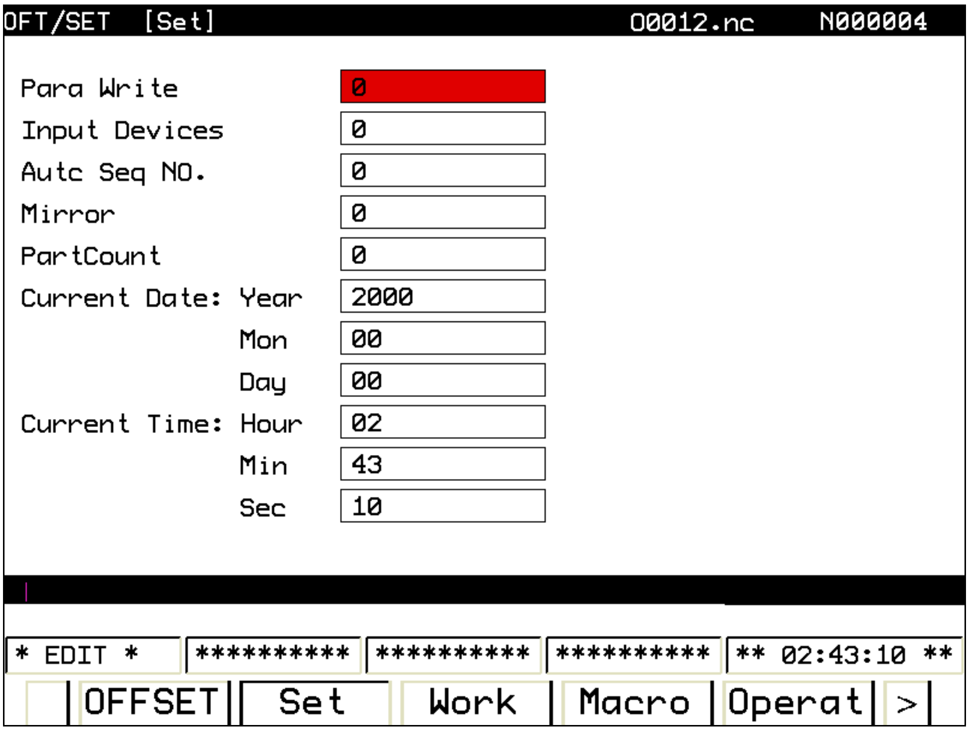


Fig. 3-21

Either 1 or 0 is input based on the following descriptions:

1) Parameter writting

0: It can not be written, 1: It can be written

2) Equipment input

The codes are selected when the data in the memorizer are input/output.

1: **ISO** code. 0: **EIA** code.

3) The sequence number of automatic accumulation

0: In the Edit mode, when the program is registered by the keyboard, the system would not being inserted the sequence number automatically.

1: In the Edit mode, when the program is registered by the keyboard, the system may insert the sequence number automatically.

4) Mirror image

1: The mirror image function is enabled.

0: The mirror image function is disabled.

5) The machining part number means that the number of the parts which are being machined currently.

All of these operations are modified, and then press the softkey **[Input]** to execute it.

3. The operating description of **[Workpiece coordinate system]** interface

Enter the workpiece coordinate system interface by pressing the softkey **[Workpiece coordinate system]**; refer to the figure 3-22:

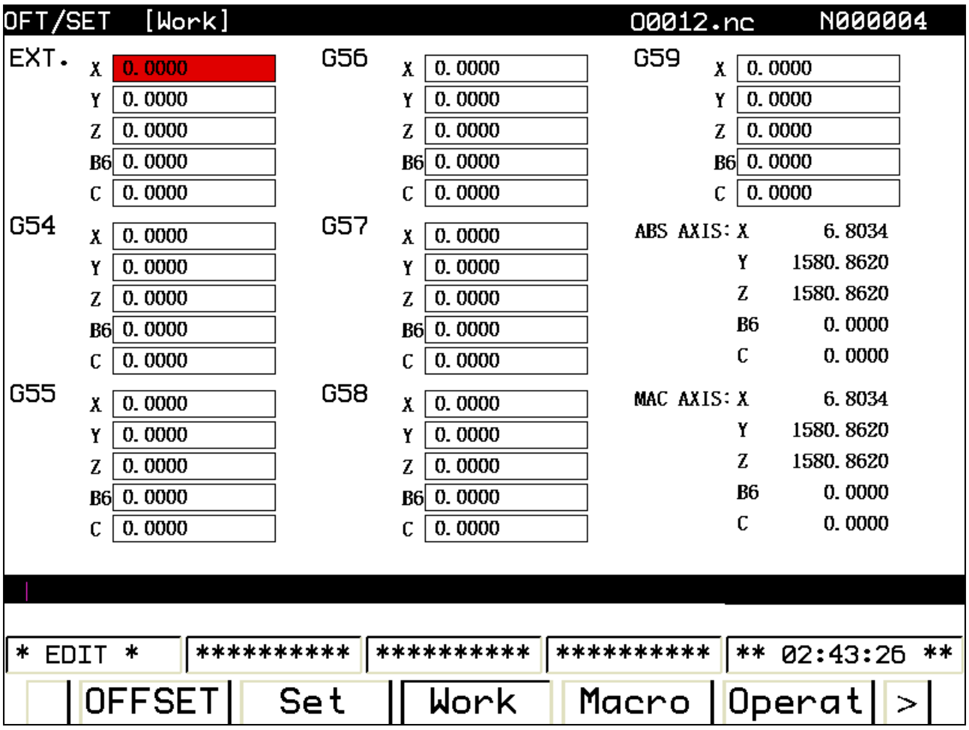


Fig. 3-22

The following operations are shown below:

(a) Enter the <MDI>/<Edit> mode;

(b) Move the cursor by the direction key (either up or down) on the item to be changed;

(c) Enter the following screen by controlling the **[Operation]** key. (Refer to the figure 3-23):

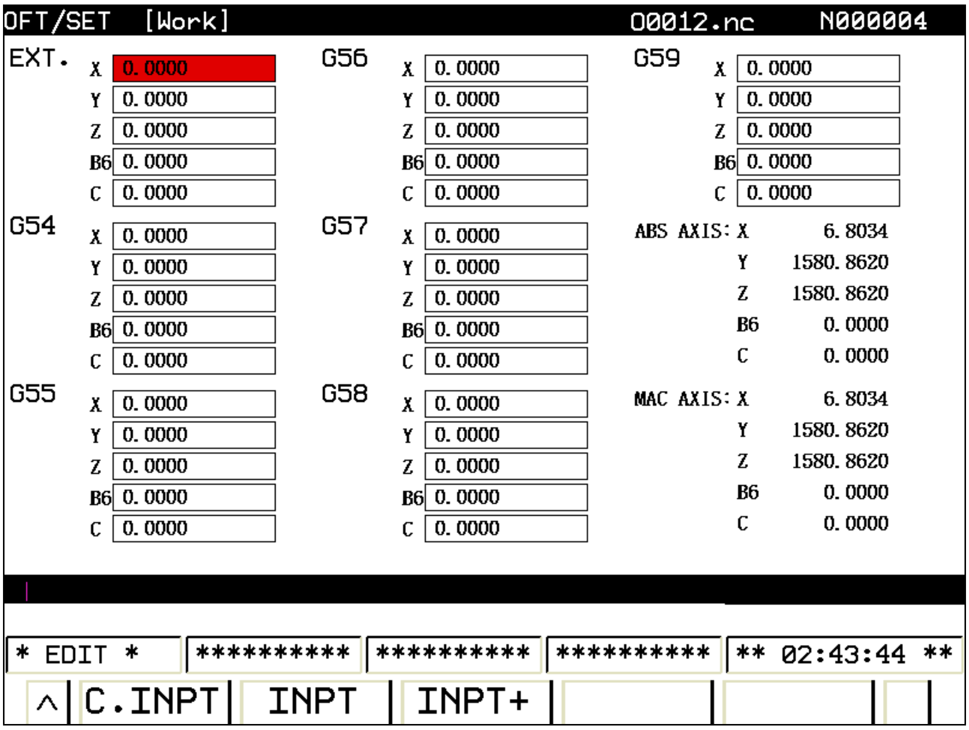


Fig. 3-23

The EXT. is a basic offset; the user can set it by **[Input]** or **[input+]** softkey like the operation of G54~G59. It is very convenient to user, which the [**C. Input**] is read into the current machine coordinate directly. The corresponding machine coordinates can be read by pressing the **[C. Input]** when the cursor is moved on the corresponding axis. Simultaneously, the absolute coordinates displayed on the interface may vary from the read machine coordinate value based on each coordinate of G54~ G59, and it is very convenient for user to operate.

4. The operating description of **[Macro variable]** interface

Enter the macro variable interface by pressing the softkey **[Macro variable]** (Refer to the figure 3-24):

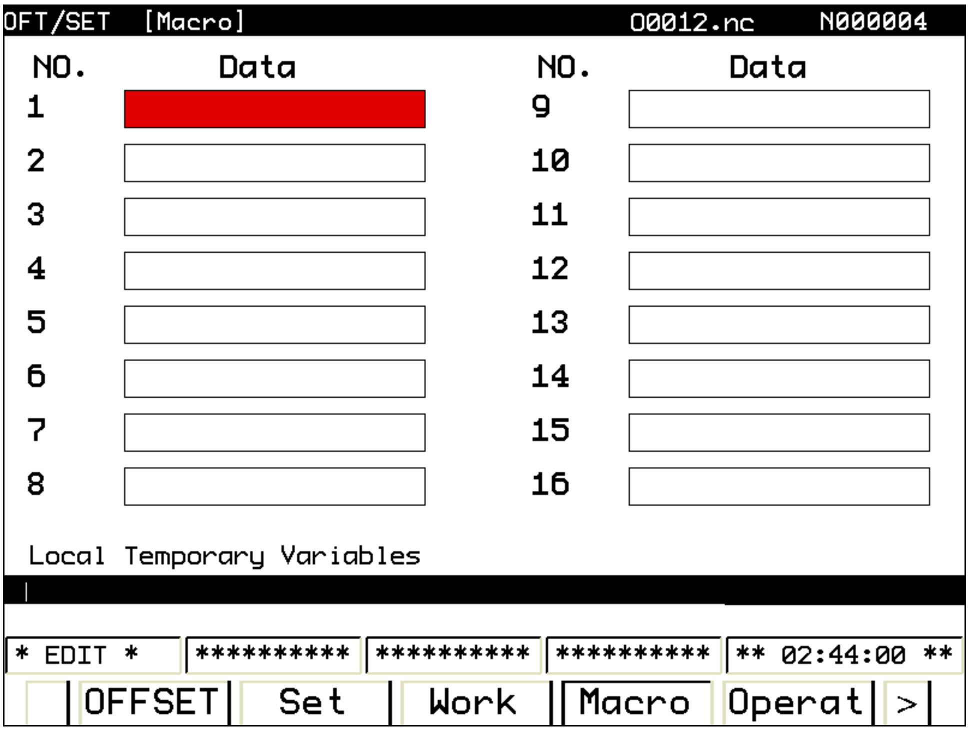


Fig. 3-24

Enter the macro variable setting interface by pressing the softkey **[Operation]**.

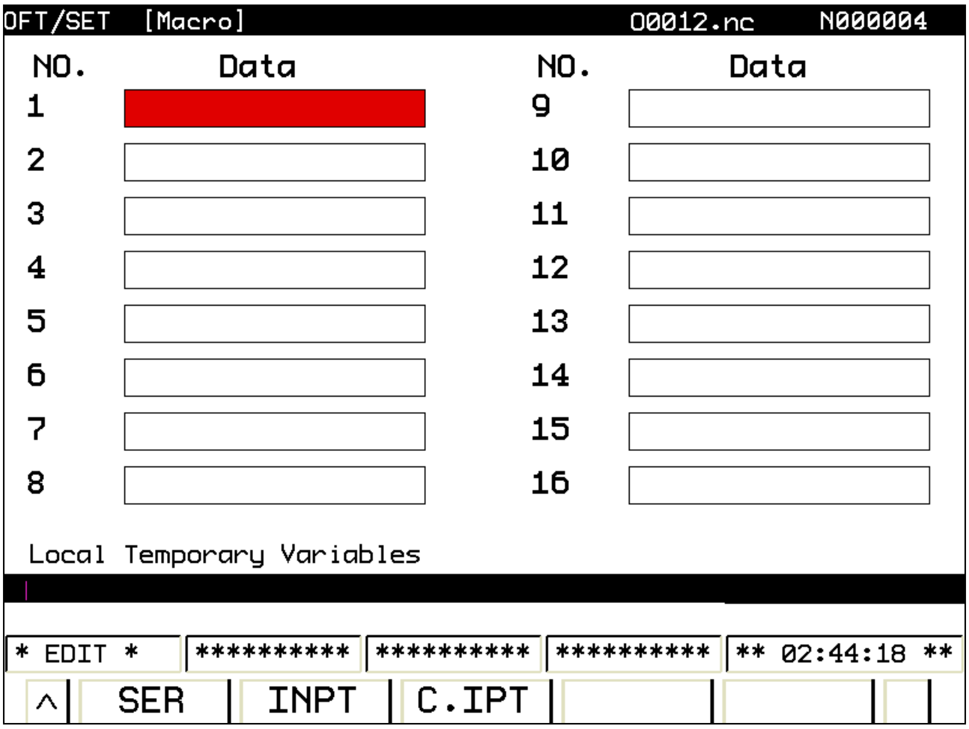


Fig. 3-25

The operations are shown below:

(a) Enter the <MDI>/<Edit> mode;

(b) Move the cursor by the page key or the direction key (either up or down) on the sequence number to be changed; or enter the variable sequence number to be modified, and then press the softkey of [Search] directly.

(c) The methods of the data modification and the machine coordinate reading on this interface are similar as the mentioned previously.

5. The operating description of **[Pitch error compensation]** interface

Enter the pitch error compensation interface by pressing the softkey **[Pitch error compensation]**, which is shown as the above figure. Both the operations of interface search and the modification can be performed on this screen by controlling the softkey of **[Operation]** (Refer to the fig. 3-26):

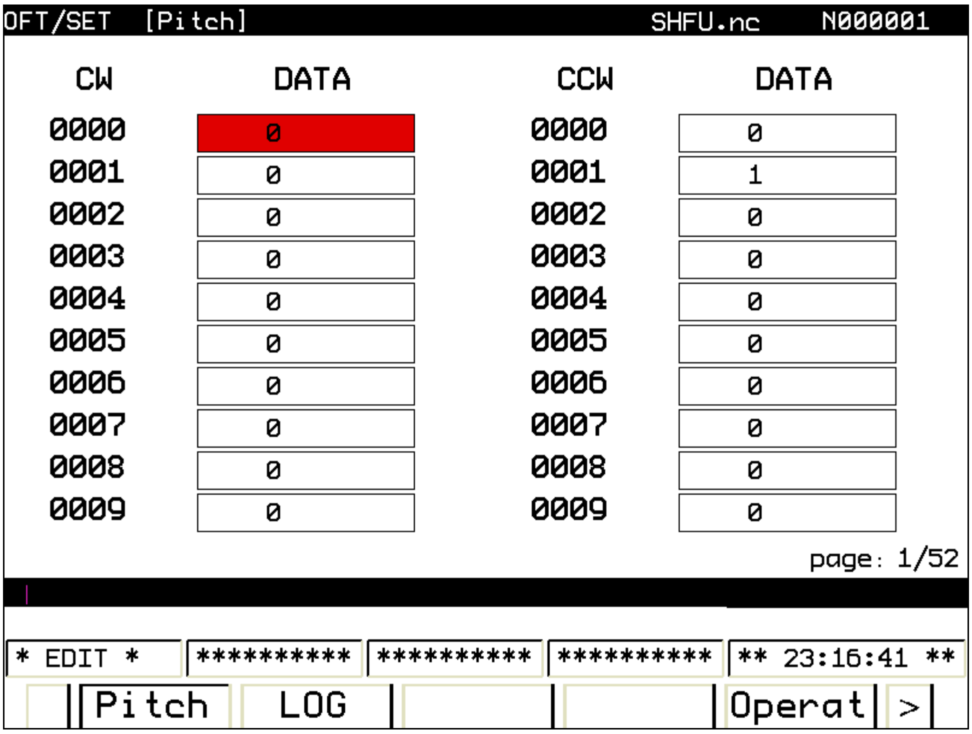


Fig. 3-26

If pitch error compensation data is specified, pitch errors of each axis can be compensated in detection unit per axis.

Pitch error compensation data is set for each compensation point at the intervals specified for each axis. The origin of compensation is the reference position to which the tool is returned.
The pitch error compensation data is set according to the characteristics of the machine connected to the CNC. The content of this data varies according to the machine model. If it is changed, the machine accuracy is reduced. In principle, the end user must not alter this data.

Pitch error compensation data can be set with external devices such as the Handy File.
Compensation data can also be written directly with the MDI panel.

The following parameters must be set for pitch error compensation. Set the pitch error compensation value for each pitch error compensation point number set by these parameters.

6. The operation of **[Log-in]** interface

To prevent the machining program or CNC parameter from being maliciously modified, **GSK25i** system offers an authority function. The password can be classified into levels, such as: the 1st level (The system manufacturer level), the 2nd level (The machine manufacturer level), the 3rd level (the high-level user) and the 4th level (the common user) based on the rank is from high to low, the system is the lowest level by default when the machine is power on. (See the figure 3-27)

The 1st and the 2nd levels: The state and data parameters, as well as the tool compensation data and PLC ladder diagram transmission of CNC are allowable to be modified.

The 3rd level: The state parameter, the data parameter and the tool compensation data of CNC are allowable to be modified.

The 4th level: the tool compensation data and the macro variable can be modified instead of CNC state parameter, the data parameter and the pitch error compensation.

The 5th level: No password level, the machine operator panel can be performed, however, the parameters, such as the tool compensation, the CNC state and the data parameters, as well as the pitch compensation data, which can not be modified.

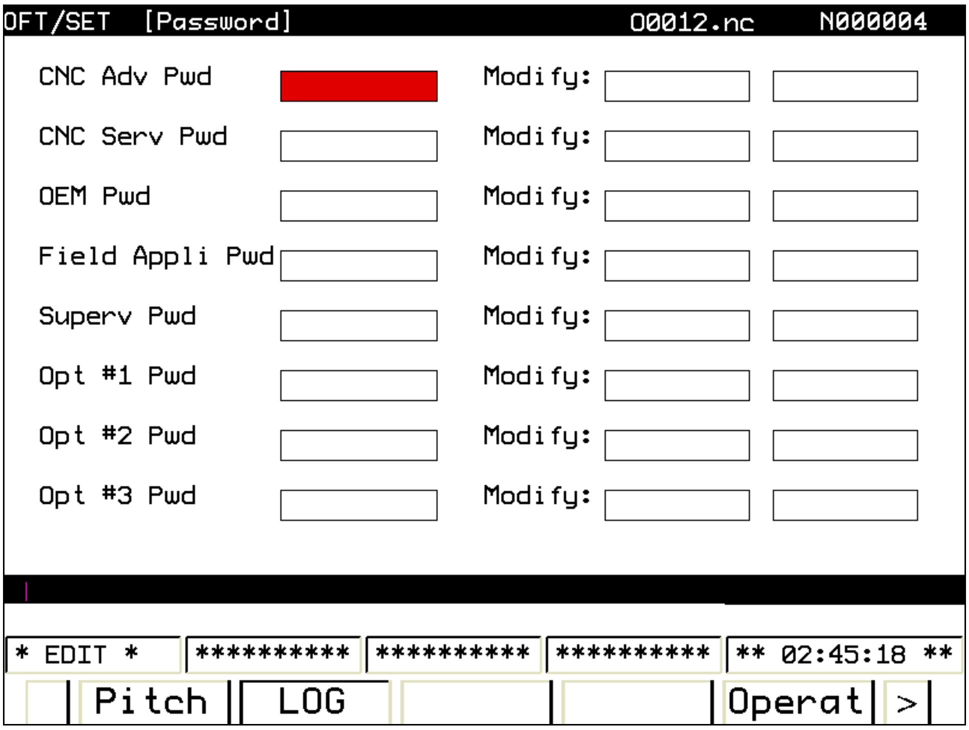
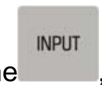


Fig. 3-27

The Logg-on interface can be modified by pressing the softkey **[Operation]**.

The modification processes are shown below:

1) After entering this interface in the mode of [MDI/Edit], move the cursor to the item to be changed.

2) The corresponding level password is input by pressing the softkey **[Input]** or the 

, if it

is correct, the system may show prompt "Correct password"; otherwise, "Incorrect password". The password is immediately cancelled and exit by pressing the **[Log-off]**.

3) Modify the corresponding parameter and setting.

4) The password is cancelled automatically after the modification is executed.

3.4 Figure Display

A tool path on a program can be drawn out on the screen, the machining process displayed on the figure can be checked by viewing the path on the screen, the displayed figure can be scaled up or down, and the figure parameter must be set before drawing a figure.

Enter the figure interface by pressing the 

, there are two display methods: **[Figure parameter]** and **[Figure]**, which can be switched by the corresponding softkeys. Refer to the figure **3-29** for details:

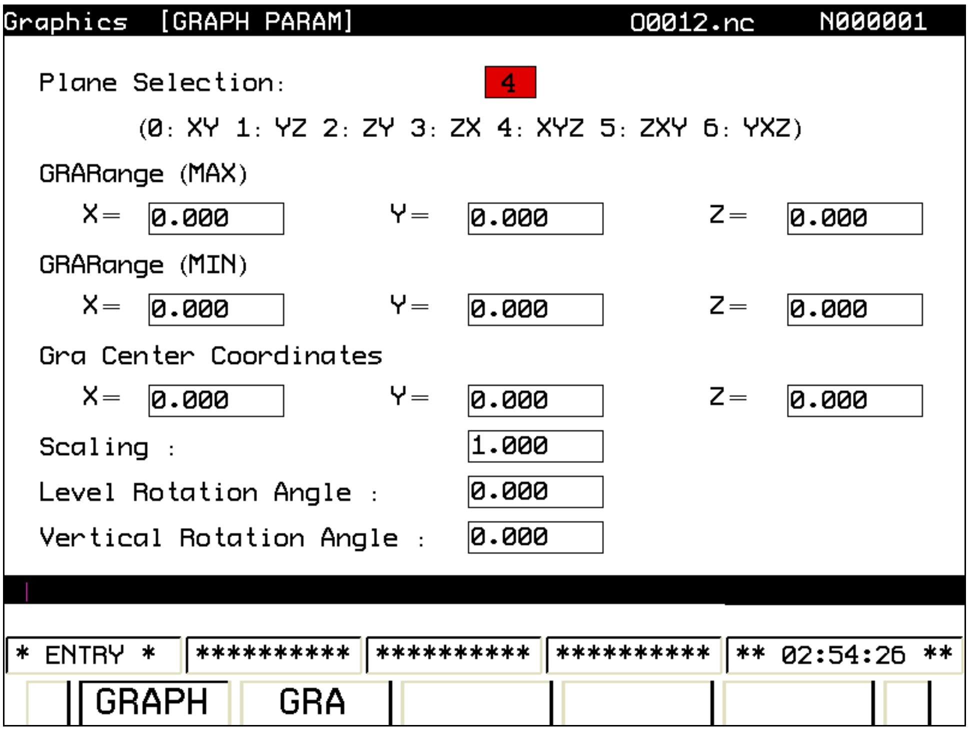


Fig. 3-29

1) The figure parameter interface

Enter a figure parameter interface by pressing the softkey of **[Figure parameter]**; refer to the figure **3-29**.

A. A signification of the figure parameter

Coordinate selection: Set a graphic plane, there are 6 methods for selecting, such as the 2nd line.

Figure mode: Set the figure display mode.

Scaling: Set the graphic proportion.

Figure center: The workpiece coordinate value corresponding to LCD center is set in the work piece coordinate system.

The maximum or minimum value: CNC system may be automatically set for a scaling and figure center value after the maximum or the minimum value of the display axis is set.

The maximum value of X: The maximum value along with X direction displayed in the figure

(Unit: 0.001mm)

The minimum value of X: The minimum value along with X direction displayed in the figure (Unit: 0.001mm)

The maximum value of Y: The maximum value along with Y direction displayed in the figure (Unit: 0.001mm)

The minimum value of Y: The minimum value along with Y direction displayed in the figure (Unit: 0.001mm)

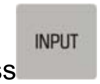
The maximum value of Z: The maximum value along with Z direction displayed in the figure (Unit: 0.001mm)

The minimum value of Z: The maximum value along with Y direction displayed in the figure (Unit: 0.001mm)

B. The setting method of the graphic parameter

a. Move the cursor to the parameter to be set;

b. Input the corresponding value in terms of the actual requirement;

c. Press 

to confirm.

d. The machine moves with automatic operation start.

Details：

● Range (the actual graphics range), the graphics screen dimension is shown below:

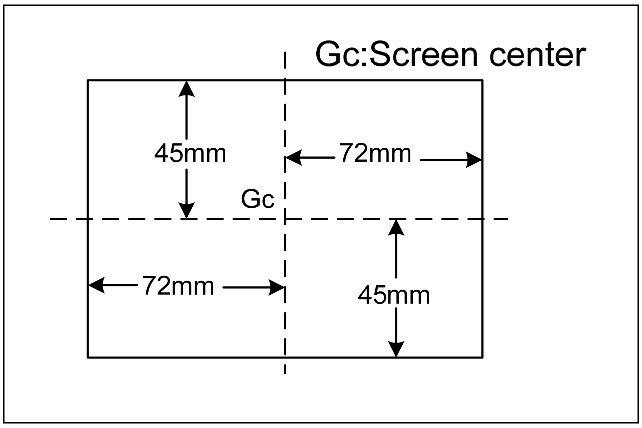


Fig. 3-30 Graphic range

The maximum graphic range value is indicated as 144mm (width) ×90mm (height), refer to the figure 3-30(a).

●To draw a section of the program within the actual graphics range, set the graphics range using one of the following two methods:

Range:

1) Set the center coordinates of the range and the magnification.

2) Set the maximum and minimum coordinates for the range in the program.

Whether 1 or 2 is used depends on which parameters are set last. A graphics range which has been set is retained when the power is turned off.

1. Setting the center coordinate of the graphics range and graphics magnification

Set the center of the graphic range to the center of the screen. If the drawing range in the program can be contained in the above actual graphics range, set the magnification to 1 (actual value set is 100)

When the drawing range is larger than the maximum graphics range or much smaller than the maximum graphics range, the amplification rate should be modified, which is usually determined as follows:

Graphic magnification = Graphics magnification (**H**), or graphics magnifications (**V**), whichever is smaller Q

Graphics magnification **H** =α/ (length along with program to horizontal)

Graphics magnification V= β/（length on program to vertical direction axis）

α:144mm

β:90mm

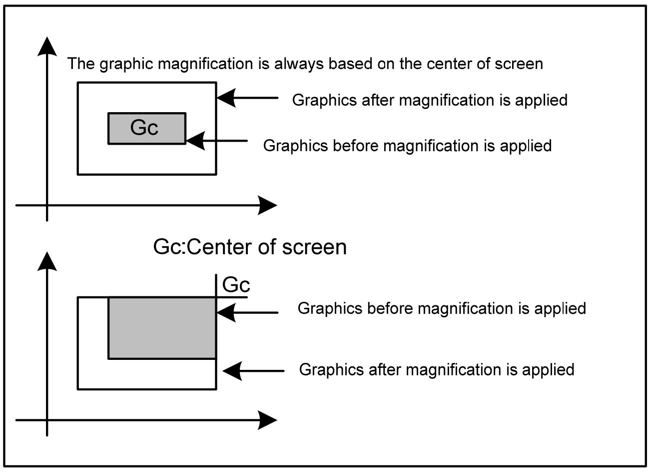


Fig. **3-30 (b)** Applying graphics magnification (Example of amplification)

The supplement of graphic scale up or down:

1) The rotation of the graph: It can be rotated by the four keys [F], [D], [H] and [B] on the operator panel. The graph with large-capacity may cause to a little slowly response.
2) The scaling of graph: The graph scaling can be controlled by the G code for which it can be controlled by the up or down key on the operator panel, too.
3) The drawing origin and the graphic center coordinate can be changed by the direction keys ⇧ ⇩ ⇨ and ⇦.

Workpiece coordinate system and graph

The drawing origin and graphic center point will not be changed even if the workpiece coordinate origin is changed. In another word, the workpiece coordinate system origin is always consistent with the graphic origin.

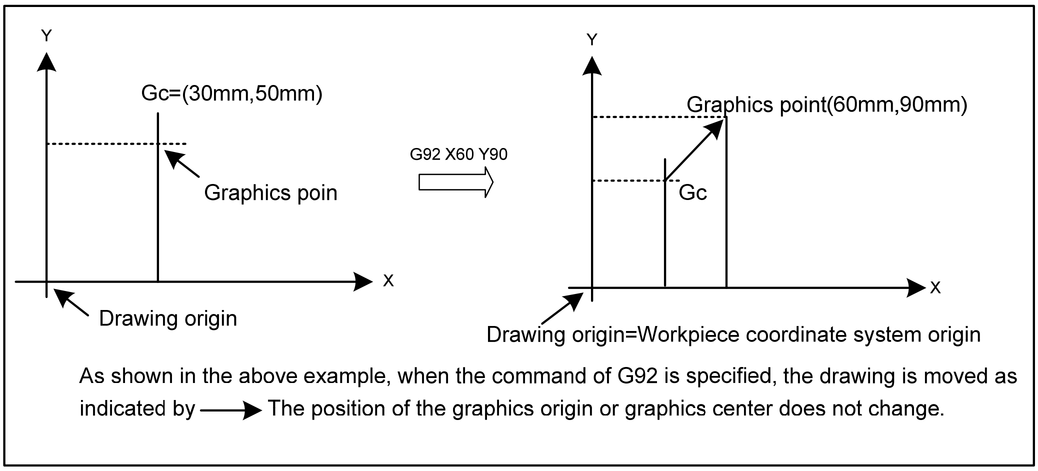


Fig. 3-30 (c) Workpiece coordinates origin and graphics origin

The valid range of graphic parameter axis is: 0 ~±9999999

Notice

1. The unit is either 0.001mm or 0.0001inch. Note that the maximum value must be greater than the minimum value for each axis.
2. When setting the graphics range with the graphics parameters for the maximum and minimum values, do not set the parameters for the magnification and screen center coordinates afterwards. Only the parameters set last are effective.

SCALE

The amplification rate of graph is set, namely, the graph parameter scaling is modified.

Graphic center

X=_

Y=_

Z=_

B=_

C=_

Set the coordinate value on the workpiece coordinate system at graphic center.

Notice:

1. When MAX. and MIN. of RANGE are set, the values will be set automatically once drawing is executed
2. When setting the graphics range with the graphics parameters for the maginfication and screen center coordinates, do not set the parameters for the maximum and minimum values afterward. Only the parameters set last are effective.

EXECUTING DRAWING ONLY

Since the graphic drawing is done when coordinate value is renewed during automatic operation, etc., it is necessary to start the program by automatic operation. To execute drawing without moving the machine, therefore, enter the machine lock state.

Deleting the drawn graph

The previous drawn graphs can be randomly deleted by pressing the softkey **[Clear]**.

2) Graph interface

Enter a graph interface by pressing the softkey **[Graph]**. (Refer to the figure 3-31):

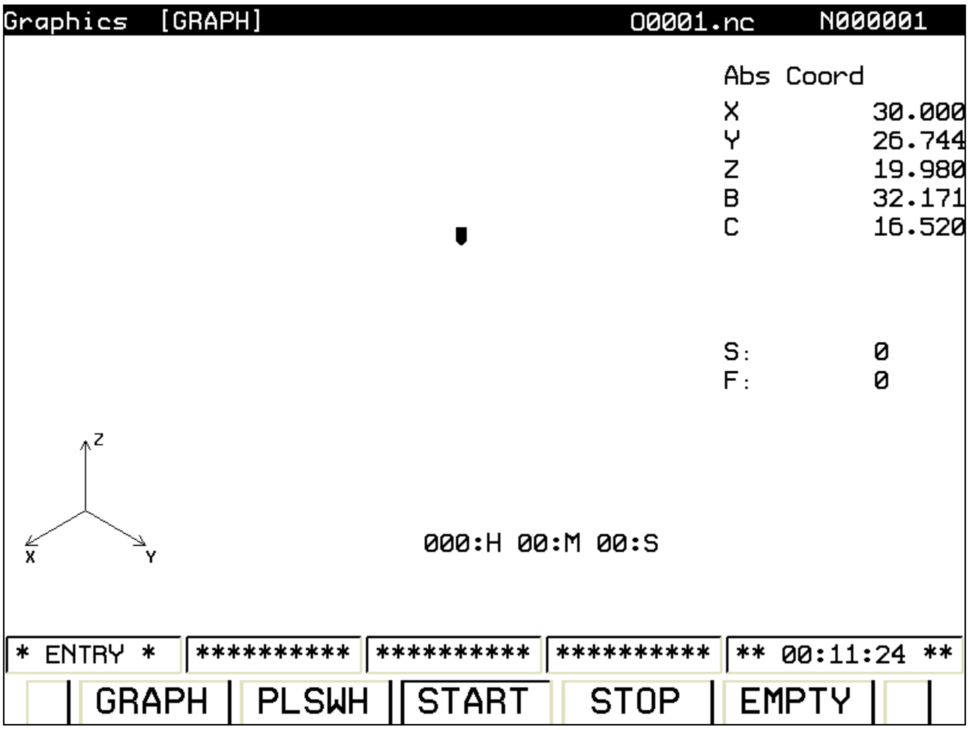


Fig. 3-31

On the page of the graph, the program machining path operation can be monitored.

A. The drawing is performed by pressing the softkey of **[Start]**, the selected stated is performed after the softkey of **[Start]** is displayed, you can view that the tool head moves to draw;

B. The **[Stop]** softkey displays a selected state by pressing the softkey ~~of~~ **[Stop]**; in this case, the drawing is stopped;

C. The graph is shifted to display on the coordinate systems corresponding to **0**～**6** when the **[Panel shift]** softkey is pressed each time.

D. The drawn graph is eliminated by pressing the softkey **[Clear]**.

E. This system has the functions of both the graph rotation and scaling: (Refer to the above-mentioned description)

3.5 Alarm Display

When the system alarms, the “alarm” information is displayed with flashing at the last line of LCD.

In this case, the alarm page is appeared by pressing the key 

, the operation softkeys such as **[Current alarm]**, **[Alarm history]**, **[Operation history]** and **[Clear]** are shown on this interface, shift or view can be performed by these corresponding softkeys (**Refer to the following graphs**).

1. Alarm interface

Check the current alarm information on the <Current alarm> interface, which is shown as figure 3-32:

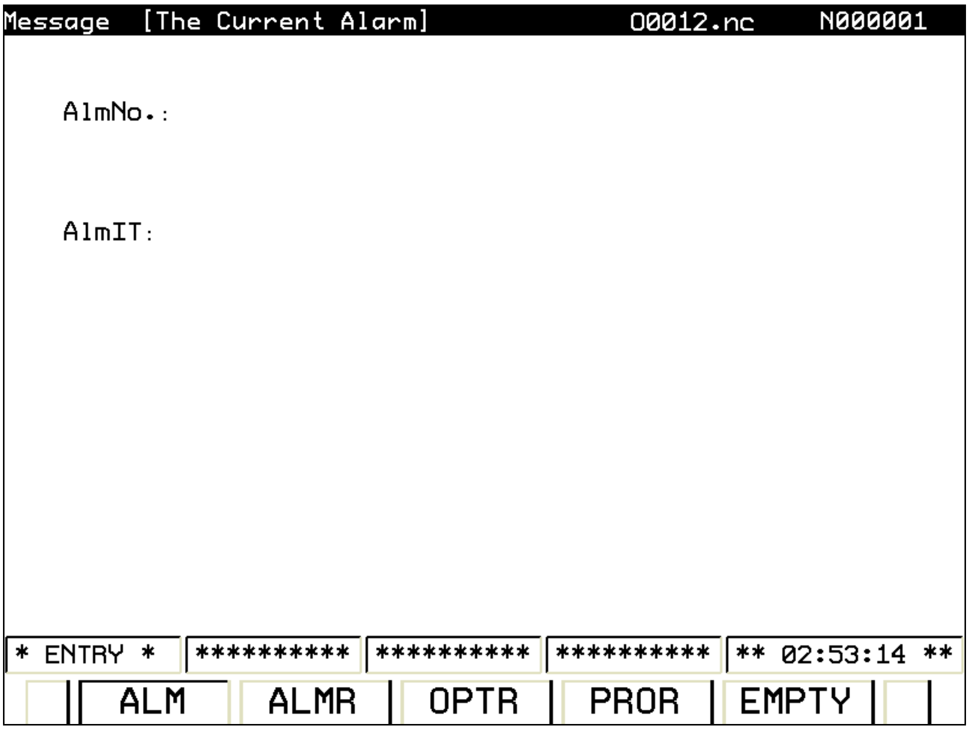


Fig. 3-32

The details of current P/S alarm number are displayed on the alarm display screen. Refer to the appendix for the alarm content.

2. **Alarm history**

Enter the alarm history interface on <alarm> interface by pressing the softkey of **[Alarm history]**. Refer to the figure 3-33:

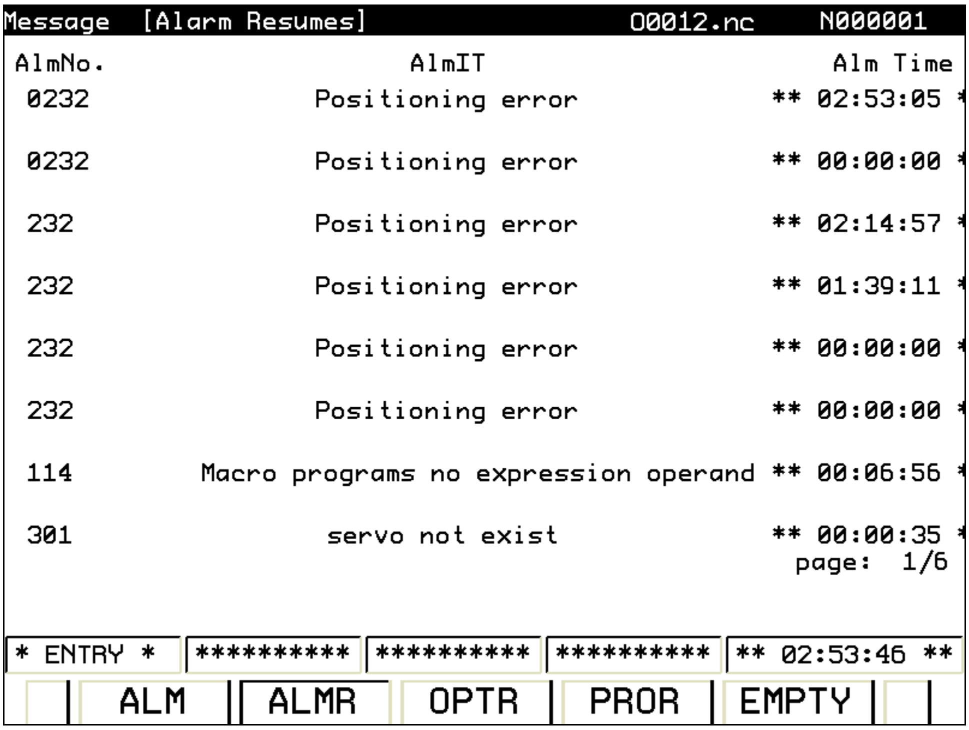


Fig. 3-33

On the interface, the alarm time is arranged from close to far so that the user can easily check it.

Procedure for Alarm History Display:

1) Press the function key.
2) Press the chapter selection soft key **[Alarm history]**.
The following information items are displayed:
a. The alarm date issues
b. Alarm No.
c. Alarm message
3) Change the page by the 1-page change key. Switch the interface by the page keys.
4) Press the [Clear] key to delete the recorded information.
3. Operation history

On the alarm interface, enter the operation history interface by pressing the softkey ~~of~~ **[Operation history]**. Refer to the figure 3-34:

The content displayed on the operation history interface is the detailed modificative information about the system parameter and ladder diagram, such as the content and time.

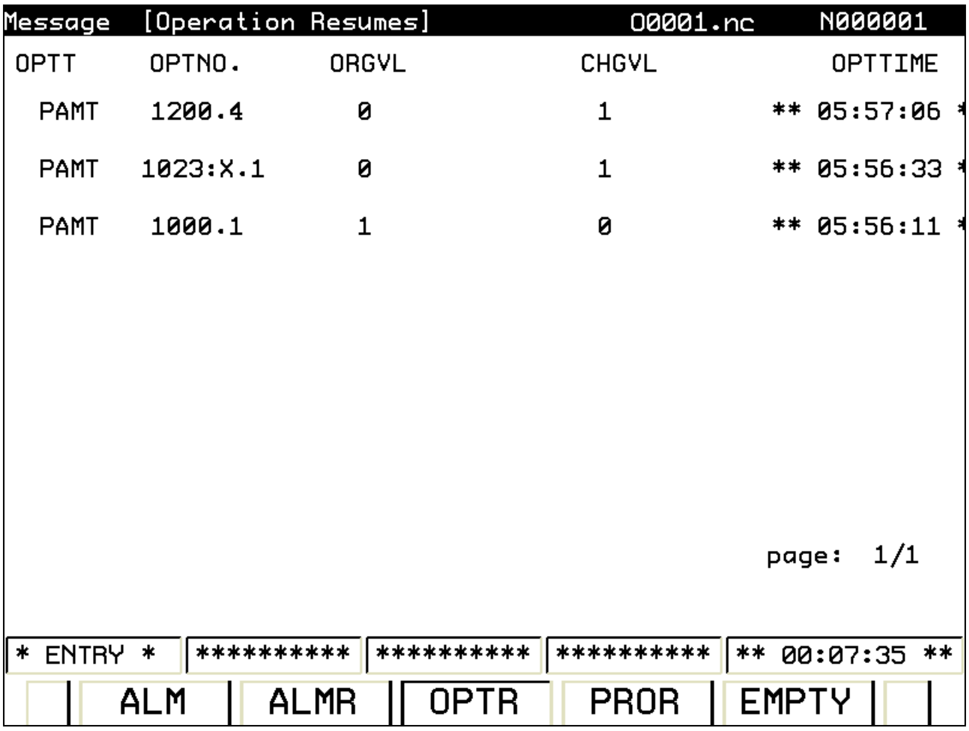


Fig. 3-34

The operation history is checked by the page keys. Can be deleted by pressing <Clear> (on debugging level or above)

3.6 System Interface Display

3.6.1 System Interface Display

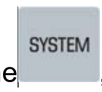
Enter display screen by pressing the, four display methods are available, **[Parameter]**, **[Diagnosis]**, **[PLC]** and **[System]** which are shifted by the corresponding softkeys. Enter each operation interface by the **[Operation]** key. Refer to the following content for details (Fig. **3-35**～

Fig **.3-38**).

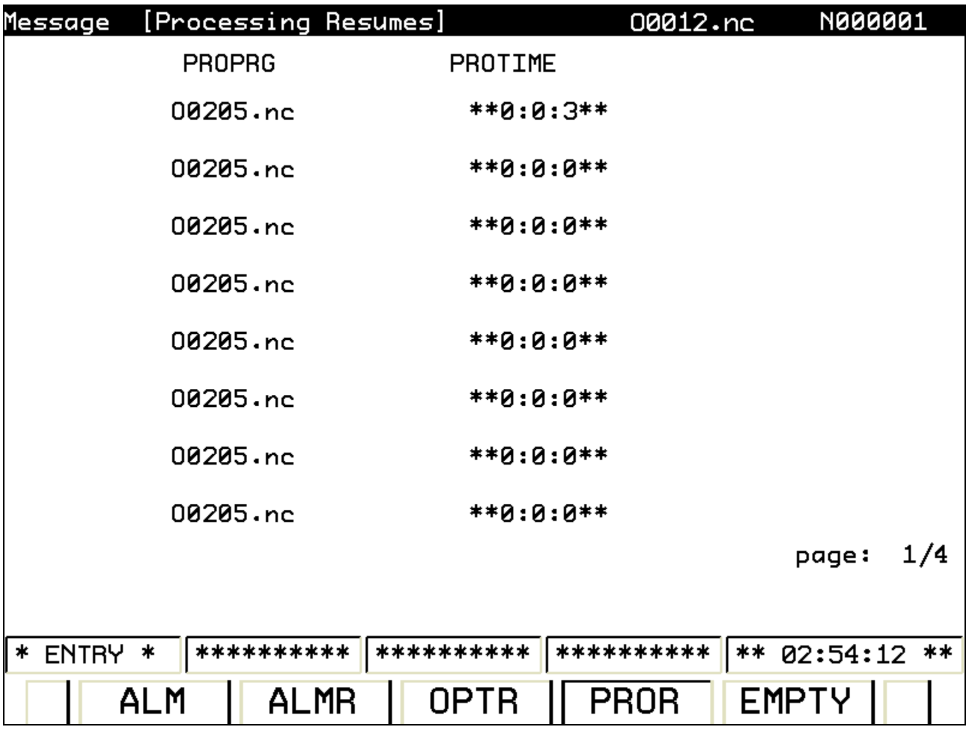


Fig. 3-35

1. [Parameter]

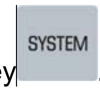
Press the **[Parameter]** key, then the **[Operation]** key to enter the parameter setting interface, five keys are available: **[All parameters]**, **[Spindle parameter]**, **[Servo parameter]**, **[Input]** and **[Search]**.

Procedure for displaying and setting parameters:

1) Firstly, press the 

on edit panel to enter the **[Log-in]** setting interface, then input the corresponding password.

2) Enter the set interface by pressing the **[Set]** softkey, the **[Parameter write]** in this interface is set to 1 so that the parameter can be written.

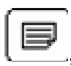
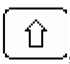
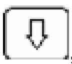
3) Enter the system interface by pressing the function key 

.

4) The parameter interface is displayed by pressing the softkey of **[Parameter]**; refer to the figure 3-27.

5) Move the cursor to the parameter number to be set or displayed in either of the following ways.

a. Enter the parameter number and press soft key **[Search]**.

b. Move the cursor to the parameter number using the page keys  and , and the direction keys , ,  and .

6) Input a numerical value by digit keys or **[Input]** key.

7) Set 0 for **[Parameter write]** to disable writing by pressing the soft key of **[Set]**, so that the parameter write state is stopped and can not be written.

Checking and modification can be performed by corresponding softkeys, which are as follows:

1) All parameters page

Enter the parameter page by pressing the softkey of **[All parameters]** (Refer to the figure 3-36)

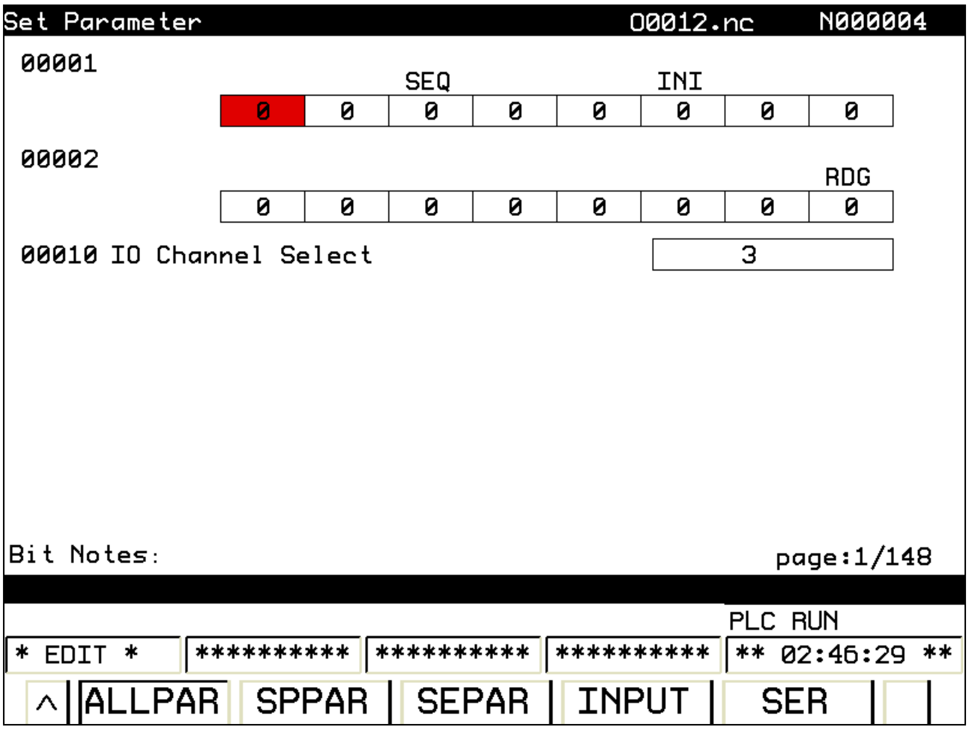


Fig. 3-36

2) Spindle parameter page

Enter the spindle parameter interface by pressing the softkey of **[Spindle parameter]**. (Refer to the figure 3-27)

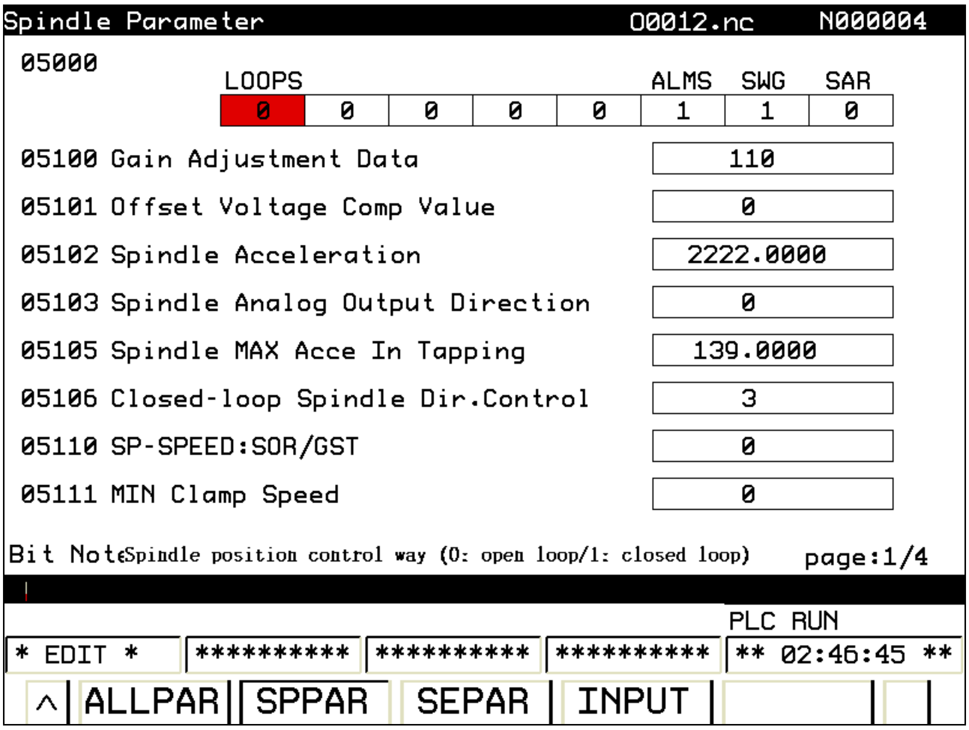


Fig. 3-37

3) Servo parameter page

Enter the servo parameter interface by pressing the softkey of **[Servo parameter]**. (Refer to the figure 3-38):

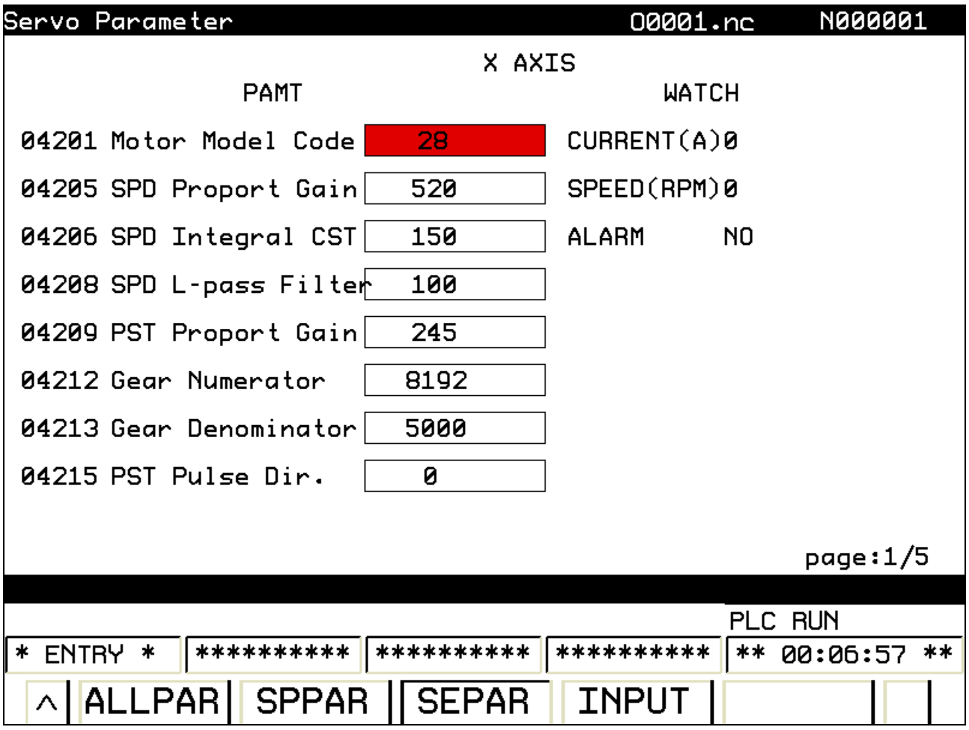


Fig. 3-38

Note: Refer to the 3rd fascicule *PARAMETER* of the manual for the definition of each parameter

2 [Diagnosis]

Press the softkey **[Diagnosis]**, and then the **[Operation]** key to enter a diagnosis display interface.

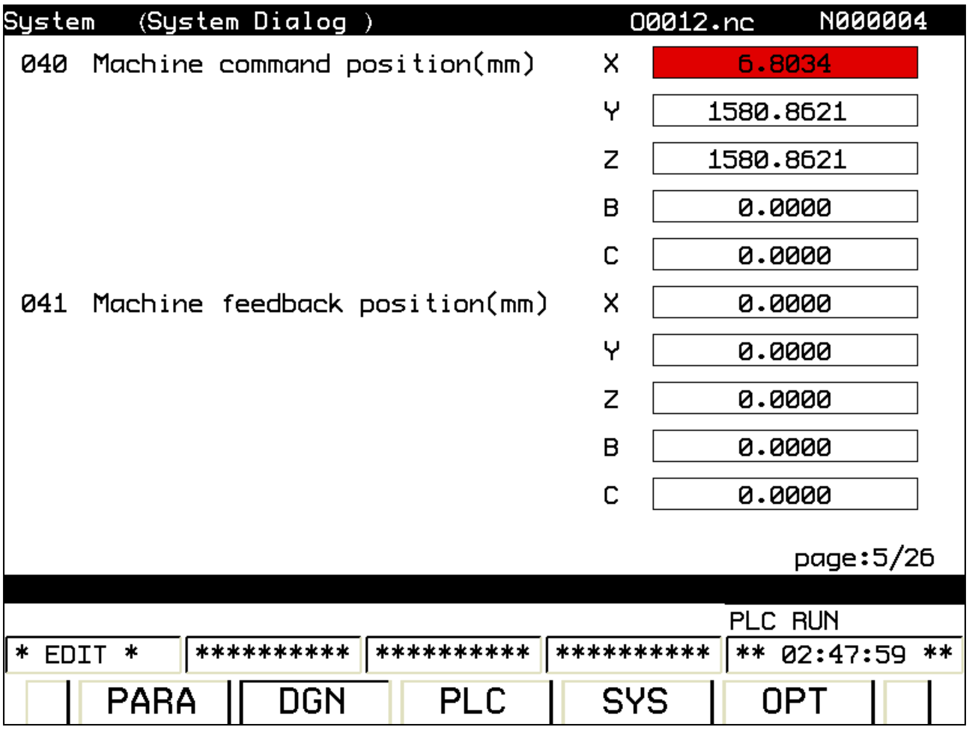


Fig. 3-39

Refer to the whole diagnoses by the upward and downward page keys. Search each number by pressing the softkey of **[Operation]**.

3【PLC】

Enter PLC operation interface by pressing the **[PLC]**. The softkeys, such as **[Integrated display]**, **[PLC diagnosis]**, **[PLC parameter]**, **[File list]** and **[Operation]** are available, wherein, the **[Operation]** is performed for another interface. Enter next interface by controlling the softkey **[>]**, which includes four softkeys: **[Set]**, **[Edit]**, **[Stop]** and **[Operation]**. (Note: The **[Stop]** option does not performed by pressing **[>]** based upon both on the **[PLC parameter]** and **[PLC diagnosis]** interfaces) The details are as follows:

1) Integrated display interface

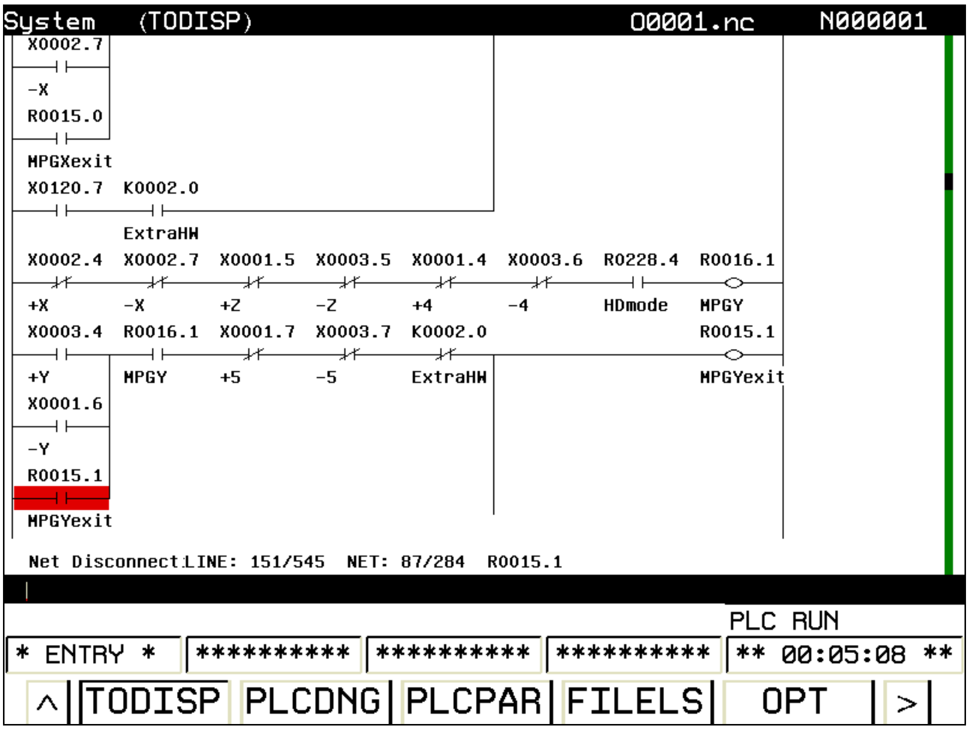


Fig. 3-40

Enter the following screen by pressing the **[Operation]** in the mode of **[Edit]**, the PLC program can be modified or edited.

Procedure of operation:

a. The displayed content can be set by the **[Set]** interface. (Refer to 3-41)

The integrated display of the ladder diagram can be controlled by the cursor, for example the component name display: move the cursor to the address where it may turn into red, which means that it is selected, the address displays in figure 3-32 (X0008.4 etc.). The component note display is similar as that of the above component name; for example, the (EEEEEEE) displayed on X0008.4 is a note for this element component. Network line note is at the end of each line at the right side.

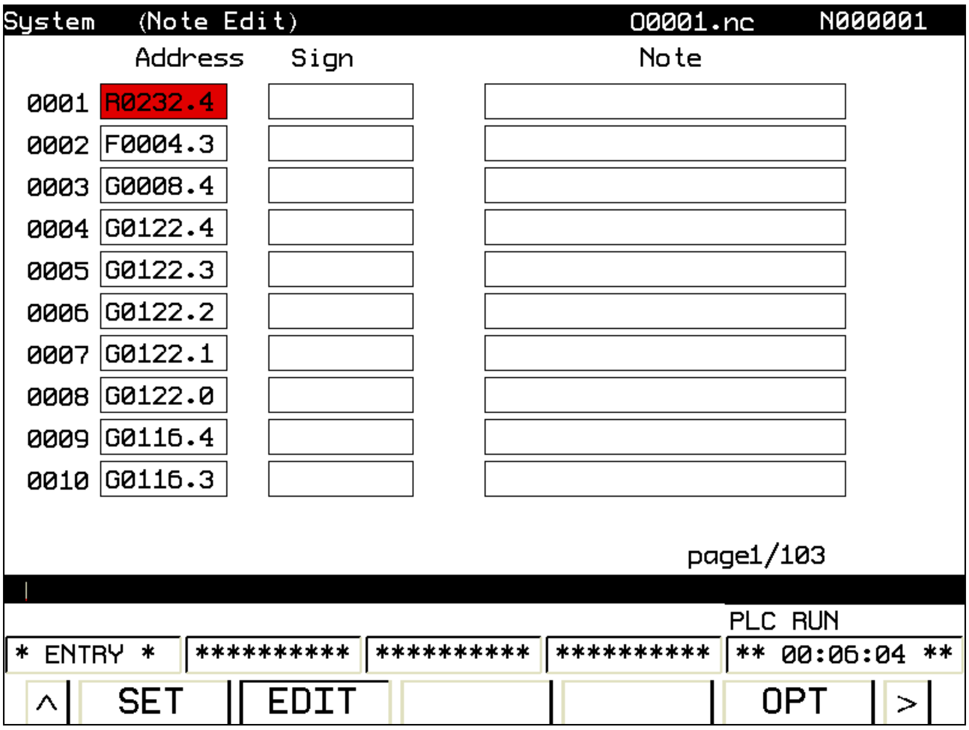


Fig. 3-41

a. The network line and component notes can be changed by controlling the softkey [Edit], and the note can be deleted and the line can be searched.
b. After the display format is set, select the **[Integrated display]**, and then the **[Operation]** to enter an editing and modification interface of the ladder diagram. (Refer to the figure 3-42)

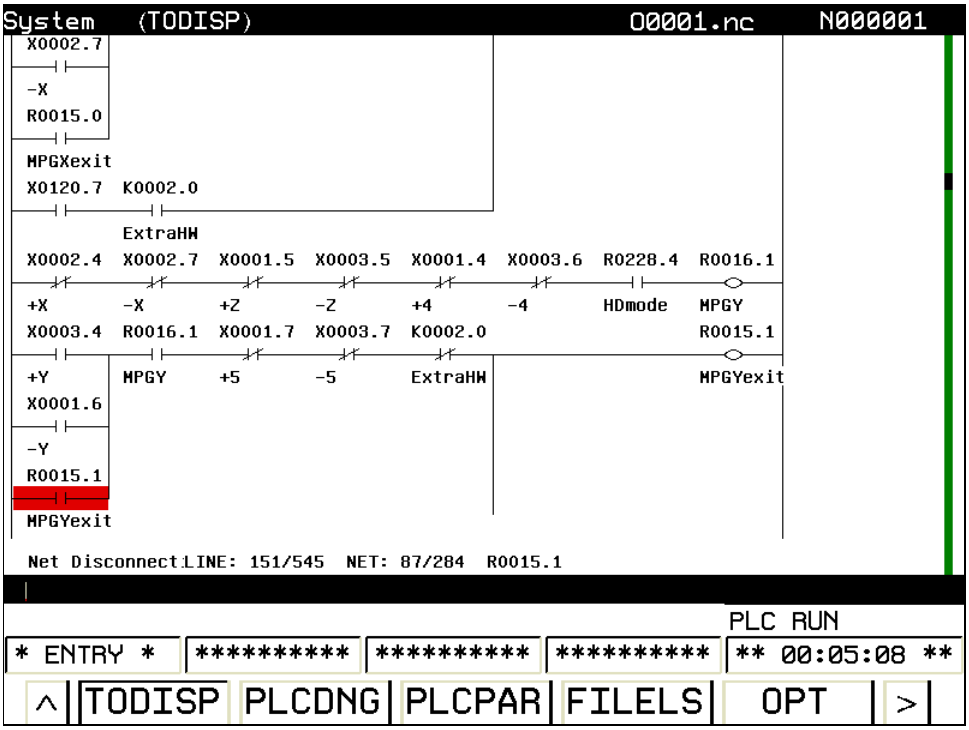


Fig. 3-42

The integrated edit of PLC can be performed in the case of the allowable operation authority, for example, the functions of the selection, copying, cutting or and deletion, and it is basically similar as the program interface editing function.

2) **[PLC diagnosis]** interface

The signal states are all displayed on the diagnosis interface, such as the signal state between **CNC** and **DI/DO** of machines, **CNC** and **PLC**, and **PLC** internal data and **CNC** internal state. This diagnosis is used for checking the **CNC** interface signal and internal signal operation state, which can not be modified.

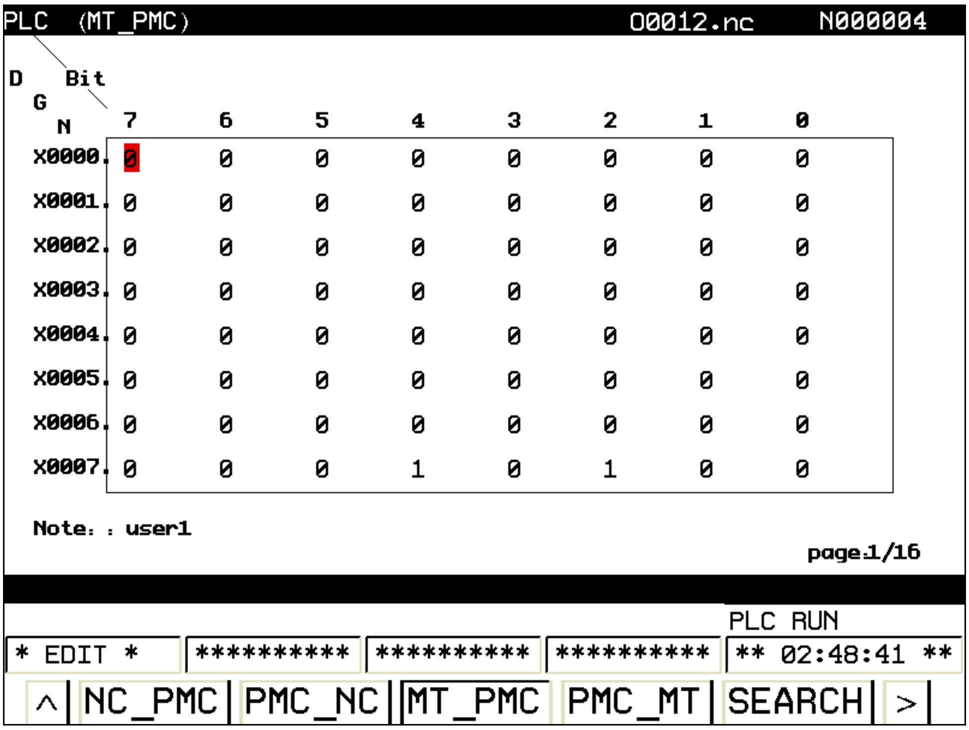


Fig. 3-43

Check each parameter by the page key. Enter the following interface by the softkey **[Operation]**:

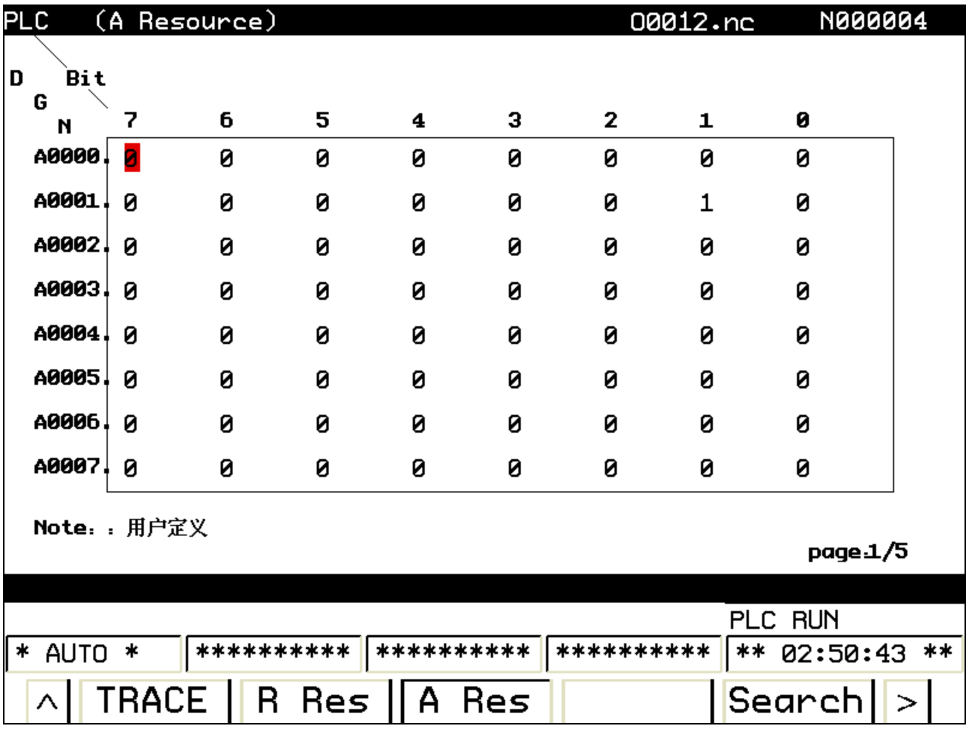


Fig. 3-44

Several corresponding interfaces are available: **[F resource]**, **[G resource]**, **[X resource]**, **[Y resource]** and **[TRACE]**, refer to the matched manual of *the 3rd fascicule of* ***GSK25i*** *CNC system:* ***PLC*** *and installation connection* for the significance of each diagnosis number and setting method. A signal trace is shown on figure 3-45.

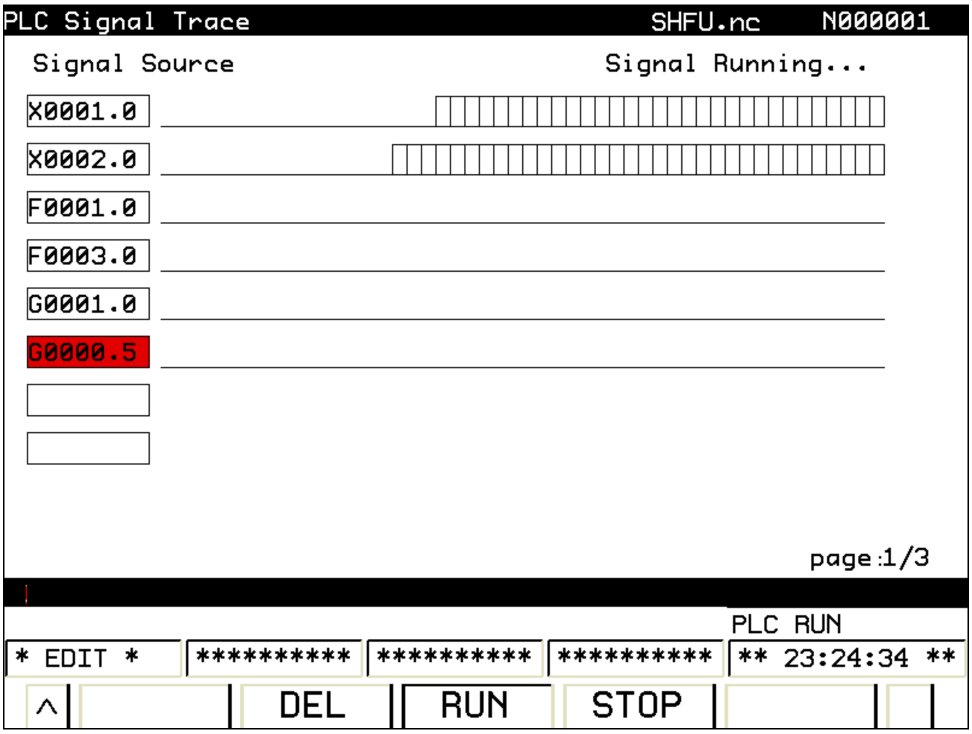


Fig. 3-45

The operations are as follows:

A. Enter PLC signal trace interface: press the softkey **[TRACE]**

B. The signal address is input regardless of any operation mode, for example, X0001.0. is input as above-mentioned figure.

C. Press the softkey **[Operation]**, when this signal is performed, the figure 3-45 frame may occur. If no signal is transmitted, a straight line is displayed in this case.

3) **[PLC parameter]** interface

Press **[PLC parameter]** to enter PLC parameter setting interface:

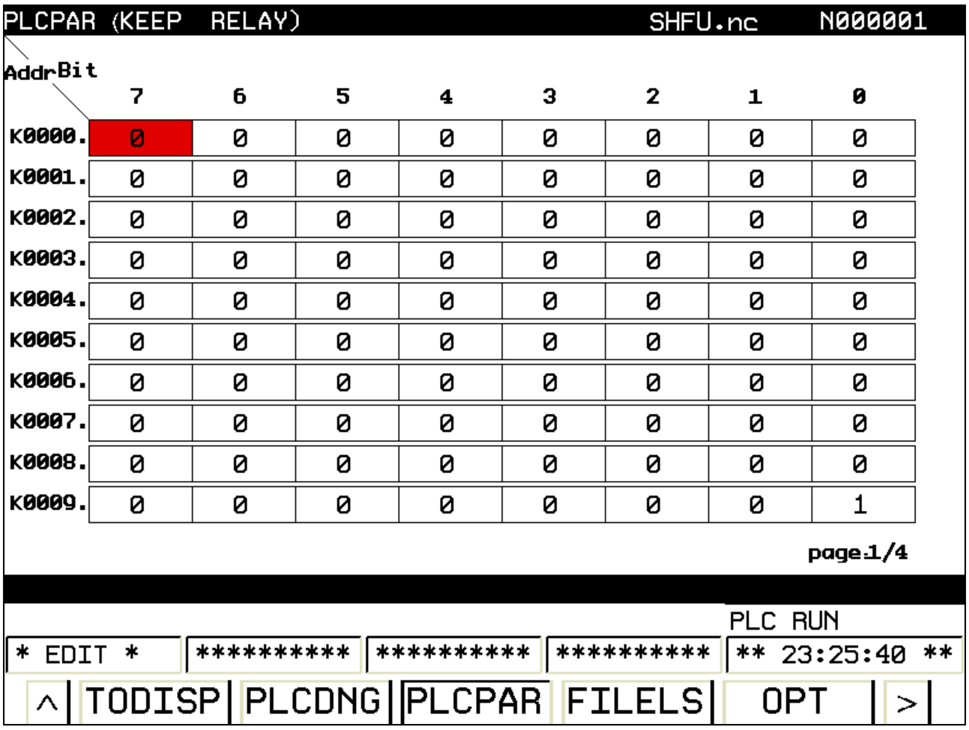


Fig. 3-46

Press the softkey **[Operation]** to enter a detailed parameter modification interface:

Relay:

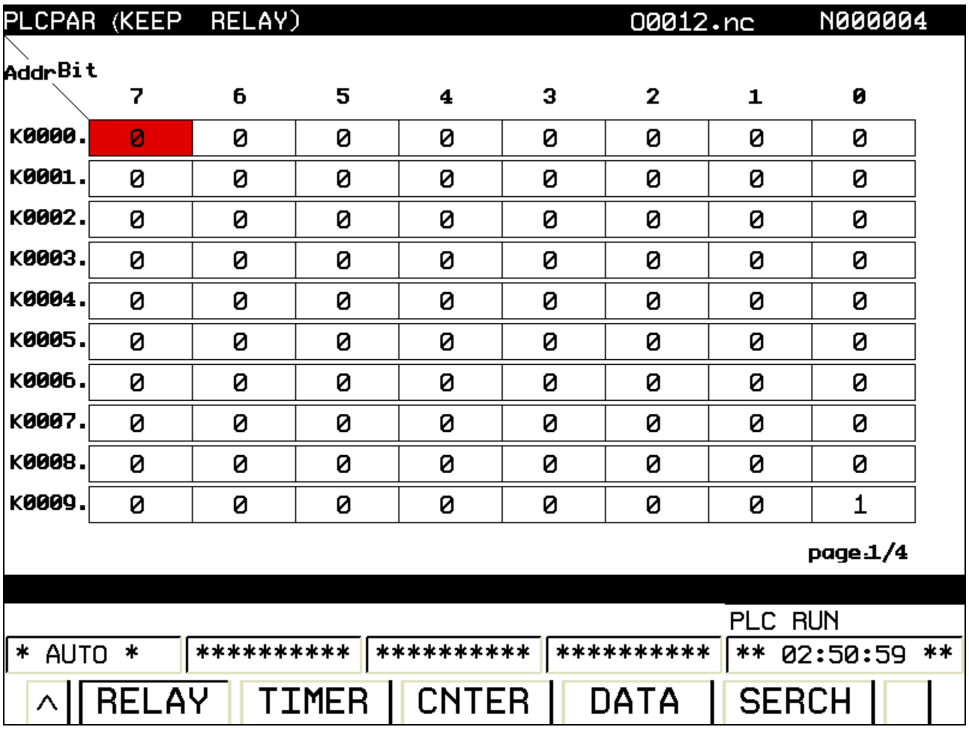


Fig. 3-47

Timer

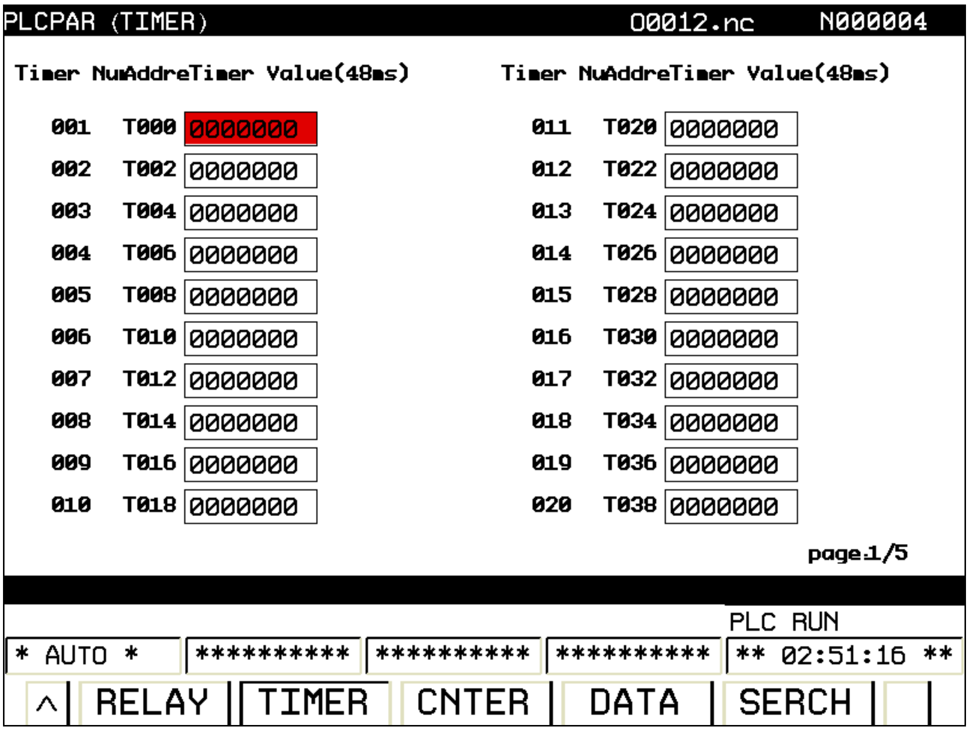


Fig. 3-48

Counter

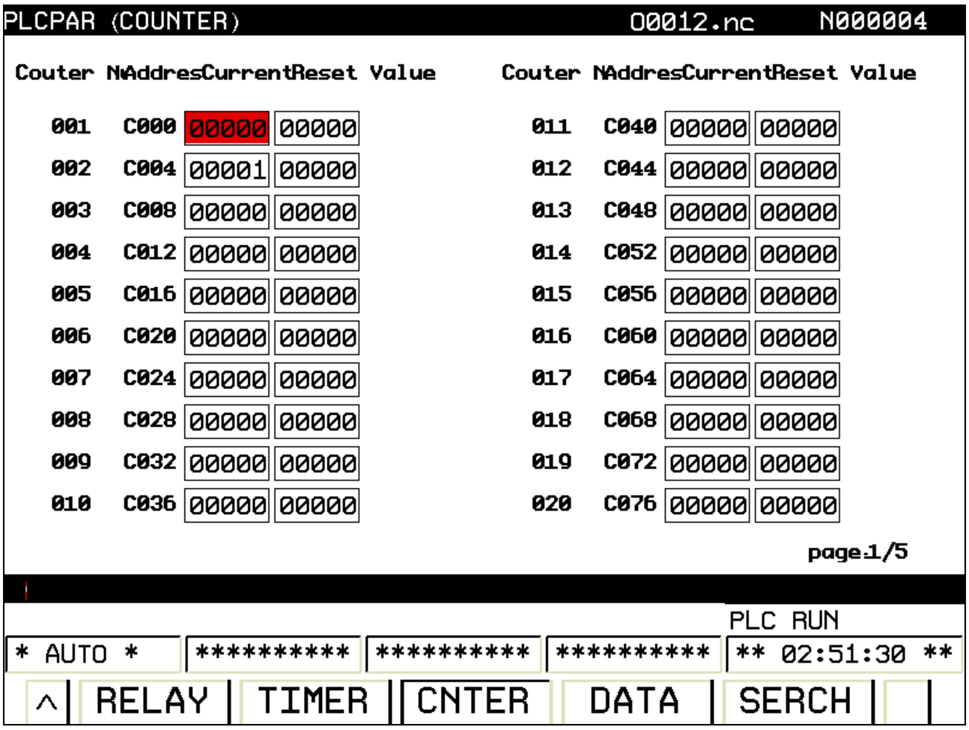


Fig. 3-49

Data

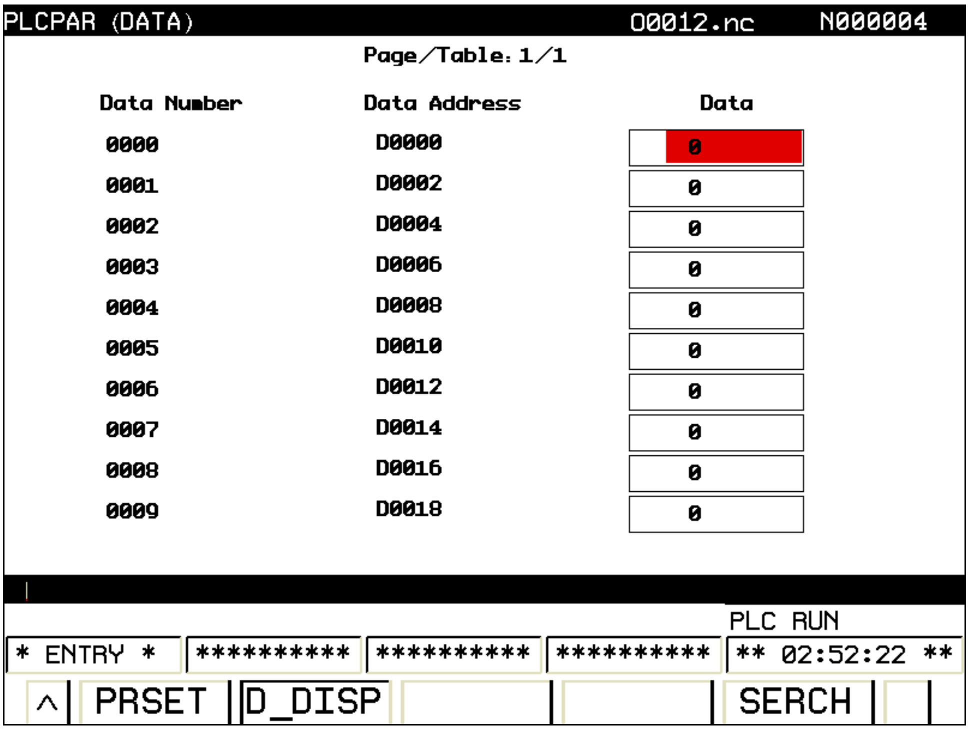


Fig. 3-50

4) **[File list]** interface

Enter PLC file list operation interface by pressing the softkey of **[File list]**, this interface includes:

(a) The stored program number: it includes subprogram.

Remainder: the program number to be registered.

(b) The spent storage capacity: the stored program occupies the storage capacity (it indicates by characters)

Remainder: The unoccupied storage capacity.

(c) The list of existing file name and file size

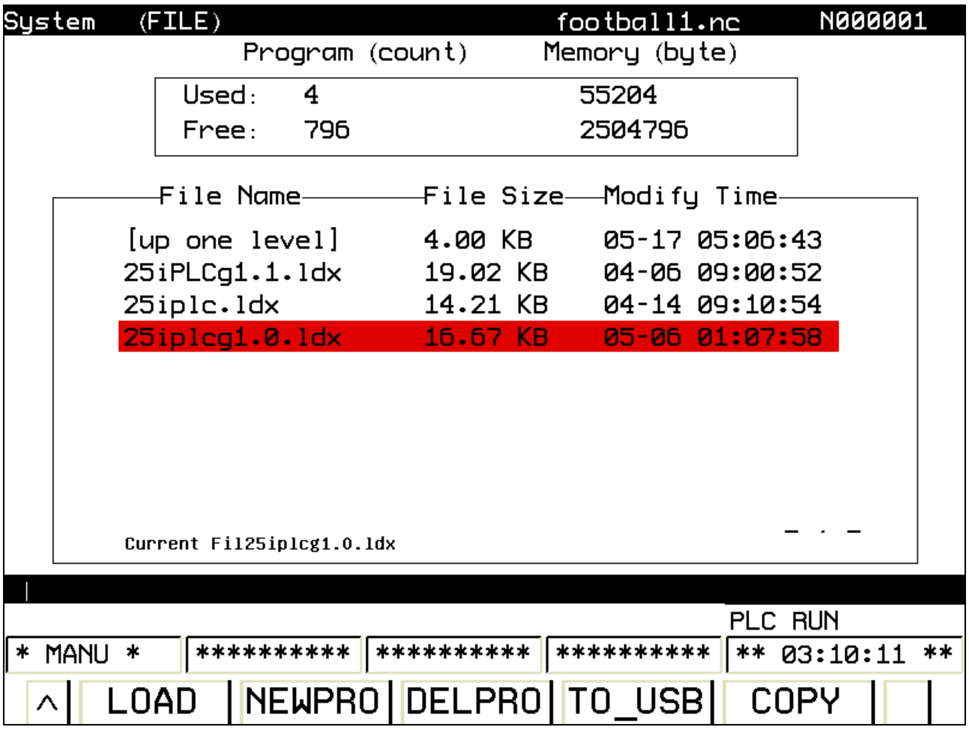


Fig. 3-51

Press the softkey **[Operation]** to change and operate the memory program which is similar as the corresponding operation of the file list interface within the program function.

4. **[System]** interface

1) **[System structure]**

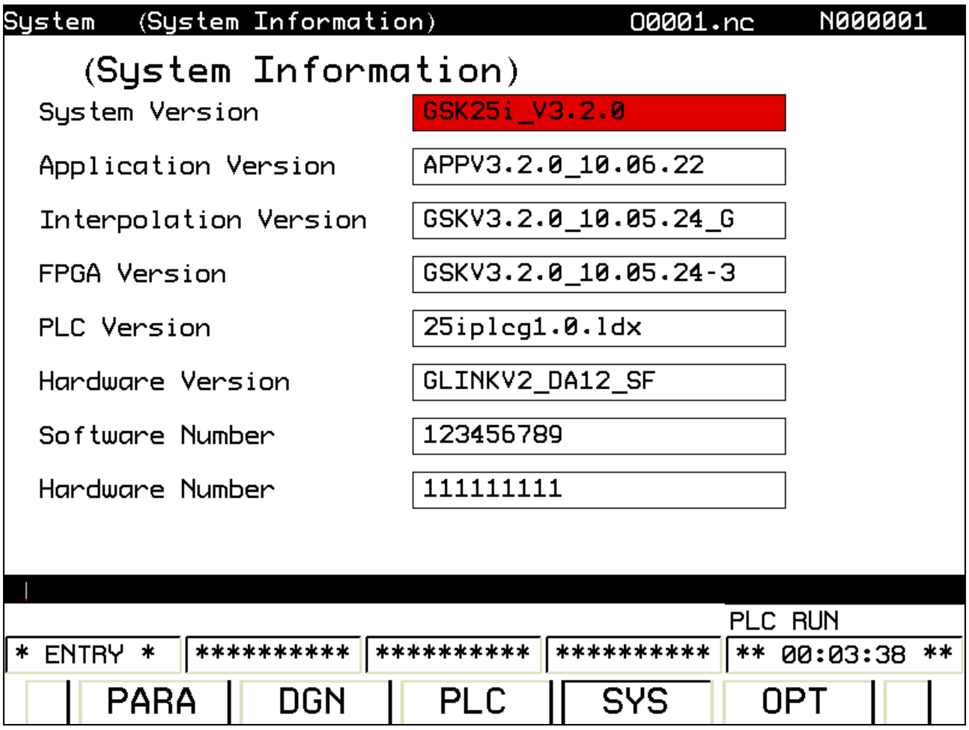


Fig. 3-52

This interface displays the current software and hardware version information in the system; the software information can not be modified, but the hardware information can be done in the case of the allowable condition.

2) **[Servo information]**

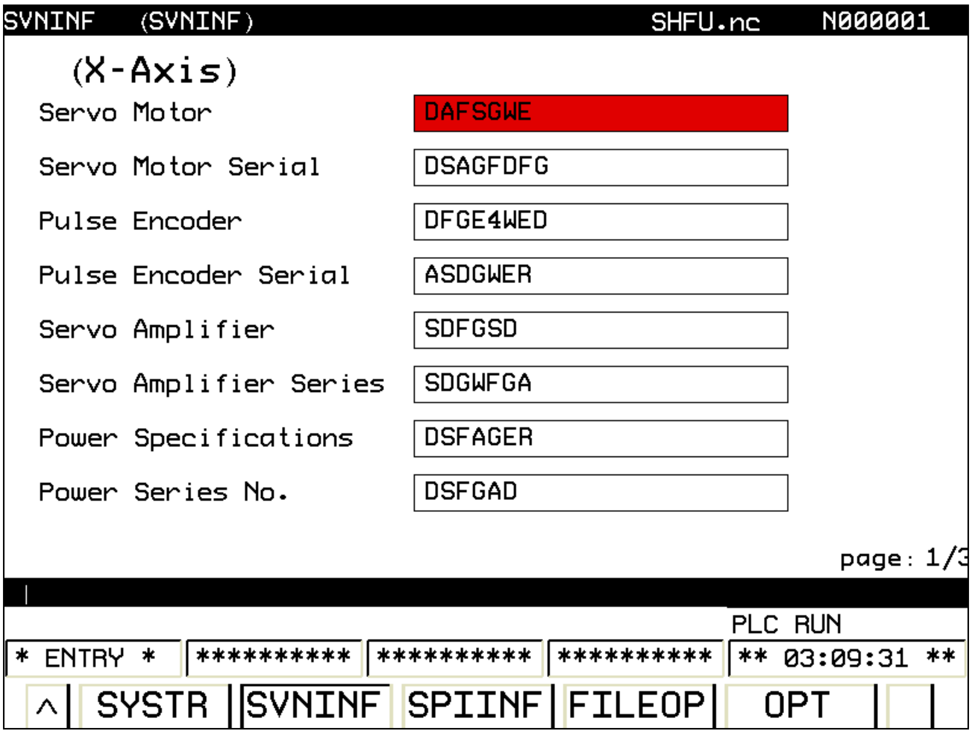


Fig. 3-53

This interface displays some character of each axis, which can be modified in the case of the allowable authority.

3) **[Spindle information]**

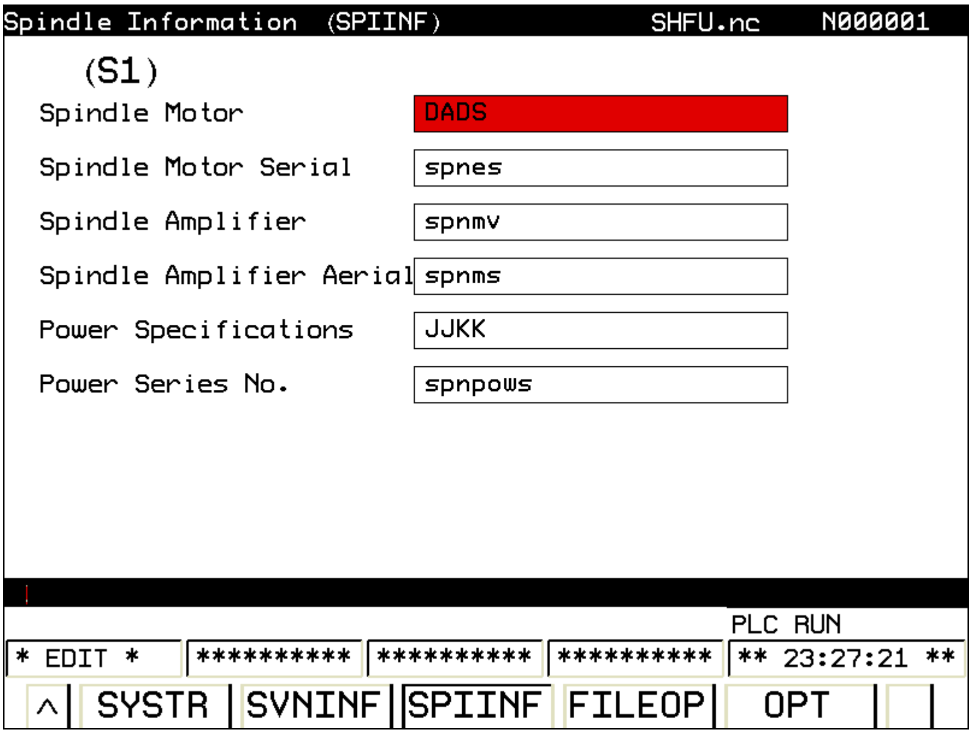


Fig. 3-54

This interface displays some relative attribute of the spindle, which can be modified in the case of the allowable authority.

3.7 Help Interface Display

Enter help display interface by pressing the 

, seven display methods are available: **[Operation]**, **[Alarm]**, **[G code]**, **[Parameter]**, **[Macro command]**, **[PLC address]** and **[Calculator]**, which can be checked by the corresponding softkeys. Refer to the following content for details. (see the figure **3-55**～**3-61**).

1. Operation interface

Press the softkey **[Operation]** to enter an operation interface on the <Help> interface. See the figure 3-55:

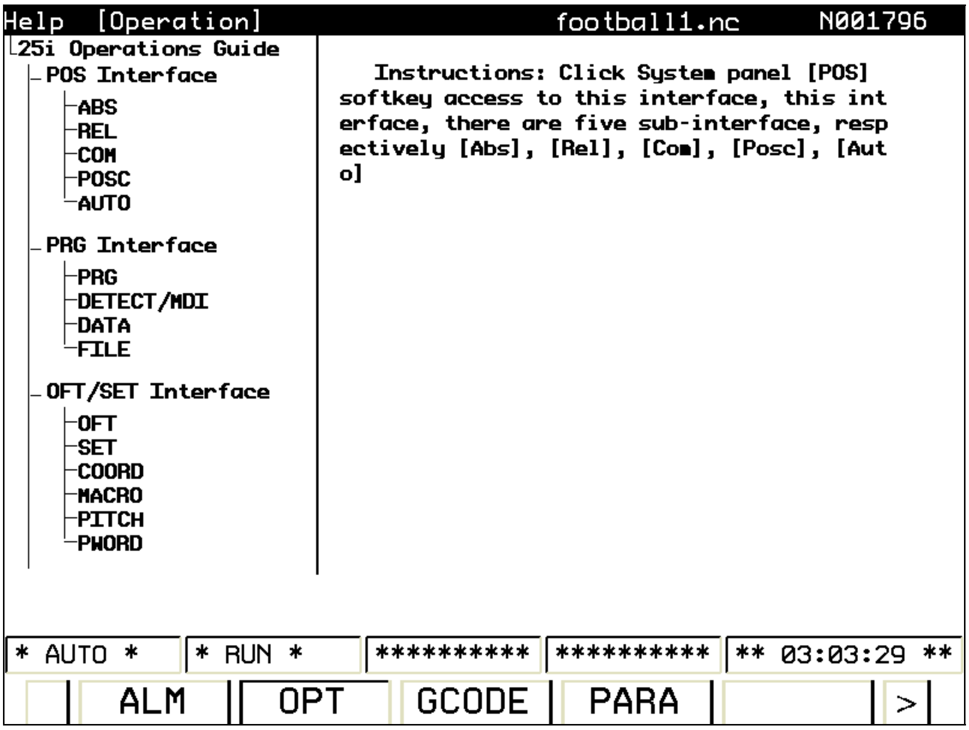


Fig. 3-55

Operation explanation: Enter this interface by pressing the **[Position]** softkey on the system panel, the interface totally has four levels, namely, **[Absolute]**, **[Relative]** as well as **[Integration]** and **[Monitor]**.

On the **[Operation]** interface, the manual operation steps and methods on each interface may be generally introduced. If the user does not familiar with the operation or unclear about the content, the search and check can be performed on this interface. Check the relative operation by selecting the corresponding items by pressing the keys ⇧, ⇩, ⇨ and ⇦.

2. Alarm interface

On the <Help> interface, enter the alarm table interface by pressing the softkey of [Alarm]. See the figure 3-56:

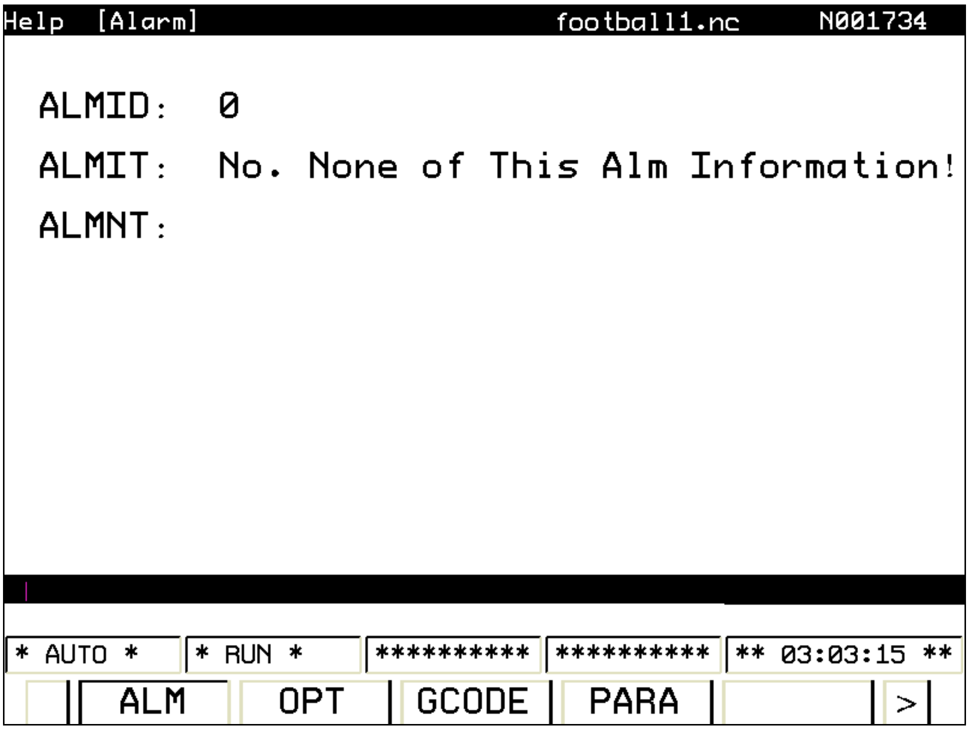


Fig. 3-56

The meaning and troubleshooting for each alarm number are described on this interface.

The corresponding content can be checked gradually by the direction keys ⇧ and ⇩. An alarm number can be input in the input column, and press the **[Input]** key to check the alarm number and its meaning which is the related treatment method.

3. G code interface

Press the softkey of **[G code]** to enter a G code interface based upon the <Help> interface. Refer to the figure 3-57:

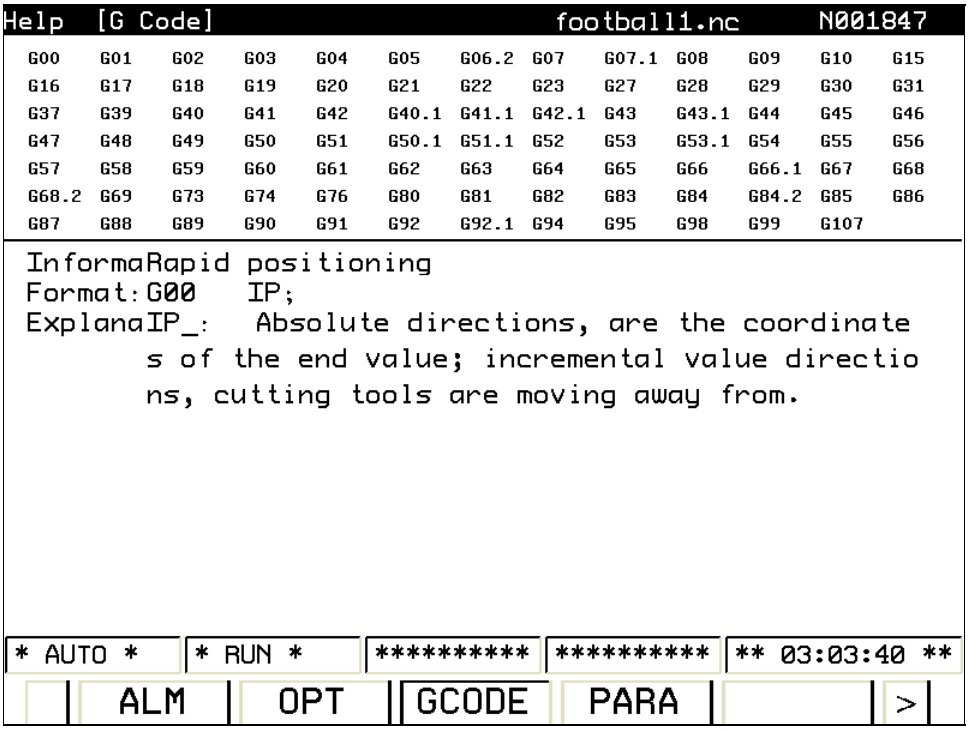


Fig. 3-57

Command description: It is a terminal coordinate value when the absolute value command is executed; and it is a tool's movement distance when the incremental command is performed.

G code definition used in the system are described on G code interface, the G code to be viewed checked should be selected based on , , and , and the G code definition is displayed below the interface. Refer to the fig.3-49, if the format and usage of G codes is being known, the G code's relative information can be checked after the G code is selected. The command format, function and explanation are described on this interface, and you can search and check the command that you are not familiar with or clear about on the interface.

4. Parameter table interface

On ＜**Help**＞interface, press the softkey of **[Parameter]** to enter the parameter table interface. Refer to the figure 3-58:

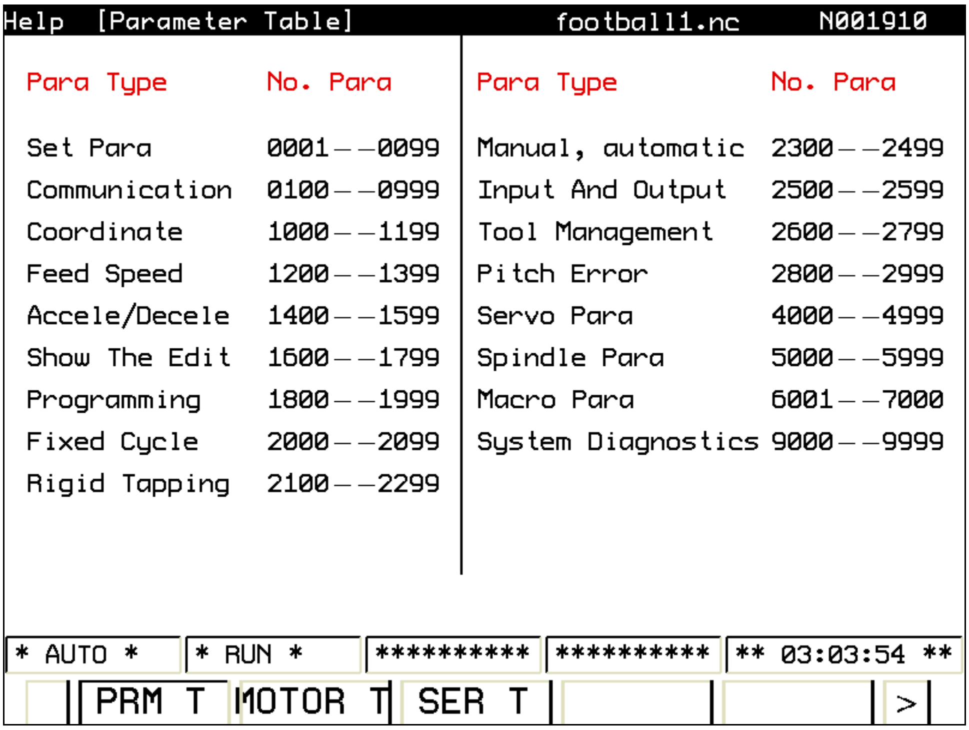


Fig. 3-58

On this interface, the parameter range corresponding to the parameter of each function is described, if you are unfamiliar or unclear about the parameter, you can check each parameter for each function in terms of the following parameter appendix on this interface, or the related function parameter search can be performed based upon this range on parameter interface.

5. Marco command interface

Press the softkey **[Marco command]** to enter the macro command interface on the <Help> interface. Refer to the figure 3-59:

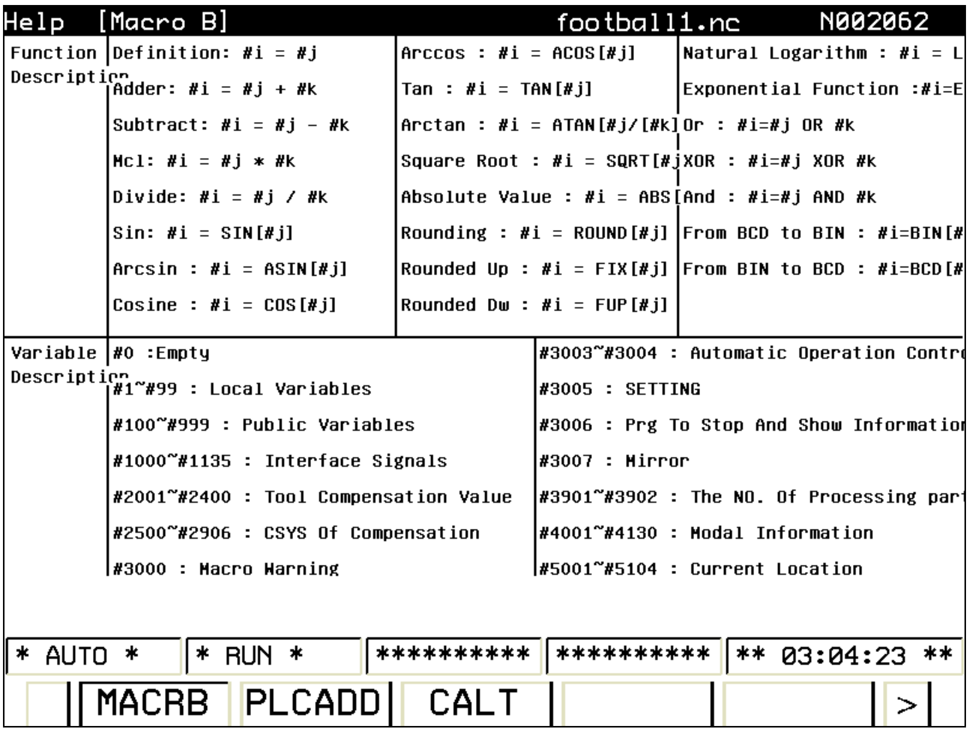


Fig. 3-59

The format of macro command and various calculation commands are described on this interface, the local variable, the common variable and the setting range of the system are available. You can search and check the command that you are unfamiliar or unclear about the Marco command on the interface.

6. PLC address interface

Press the softkey **[PLC address]** to enter the PLC address interface on the <Help> interface (Refer to the figure 3-60).

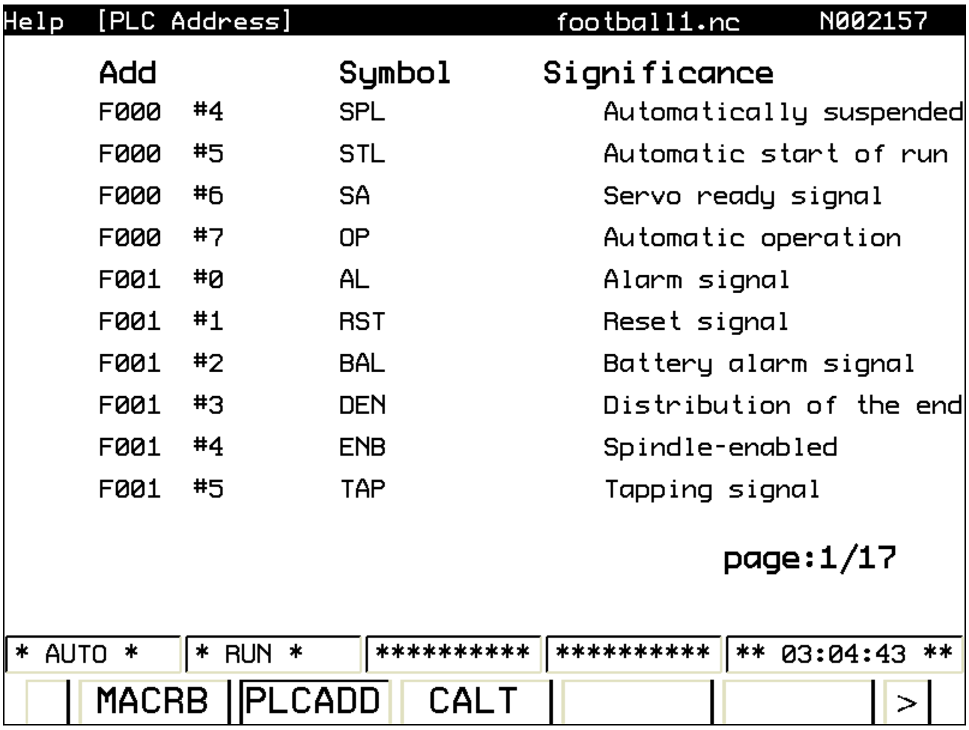


Fig. 3-60

PLC address, symbol and significance are described on this interface; you can search and check PLC address that you are unfamiliar with or unclear about on the interface. Totally 17 pages; you can view them by the page keys.

7. Counter interface

At the 2nd page of <Help> interface, press the softkey **[Counter]** to enter the counter address interface. See the figure 3-61:

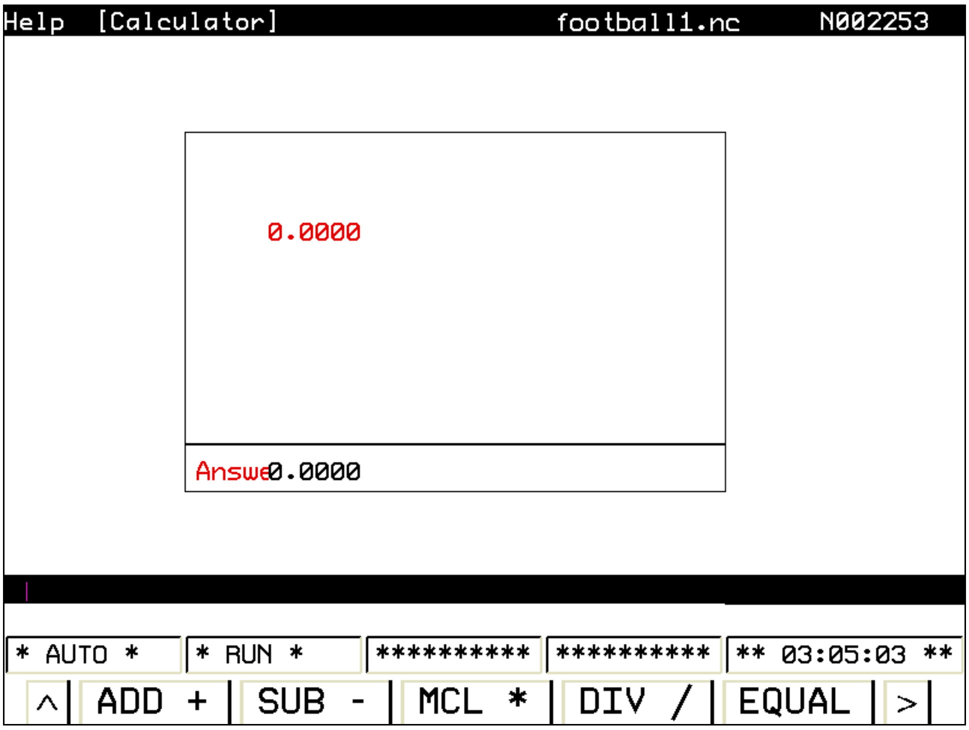


Fig. 3-61

On the interface, the operation formats: addition, subtraction, multiplication, division, sine and cosine are offered. After input the required data, and then press the button of relative operator, while the binary operator requires inputting the data, again, and press <Equal > key to confirm, the system auto calculates the result and output in the blank behind the “=”.

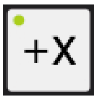
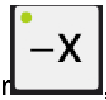
4 MANUAL OPERATION

Press the  to enter manual operation mode, which it mainly includes the manual feed, the spindle control and the machine panel control etc.

4.1 Coordinate Axis Move

In the mode of the manual operation, the five-axis can be operated at the manual feedrate or the manual rapid traverse rate.

4.1.1 Manual Feed

Press the feed axis or direction selection key  or , the direction key along with X axis moves the X axis in positive or negative, the axis movement is stopped after releasing the key. In this case, the feedrate override can be adjusted to change the feedrate and the operation is similar as other axes. This system simultaneously supports the manual five-axis movement, and the zero return also can be performed by five-axis.

Note: The manual consecution feedrate for each axis is determined by parameter **N1232;** and the manual rapid traverse rate setting is depended on **N1233.**

4.1.2 Manual Rapid Traverse Move

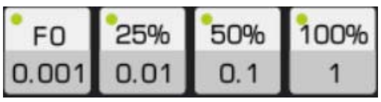
Press  key and enter the manual rapid traverse state after the indicator is lighted up, and then press the key of manual feed axis, each axis operation moves at the rapid traverse rate.. The manual rapid traverse is disabled in the manual single step mode.

Note 1: The manual rapid traverse rate is set by **N1233**.

Note 2: The manual rapid traverse move set by bit parameter **N01200#0** is valid before the reference position returns till the power is turned on.

Note 3: The feedrate is performed in manual rapid traverse, and the time constant and the acceleration/deceleration mode are same as rapid traverse rate specified with G00 program commands.

4.1.3 Manual Feed and Manual Rapid Traverse Rate Selection

When the consecution operation is performed, the manual rapid traverse rate can be selected by the  after pressing the key of rapid operation. Four gears rapid feedrate are available: Fo, 25％, 50％ and 100％. (The manual rapid traverse rate is set by **N1233**, Fo speed is set by the data parameter **No1231**). The movement speed can be selected by the feedrate override knob without performing the rapid operation key.

Note 1: The rapid feedrate selection is valid for the following traverse speed

(1) G00 rapid feed

(2) Rapid feed in canned cycle

(3) Rapid feed in the command G28

(4) Manual rapid feed

For example: When the rapid feedrate is 6m/min. If the override is 50%, the speed is 3m/min.

4.1.4 Manual Intervention

When the program is operated in the Auto, MDI and DNC modes, the program can be converted into manual mode directly. And then the manual operation can be performed to move each axis. This program can be operated in automatic mode after the operation is executed, and it is continually operated after each axis returns to the original position in G00 rapid traverse mode.

4.2 Spindle Control

4.2.1 Spindle Rotation CW

: S rotation speed can be specified in MDI mode, the spindle rotates CW by pressing this key in the mode of manual/MPG/single.

4.2.2 Spindle Rotation CCW

: S rotation speed can be specified in MDI mode, the spindle rotates CCW by pressing this key in the mode of manual/MPG/single.

4.2.3 Spindle Stop

: The spindle stops in the mode of manual/MPG/single by pressing this key.

4.2.4 Spindle Exact Stop

: The spindle accurately stops after it rotates to a fixed angle in the modes of manual and MPG by controlling this key. The spindle exact stop should be released by pressing


, otherwise, the manual rotation can not be executed.

4.3 Other Manual Operations

4.3.1 Coolant Control

: It is a compound key. The coolant is shifted between on and off. The indicator is ON when the power is turned; otherwise, it is OFF.

4.3.2 Lubricating Control

: It is a compound key. The lubriciting function is shifted between on and off. The indicator is ON when the power is turned on; otherwise, it is OFF.

4.3.3 Peck Control

: It is a compound key. The peck is shifted between on and off. The indicator is ON when the power is turned on; otherwise, it is OFF.

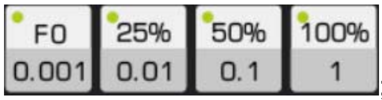
5 SINGLE STEP OPERATION

5.1 Single Step Feed

Enter single mode by pressing 

, in the single step feed mode, the machine moves based on the defined step length in the system each time.

5.1.1 The Selection of Movement Amount

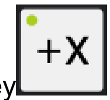
The movement increment can be selected by pressing any key of 

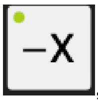
; the corresponding movement increment is then selected. For example: The single step length 0.100 is selected by pressing 

.

The corresponding machine axis moves for 0.1mm by pressing a move key each time.

5.1.2 The Selection of Move Axis and Move Direction Key

Press the feed axis and direction key 

or 

, and the X axis direction key can move in positive or negative direction along with X axis; when the key is pressed for one time, the corresponding axis moves the distance defined by the system single step, and the feedrate can be modified by adjusting the feedrate override, as other axes.

5.2 Single Step Interruption

When a program is performed in the Auto, MDI and DNC modes, it is shifted to a single mode and the single step interruption function is disabled. The single step mode becomes effective only in the manual mode.

5.3 Miscellaneous Control in Single Step Operation

It is same as the manual operation mode, and; refer to the section 4.2 and 4.3 of this operation manual for details.

6 MPG OPERATION

6.1 MPG Feed

Enter MPG mode by pressing 

, in MPG feed mode, the MPG controls the machine movement and the machine feed is accurately adjusted.

The MPG move steps:

The “mode selection” switch is set on the “MPG” position

(1) Select move axis

(2) Rotate the external hand unit of MPG

CW + direction

CCW - direction

(The direction described varies from one machine manufacture to another)

(3) Movement amount: Some panel has the following selection buttons: ×10 indicates that the movement amount multiplies 10; ×100 indicates that the movement amount multiplies 100.

Input system	**Movement amount of each grid**		
	×1	×10	×100
Metric input	0.001mm	0.01mm	0.1mm
Inch input	0.0001inch	0.001inch	0.01inch

(4) The relation between MPG scale and machine movement amount is as follows:

	The movement amount on MPG of each scale			
MPG increment (mm)	0.001	0.01	0.1	1
Machine movement amount (mm)	0.001	0.01	0.1	1

The numbers displayed on the above table vary from the mechanical drive; refer to the manual of machine manufacture for details;

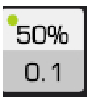
Note 1: If the MPG is rotated up to 5 rev/s, the difference may occur between the MPG rotation amount and machine movement distance, so the MPG speed must not be too fast.

Note 2: The machine tool or workpiece is moved in “rapid traverse” rotation by rotating the hand unit by ×100 override, and the machine may be impacted if it is stopped suddenly. The automatic acceleration/deceleration function becomes effective in the manual feed; therefore the mechanical impact can be reduced.

6.1.1 Selection of Movement Amount

The MPG control mode may be selected by parameter number N1401#6, (0: Reservoir, 1: Real-time).

The MPG movement increment can be selected by pressing any of the 

, for example, pressing 

means that the move increment of each MPG each scale is 0.100mm. The MPG interruption move distance is determined by the rotation amount of MPG and MPG feedrate

(This system is selected by 3 keys). Since this movement is not accelerated or decelerated, it is very dangerous to use a large magnification value for handle interruption.

Note: 1. it is useless to select 

in MPG interruption.

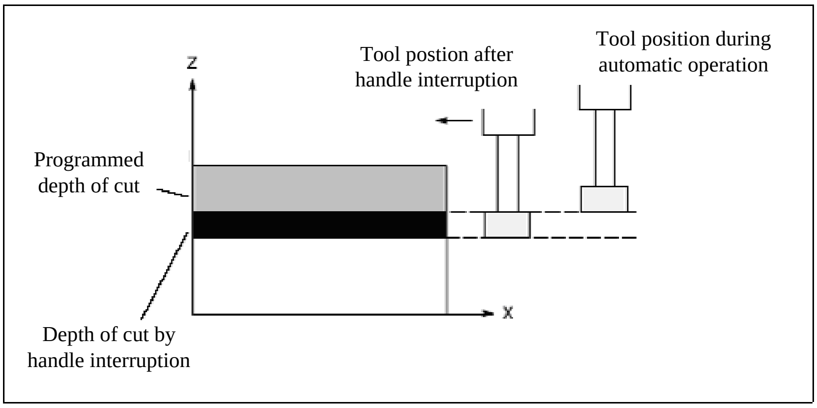
2. The MPG interruption is disabled when the machine is locked during automatic operation.

3. MPG interrupt move amount is cleared when the manual reference position return ends every axis.

6.2 Operation Control in MPG Interruption

6.2.1 The Operation of MPG Interruption

The movement by MPG interruption operation can be done by overlapping it with the movement by automatic operation in the automatic operation mode.



The operations are as follows:

1) The program operation can be switched to the MPG mode in the automatic mode.

2) MPG offset tool position, for example, The Z-axis is moved downwards, or the X and Y axes are shifted in parallel to modify the coordinate system.

3) The workpiece coordinate remains unchanged and the coordinate restores an actual value till the mechanical zero return is performed again when it is switched to the Auto mode and starts. When a program is being performed in the modes of Auto, MDI and DNC, the MPG interruption function can be performed after converting to the MPG mode.

6.2.2 The Relationship between MPG Interruption and Other Functions

Display	Relationship
Machine lock	The machine lock is enabled. Even if the MPG stops, the machine does not perform.
Absolute coordinate value	The MPG interruption does not change an absolute coordinate value.
Relative coordinate value	The MPG interruption does not change a relative coordinate value.
Machine coordinate value	Machine coordinates are changed by the travel distance specified by MPG interruption.

Note : MPG interrupt move amount is cleared when the manual reference position return ends every axis.

6.3 The Miscellaneous Control in MPG Operation

It is same as the manual operation mode; refer to the section **4.2** and **4.3** of this operation manual for details.

7 AUTOMATIC OPERATION

7.1 Automatic Operation

7.1.1 The Operation Procedure of Automatic Operation Program

A program can be loaded as long as in the mode of edit:

(a) Enter edit operation mode by pressing 

;

(b) Enter program list page by pressing 

, and move the cursor to find a target program file;

(c) A target program file is loaded by pressing the softkey of **[Operation]** to select the **[Loading program]**;

(d) Enter automatic mode by pressing 

. One line to be operated can be selected using the up/down key to enter automatic line.

Note: The current coordinate position is on the end position of the previous block operation which to be operated (If the block to be operated is an absolute programming and it is G00/G01 mode, the current coordinate position does not confirm.);

If the block to be operated is tool-change movement, it is better to confirm the current position does not interrupt or impact to the workpiece; so that the machine may result in the machine behaving unexpectedly, possibly injury to the user.

7.1.2 The Start of Automatic Operation

Press 

to operate a program automatically before the program to be started is selected in terms of the section **7.1.1,** and the program operation can be checked after shifting switching to the interface of <Position>, <Check> and <Graph>.

The program operation starts from the start line where the cursor is, so it is better to check whether the cursor is on the program line to be needed before controlling the automatic operation key. If it begins from the start line on which the cursor is not performed, the automatic operation program can be achieved from the starting line by pressing the 

and then 

.

7.1.3 Automatic Operation Stop

During program automatic operation, the system is offered five methods to stop the automatic operation program:

1. Program stop (M00)

The program operation dwells after the block containing M00 is performed, all modal information is totally registered. The program is continually performed after pressing 

.

2. Program optional stop (M01)

If 

is controlled before the program is operated, the program dwells after it is executed to the block including M01, and all modal information is totally registered. The program can be continually performed after pressing 

. If the 

does not press, it is regarded as code that the M01 does not executed.

3. Press 

The machine displays in the following status after pressing 

in automatic operation:

1) Machine feed decelerates to stop;

2) The machine still stops when a dwell (G04 command) is performed;

3) The other modal information is registered;

4) The program is continually performed after pressing 

.

4. Press 

Program skips to the head of the program by pressing the reset key, and the reset key is enabled when the **[RESET]** is displayed on its interface. The program is performed from beginning after pressing 

.

5. Press <**ESP**> 

When the program can be performed on the MDI interface of Auto, DNC and MDI modes, the machine can be stopped after shifting to the other modes. The details are as follows:

1) Shift to the edit, manual, MPG or zero return mode, the machine interruption operation is then stopped immediately.

2) Shift to the single-step mode, the machine stops after the current block is performed.

7.1.4 Spindle Control Speed in Automatic Operation

The spindle speed can be adjusted in Auto operation when the analog amount is controlled the spindle speed.

When the automatic operation is executed, the spindle speed is changed along with the spindle override varies by pressing the spindle knob, and the spindle override can be achieved **50％～120％,** total 8-level real-time adjustment:

Spindle actual speed = program command speed × spindle override. The maximum spindle speed is determined by parameter NO5142. If it exceeds this digit speed, it is then rotated at the speed.

7.1.5 Speed Control in Automatic Operation

When the automatic operation is performed, the system can be changed the feedrate by modifying the override.

The federate override can be modified by the rotation knob, and it can be achieved **0％～200％,** totally 20-level real-time adjustment.

Note: In feedrate adjusting program, the value set by F is programming speed.

The acutal feedrate = the value set by F × feedrate

However, in the automatic operation, the rapid traverse speed can be selected by pressing the 

, and the rapid override can be achieved four-gear adjustment, namely, Fo, 25％, 50％ and 100％.

Note 1: The value of rapid traverse speed is calculated by the data parameter No.1230 and the final modification at the rapid traverse rate, which is as follows:

X axis actual rapid traverse speed = the value set by No.1230 X rapid override

If the override is set to Fo, whether the axis is stopped by parameter N01200＃4, if it is set to 0 but not stop, the actual rapid traverse speed is determined by parameter No.1231 (Generally use for all axes).

The actual rapid traverse speed both Y and Z axes are calculated as above-mentioned.

7.1.6 Dry Run

The program can be checked by the “dry run” before the program is automatically operated.

Enter automatic operation mode by pressing 

, then 

(In the state indication area, the indicator goes on means that the dry run is is performed).

The program speed in rapid feed is dry run speed.

The program speed in cutting feed is dry run speed.

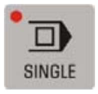
Note: 1. The dry run speed is determined by data parameter No1210 (Generally use for all axes);

2. Whether the dry run is enabled at rapid feed which is determined by bit parameter No.1200＃6;

7.1.7 Single Block Operation

Before the automatic operation, the program single-block operation can be selected if its operation situation is required to be checked.

Enter the automatic operation mode by pressing 

, then the 

(In state indication area, the single-block operation indicator is ON means that the single operation state is performed). When the single-block operation is executed, the system stops running after each block is completed.

In this case, it is necessary to press 

again if you want to perform it continue, and the operation should be repeatedly executed till the program running completes.

Note 1: In G28, the single block stop can also be performed at an intermediate point;

Note 2: The single block is disabled when the called subprogram (98_) and subprogram call return command (M99) are performed. But in the M98 and M99 blocks, the single-block operation is enabled when the addresses other than N, O and P are still specified.

7.1.8 All Axes Function Lock Operation

Press 

(In the state indication area, the machine lock operation indicator **ON** means that the machine lock operation state has been performed) in automatic operation mode. In this case, the machine does not move, but the position coordinate display is same as that of the machine movement, the current operation situation can be checked from **[Monitor]** interface, and then the **M, S and T** can be performed. This function is used for program checking.

Note: After the machine lock has been pressed and then the program has been performed, the machine zero return operation should be performed due to the machine position does not agree with the coordinate one.

7.1.9 Miscellaneous Function Lock Operation

Press 

(In the state indication area, the miscellaneous function lock operation indicator **ON** means that the M.S.T function lock state has been performed) in automatic operation mode. In this case, M, S and T code commands are not performed, which is used a program check with the machine lock function.

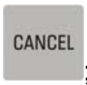
Note: M00, M30, M98 and M99 are performed normally.

7.2 MDI Operation

The MDI operation function is added which the command operation can be directly input with this function.

7.2.1 MDI Program Edit

In MDI mode, after the code is input, the functions, namely, the search, search line number as well as the single, multiple selection, copy and paste which can be performed similar to the editing mode.

If the field input is incorrect before pressing the cycle start key, cancel the input code one by one by pressing 

; if a mistake occurs after inputting, the correct content can be input again and the incorrect one can be replaced by pressing 

.

7.2.2 MDI Command Operation and Stop

The MDI can be operated by pressing 

after the command is input. The command operation can be stopped by pressing 

during operating. At the end of the program without M30, the cursor does not skip to the top of the program after the operation is execute. The operation can be performed again by a reset.

Note: MDI operation must be performed in the MDI mode!

7.3 Conversion of Operation Modes

The automatic operation may stop immediately after shifting to Manual, MPG and machine zero, the feed hold indicator goes on. In the automatic operation state, only when the current line is enabled after shifting to the MDI, DNC and Edit mode.

The MDI mode operation may stop immediately after shifting to the Manual, MPG and machine zero mode, the feed hold indicator goes on. The current line is performed before shifting to the Auto, DNC or Edit mode.

8 ZERO RETURN OPERATION

8.1 Machine Zero Return

8.1.1 Machine Zero Point Concept

Machine coordinate system is a fixed one of machine, of which its origin is called as machine zero point, and it is also referred to as a **Reference point**; generally, it is installed at the maximum stroke along with the X-axis, Y-axis, and Z-axis positive direction. This fixed origin is confirmed after the machine is designed, manufactured, and debugged. Normally, the machine zero point is not be recognized till the CNC device power is turned on, the machine zero point return is performed in Auto or by manual operation.

8.1.2 The Operation Procedures of Machine Zero Return

The machine zero return can be performed both in Auto mode and or by G code.

1. Automatic zero return

1) Enter the mechanical zero return operation mode by pressing 

, the "zero return" can be displayed on the LCD screen, in this case.

2) The X, Y, Z as well as the 4th or 5th axis, which is to be returned the machine zero point, is selected, and the zero direction is determined by bit parameter **No.1004＃5.** (This system supports five-axis zero return simultaneously)

3) The machine moves along with its zero point direction, the machine rapidly moves before decelerating (The move speed is set by data parameter **No.1231**), the machine moves to its zero point (it is also called as reference position) in terms of the FL speed (it is set by data parameter **No.1234**) after the deceleration switch is performed. The coordinate axis stops and zero return indicator is power-on when the machine zero point is returned.

2. Program command mechanical zero return

After the bit parameter **No.1001＃3** is set to 0, the reference position return can be performed by the deceleration block specified with G28 because the check line block is shared a same function with the manual mechanical zero return.

Note 1: Never attempt to use machine zero return operation if your CNC machine does not install it;

Note 2: The corresponding indicator lights up when the machine zero point return is terminated

Note 3: The indicator is power-off after the machine zero point is returned if operator moves out a corresponding axis from the machine zero point;

Note 4: Refer to the manual issued by the manufacturer for the machine zero point (reference position) direction.

3. The debugging method of zero return:

1) The related parameter of zero return:

Zero return direction setting (1004#)

Movement amount per revolution for each axis (1060#)

FL speed of reference position return for each axis (1234#)
Reference position return speed for each axis (1235#)
The 2nd FL speed of reference position return for each axis (1236#)
Mechanical zero return acceleration (1444#)

2) Zero return schematic chart

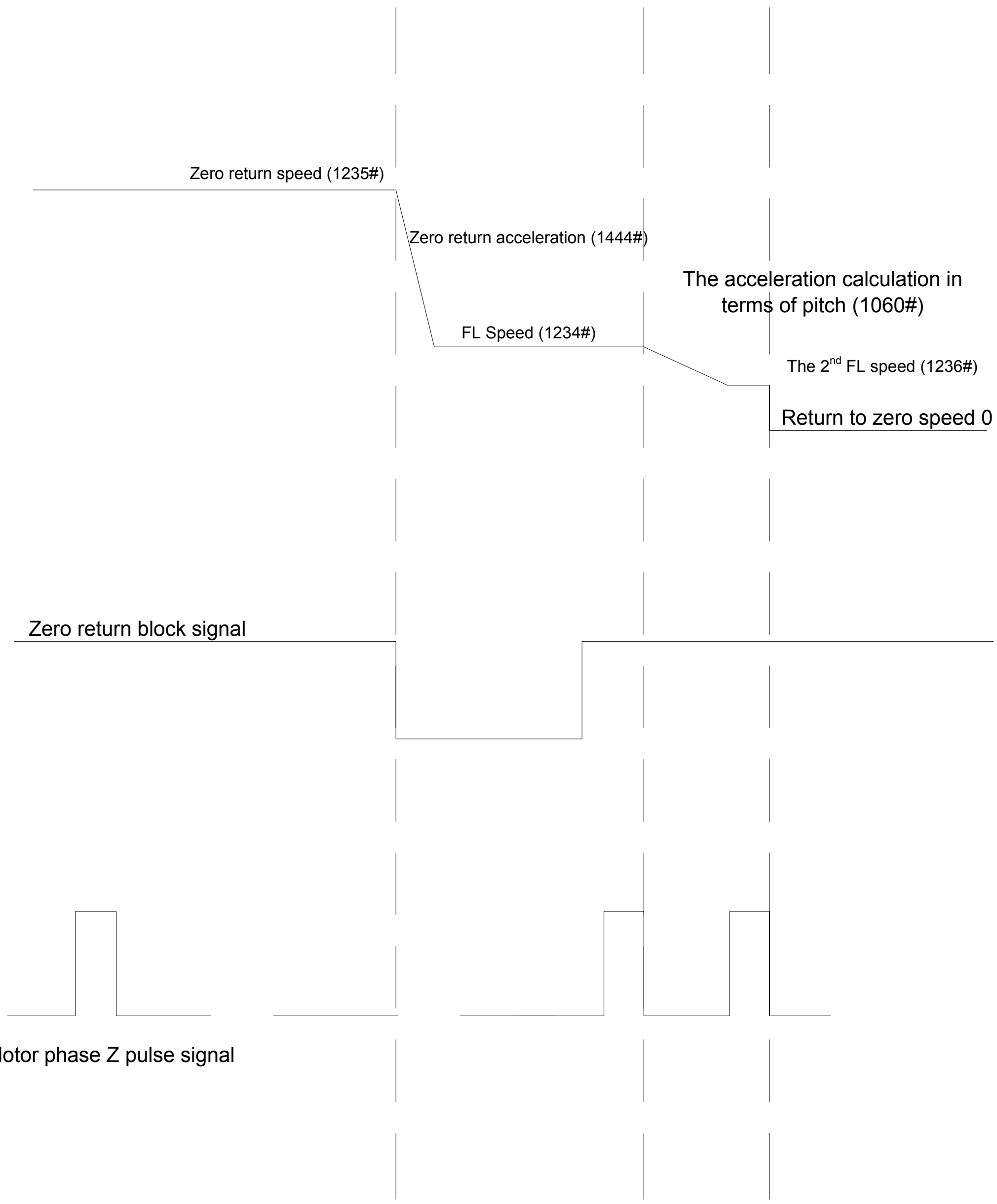


Fig. 8-1

3) The adjust steps of zero return parameter.

A. Confirm zero return direction (1004#) in terms of machine.

B. Confirm the movement amount (1060#) per revolution for each axis based upon machine.

C. Zero return speed and zero return acceleration

Reference position speed return for each axis (1235#) (default: 3000, 3000, 3000, 2000, 2000, and 2000)

FL speed of reference position return for each axis (1234#) (default: 300, 300, 300, 75, 75, and 75)

FL speed of reference position return for each axis (1236#) (default: 7, 7, 7, 2, 2, and 2)

Mechanical zero speed return (1444#) (default: 0.3, 0.3, 0.3, 150, 150, and 150)

D. Confirm whether the zero return block signal for each axis is normal.

Enter PLC diagnosis interface, and select the X resource.

The machine moves manually when machine passes the zero return block position. View whether the zero return block signal (X9.0~X9.4) input has a corresponding change.

E. Zero return operation for each axis is performed separately

View whether the reference position speed return (1235#), FL speed (1234#) and the 2^{nd} FL speed (1236#) for each axis is held more than 2 seconds.

If the reference position return speed (1235#) is held less than 2 seconds, it is essential to move this axis along with zero return negatively depart from the reference position. The zero return acceleration (1444#) parameter should be increased if it is arrived to a movement terminal.

If the FL speed of reference position return (1234#) is held less than 2 seconds, it is necessary to increase the zero return acceleration (1444#) parameter. If the acceleration increase does not valid, the block length may exceed long. In this case, attempt to reduce both the reference position return speed (1235#) and FL speed (1234#).

If the 2^{nd} FL speed (1236#) of reference position return is held less than 2 seconds, confirm whether the motor move amount parameter per revolution is correct firstly; if it is correct, the phase Z signal may be abnormal.

9 SYSTEM COMMUNICATION

This system can be communicated with PC by the series terminal port and Ethernet, as well as read the USB device directly. Refer to the operating explanation for details:

9.1 Series Terminal Port Communication

GSK25i PCCom serial terminal port communication software is window interface, which is used for DNC machining from PC port to CNC port. This software can be applied to Win98, WinMe, WinXP and Win2K.

9.1.1 Program Start

The “serial terminal port DNC” program in desktop shortcut mode is performed directly; the interface is displayed after the program starts, refer to the following figure (9-1):

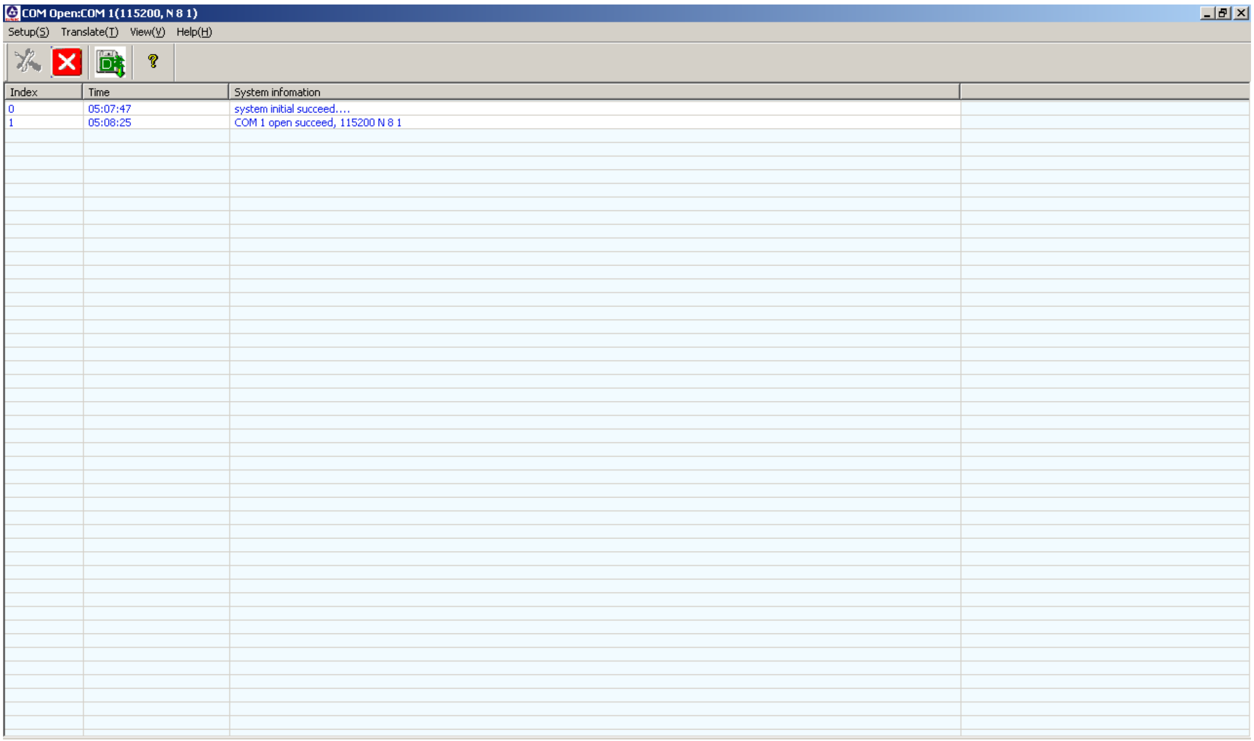


Fig. 9-1

9.1.2 Function Introduction

1) Setting

The setting menu mainly contains: serial port setting, serial port close and exit the program

2) Transmission

The transmission menu is DNC machining transmission

3) View

Display and concealment of both toolbar and status bar

4) Help

Version information of this software

9.1.3 Software Usage

The details of this software are as follows:

1. Both CNC and PC are connected.

2. Open and set a serial port. The communication parameter of serial port can be set by , refer to the following graph. The DNC Baud rate of GSk25i system is 38400 by default, which can be set by parameter anew. The data bit is 8-digit, the stop bit is 1 digit, and the parity check is not performed. The data bit, stop bit and parity check can not be changed.

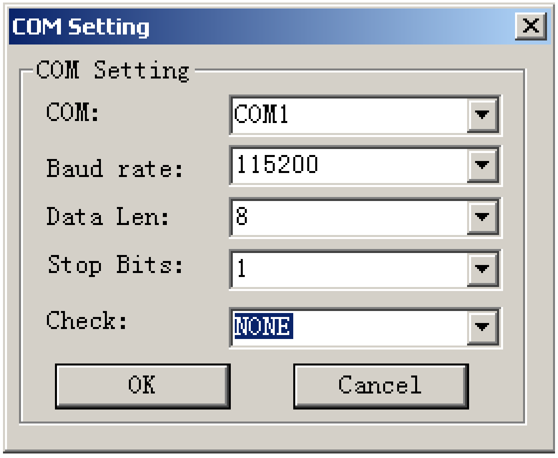
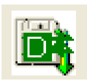


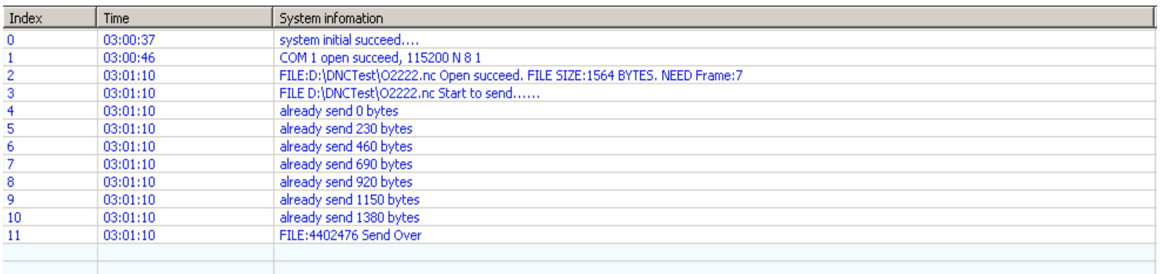
Fig. 9-2

3. Set the IO CHANNEL of DNC transmission parameter to a serial port one, switch to the DNC mode by pressing the  on CNC panel.

4. Press  on PC software, and select the NC program of DNC machining to be performed, and then the data transmission begins after confirming.

5. Press  on CNC panel to machine.

6. During the whole transmission, this program may display the time and transmission information in the mode of list. (See fig. 9-3)



Index	Time	System infomation
0	03:00:37	system initial succeed....
1	03:00:46	COM 1 open succeed, 115200 N 8 1
2	03:01:10	FILE:D:\DNCTest\O2222.nc Open succeed. FILE SIZE:1564 BYTES. NEED Frame:7
3	03:01:10	FILE D:\DNCTest\O2222.nc Start to send......
4	03:01:10	already send 0 bytes
5	03:01:10	already send 230 bytes
6	03:01:10	already send 460 bytes
7	03:01:10	already send 690 bytes
8	03:01:10	already send 920 bytes
9	03:01:10	already send 1150 bytes
10	03:01:10	already send 1380 bytes
11	03:01:10	FILE:4402476 Send Over

Fig. 9-3

9.2 Network Communication

GSK25i system can be achieved many functions, such as machine monitoring, system maintenance and remote **DNC** by the Ethernet. This software is **window** interface which can be applied to Win98, WinMe, WinXP and Win2K.

9.2.1 Program Start

The shortcut of "machine monitoring" on the desktop can be performed directly, and the interface is shown below after the program is executed. (See the figure 9-4):

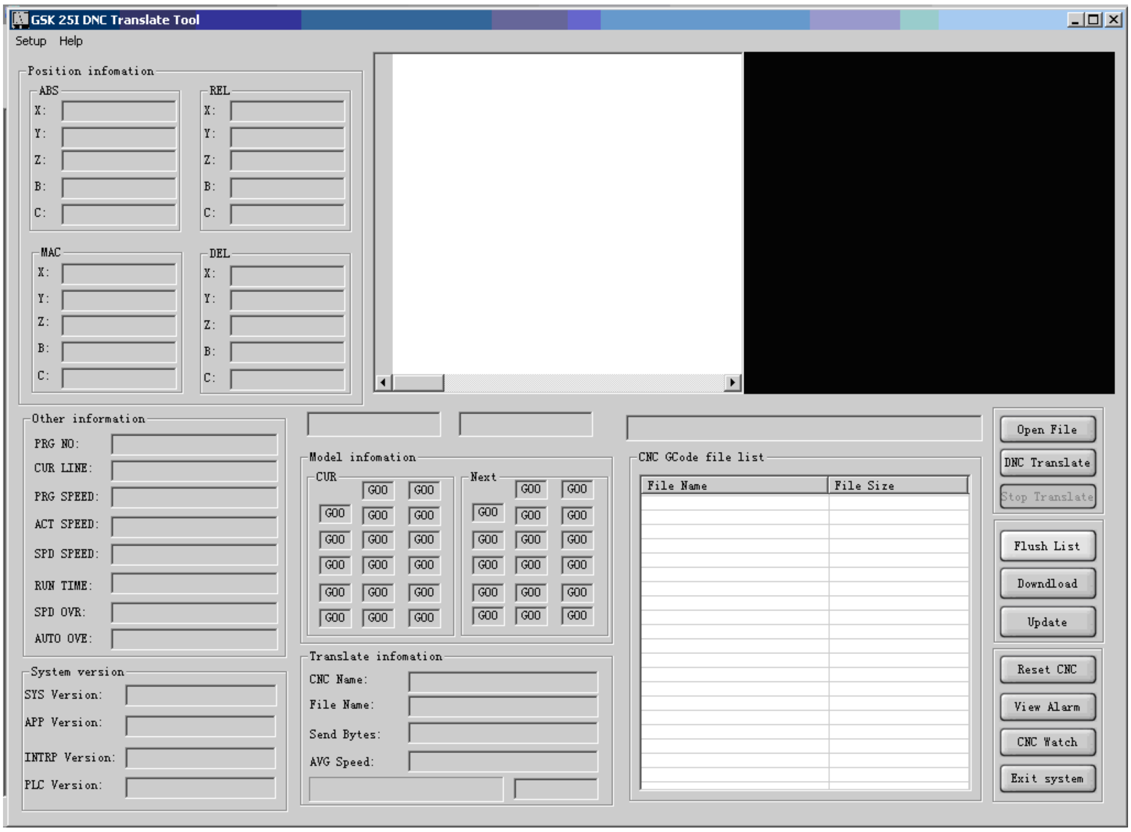


Fig. 9-4

9.2.2 Software Usage

1. Machine monitoring

a> The GSK25iCNC system and PC are connected using mesh.

b> Set CNC parameter and IP address correctly, enable the net communication function of CNC.

c> Press the "setup" button to open a communication interface setting is shown below, and after a correct connection mode is confirmed, it can be connected to the CNC system.

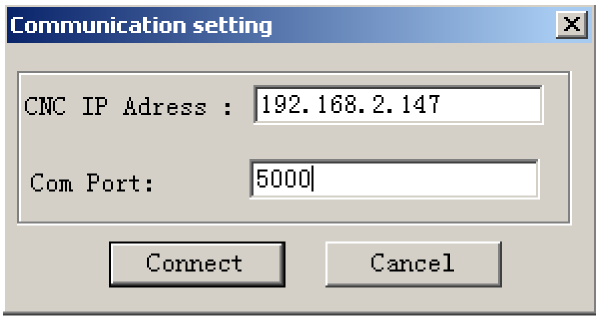


Fig. 9-5

d> After connecting the system, namely, the machine monitoring can be performed. Refer to the following interface (Fig. 9-6):

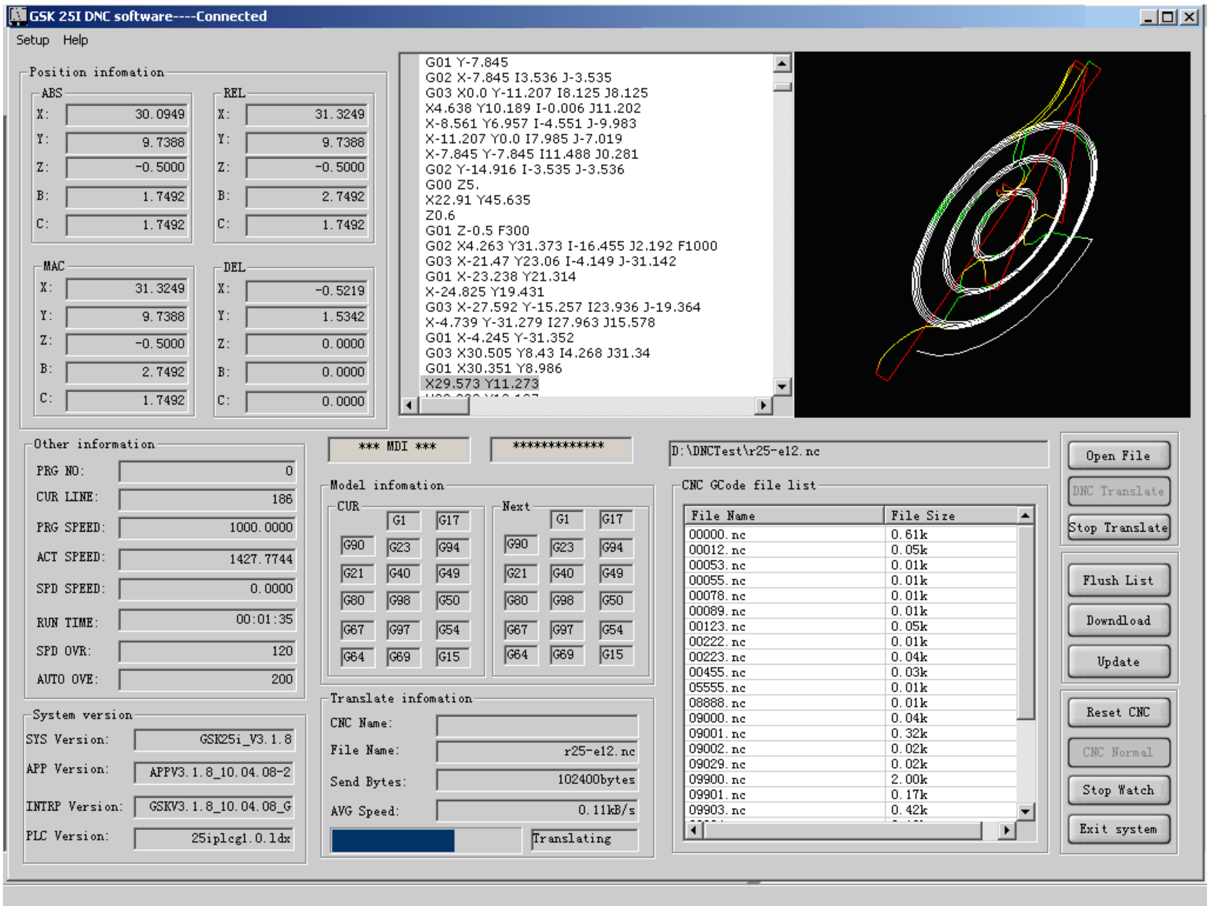


Fig. 9-6

2. Network DNC

a> Select the correct IO channel in CNC, refer to GSK25i system parameter manual.

b> Click the button of “Open file” on PC software, and select the NC file of DNC machining to be performed. After confirming, the file content may display on its software, the file path name may display at the right side of the button.

c> Click “DNC” transmission and the network data transmission begins to perform.

d> Press the button of “start” on CNC panel and the machining begins to perform.

3, File transmission

a> Upload file

Based upon the normal connection between the software and CNC, select the NC file to be uploaded by clicking “upload program”, and then the file can be uploaded after confirming. However, the machine monitoring can not be specified during uploading the file; otherwise, click the “machine monitoring” button. And the following operation is same.

b> Download file

Click the button of “Refreshing list” and the file list on the CNC system can be gained in the case of the software and CNC terminal are well connected. Select a file to be downloaded in machine NC file list and then the file can be downloaded by clicking the button of “Download program”. The machine monitoring is not performed during transmitting the file; it is necessary to click “machine monitoring” button if the machine monitoring is performed.

c> The transmission state is displayed in real-time under the monitoring interface during uploading and downloading, refer to the Fig. 9-7.

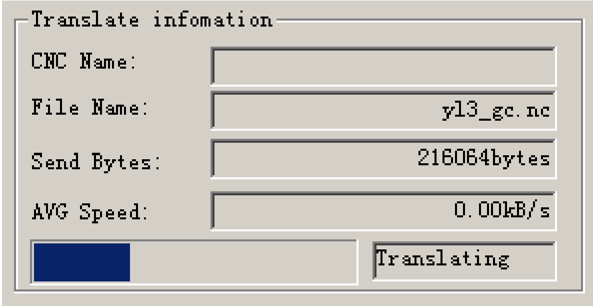


Fig. 9-7 Transmission state display

4. System maintenance

After the system and PC are connected, the buttons such as backup and update are displayed at the lower right corner on the system monitoring interface. The corresponding parameter file should be selected during uploading and updating; It is necessary to set a saved file name during downloading a backup.

Appendix Alarm List

Alarm No.	Alarm Content	Remark
0	PLEASE TURN OFF POWER	
1	PARITY ALARM	
3	TOO MANY DIGITS	
4	ADDRESS NOT FOUND	
5	NO DATA AFTER ADDRESS	
6	ILLEGAL USE OF NEGATIVE SIGN	
7	ILLEGAL USE OF DECIMAL POINT	
8	ILLEGAL DATA	
9	ILLEGAL CHARACTERS IN THE ADDRESS	
10	IMPROPER G-CODE	
11	NO FEEDRATE COMMANDED	
12	BLOCK LONG	BEYOND MAX. UNIT NO. 32
13	A LONG STRING UNIT	A UNIT CHARACTER STRING EXCEEDS 16
14	CAN NOT COMMAND G95	
15	TOO MANY AXES COMMANDED	
16	ILLEGAL SEQUENCE NO	ILLEGAL N SEQUENCE NO OR OTHER THAN THE NO. OF 1～999999
17	ILLEGAL MACRO PROGRAM NO	ILLEGAL P MACRO SEQUENCE NO. OR BEYOND MAX. REPEAT NO.
18	P/X TO SUSPEND THE TIME ILLEGAL OR OVERTIME	
20	OVER TOLERANCE OF RADIUS	
21	ILLEGAL PLANE AXIS COMMANDED	
22	NO CIRCULAR RADIUS	
25	CANNOT COMMAND F0 IN G02/G03	
27	NO AXES COMMANDED IN G43/G44	
28	ILLEGAL PLANE SELECTION	

29	ILLEGAL OFFSET VALUE	
30	ILLEGAL OFFSET NUMBER	
31	ILLEGAL P COMMAND IN G10	
32	ILLEGAL OFFSET VALUE IN G10	
33	NO INTERSECTION AT T COMP	
34	NO CIRC ALLOWED IN ST-UP/EXT BLK	
35	WORKPIECE POS CHANGED	
36	CAN NOT COMMANDED G31 IN CUTTING COMP	
37	PANLE SWITCH IN TOOL COMPE	
38	INTERFERENCE IN ARC BLK, CONSISTENT ARC CENTER WITH START & END	
39	END POINT NOT ON ARC AFTER COMP	
40	COMP NUMBER IS 0	
41	INTERFERENCE IN CRC	
44	G27—G30 NOT ALLOWED IN FIXED CYC	
45	NO Q COMMAND IN FIXED CYC	
46	ILLEGAL REFER. RETURN COMMAND	
49	CAN NOT JUDGE OR NOT EXIST	
52	NO MOVEMENT COMMAND	
53	MOVEMENT COMMAND LESS THAN 2	
56	BY ZERO	
57	ANGLE PARAMETER ERROR	
58	TOOL COMP DIR PARA ERROR	
59	PROGRAM NUMBER NOT FOUND	
60	SEQUENCE NUMBER NOT FOUND	
61	FIXED CYCLE ERROR	
70	NO PROGRAM SPACE IN MEMORY	
71	DATA NOT FOUND	
72	TOO MANY PROGRAMS	
73	PROGRAM NUMBER ALREADY IN USE	
74	ILLEGAL PROGRAM NUMBER	
75	PROGRAM PROTECTION	
76	ADDRESS P NOT DEFINED	
77	SUBPROGRAM NESTING ERROR	

78	CRRESPONDING SERIAL NO. NOT FOUND	
79	PROGRAMS FOR CALIBRATION ERROR	
80	AUTOMATIC MEASUREMENT TOOL ERROR	
85	COMMUNICATION ERROR	
90	REFERENCE POINT RETURN IS INCOMPLETE	
91	INCORRECT MACROS	
92	NO. OF ILLEGAL MACRO VARIABLES	
93	DNC USED MACROS	
94	TOO MANY EXTERNAL ALARM INFORMATION	
95	BACKGROUNDS EDIT ERROR	
96	NO COORESPONDING OF DO-END	
97	ILLEGAL CYCLE NUMBER	
100	WRITING PARAMETERS ENALBE	
101	THE END COMMAND NO M30	
102	COMPUTING DATA SPILL	
103	DNC UNDER THE M99 TO BE RE-LOADED	
104	ILLEGAL TOOL GROUP NO.	
105	TOOL GROUP NO. NOT FOUND	
106	T CODE NOT FOUND	
107	ILLEGAL VARIABLE NO. SPECIFIED 115	
108	IN G65 BLK, UNDEFINED H IS SPECIFIED 114	
109	CANCEL G51 IN G27-G30 G54-G59 G92	
110	ILLEGAL PROPORTION RATE	
111	ADVANCED PREVIEW NC FULL	
112	> MAX. SPEED CHANGE IN A CYCLE	
113	> LAST CYCLE SPEED IN ACCE	
114	> LAST CYCLE SPEED IN DECE	
115	VISIT EXCLUSIVE FAILURE	
116	ACCE CYCLES < 0	
117	DECELERATION CYCLES < 0	
118	> BLOCK LENGTH	
119	LENGTH LOGO > 0	
120	BLOCK END LESS THAN A CYCLE >1	
121	SPEED UP THE > ORIGINAL ACCELERATED	

122	SLOW DOWN > THE ORIGINAL SPEED	
123	ADJUSTED TOTAL NUMBER OF CYCLES <= 0	
124	TUNE SPEED CALCULATION ERROR	
125	ADJUST LENGTH INCREMENT <0	
126	GREATER THAN CONVERGENCE SPEED	
127	> MAX. ALLOWABLE SPEED	
128	> THE FIRST SPEED	
129	CALCULATION ERROR	
130	AT EXIT STAGE, DOES NOT MEET THE CONDITIONS	
131	MOBILE COMMAND TRANSMISSION ERROR	
132	G CODE TRANSMISSION ERROR	
133	PLANE TRANSMISSION ERROR	
134	> MAX. ARC ALLOWABLE SPEED	
135	MAX. ACCE ERROR	
136	TOTAL INTERPOLATION CYCLE ERROR	
137	ERROR FOR NO REFERENCE POINT RETURN	
138	> MAX. ALLOWABLE ACCE OF SINGLE BLOCK	
139	LEAVE DECE BLOCK	
140	MPG INTERRUPTION, STEP INVALID	
141	VP SIGNAL INVALID IN RIGID	
142	MACHINE LOCK CANCEL	
143	ALARM STOP	
144	INPROPOR POSITIONING	
147	POSITION, SERVO PARA TRANSMISSION ERROR	
148	ABNORMAL IN SERVO COMMUNICATION	
150	X-AXIS SERVO ALARM	
151	Y-AXIS SERVO ALARM	
152	Z-AXIS SERVO ALARM	
153	A-AXIS SERVO ALARM	
154	B-AXIS SERVO ALARM	
155	C-AXIS SERVO ALARM	
156	SPINDLE SERVO ALARM	
160	X-AXIS SERVO NOT READY	
161	Y-AXIS SERVO NOT READY	

162	Z-AXIS SERVO NOT READY	
163	Z-AXIS SERVO NOT READY	
164	B-AXIS SERVO NOT READY	
165	C-AXIS SERVO NOT READY	
166	SPINDLE SERVO NOT READY	
170	+X-AXIS SOFT LIMIT	
171	+Y-AXIS SOFT LIMIT	
172	+Z-AXIS SOFT LIMIT	
173	+A-AXIS SOFT LIMIT	
174	+B-AXIS SOFT LIMIT	
175	+C-AXIS SOFT LIMIT	
176	-X-AXIS SOFT LIMIT	
177	-Y-AXIS SOFT LIMIT	
178	-Z-AXIS SOFT LIMIT	
179	-A-AXIS SOFT LIMIT	
180	-B-AXIS SOFT LIMIT	
181	-C-AXIS SOFT LIMIT	
182	+X-AXIS OVERTRAVEL	
183	+Y-AXIS OVERTRAVEL	
184	+Z-AXIS OVERTRAVEL	
185	+A-AXIS OVERTRAVEL	
186	+B-AXIS OVERTRAVEL	
187	+C-AXIS OVERTRAVEL	
188	-X-AXIS OVERTRAVEL	
189	-Y-AXIS OVERTRAVEL	
190	-Z-AXIS OVERTRAVEL	
191	-A-AXIS OVERTRAVEL	
192	-B-AXIS OVERTRAVEL	
193	-C-AXIS OVERTRAVEL	
194	OVERHEAT CONTROL UNIT	
195	MOTOR OVERHEAT	
196	OVERHEAT SPINDLE	
200	ILLEGAL S CODE COMMAND	
201	ILLEGAL M CODE COMMAND	

202	ILLEGAL T CODE	
203	ILLEGAL MACRO PROGRAM SEQUENCE NUMBER 128	
204	NO DECIMAL POINT	
205	ILLEGAL MACRO PROGRAM COMMAND	
206	MACRO PROGRAMS MALFORMED EXPRESSION	
207	MACRO PROGRAMS NO EXPRESSION OPERAND	
208	MACRO EXPRESSIONS DIVISOR IS 0	
209	MACRO BRACKETS NESTED ERROR	
210	CORNER RADIUS U IS BIGGER, OR I/J IS LESS	
211	U<TOOL RADIUS	
212	I, J EXCESSIVE SMALL, OR L IS TOO MUCH	
213	TOO MUCH L VALUE	
214	CANNED CYCLE PUNCHING G73-G89 DEFINED	
215	UNDEFINED I, OR I IS 0	
216	UNDEFINED J, OR J IS 0	
217	W IS LESS, OR UNDEFINED W	
218	Q IS LESS, OR UNDEFINED Q	
219	TOO MUCH OF L, OR UNDEFINED L	
220	V IS LESS, OR UNDEFINED V	
221	UNDIFIED D, OR D IS 0	
222	TOO MUCH OF TOOL RADIUS	
223	TOO MUCH OF L	
224	U<0	
225	SPECICAL CANNED CYCLE NO. IS 0	
226	J>1	
227	CANNED CYCLE IN G17	
228	UNDEFINED E, OR E IS 0	
231	X-SERVO OVERSPEED	
232	X-SERVO OVER VOLTAGE	
233	X-SERVO UNDERVOLTGAE	
234	X-SERVO POS OVER DIFF	
235	X-SERVO OVERHEAT	
236	X-SERVO SPEED SATURATION FAULT	
237	X-SERVO FORBID FAULT	

239	X-SERVO ENCODE FAULT	
240	X-SERVO ENCODE ZERO TIMEOVER	
241	X-SERVO IPM FAULT	
242	X-SERVO OVERCURRENT	
243	X-SERVO OVERLOAD	
244	X-SERVO BRAKE FAULT	
246	X-SERVO POWER OFF	
248	X-SERVO VERSION ERROR	
249	X-SERVO ENCODE CRC ERROR	
250	X-SERVO EEPROM ERROR	
251	Y-SERVO OVERSPEED	
252	Y-SERVO OVER VOLTAGE	
253	Y-SERVO UNDERVOLTGAE	
254	Y-SERVO POS OVER DIFF	
255	Y-SERVO OVERHEAT	
256	Y-SERVO SPEED SATURATION FAULT	
257	Y-SERVO FORBID FAULT	
259	Y-SERVO ENCODE FAULT	
260	Y-SERVO ENCODE ZERO TIMEOVER	
261	Y-AXIS SERVO ENCODER ZERO TIMEOVER	
262	Y-SERVO OVERCURRENT	
263	Y-SERVO OVERLOAD	
264	Y-SERVO BRAKE FAULT	
266	Y-SERVO POWER OFF	
268	Y-SERVO VERSION ERROR	
269	Y-SERVO ENCODE CRC ERROR	
270	Y-SERVO EEPROM ERROR	
271	Z-SERVO OVERSPEED	
272	Z-SERVO OVER VOLTAGE	
273	Z-SERVO UNDERVOLTGAE	
274	Z-SERVO POS OVER DIFF	
275	Z-SERVO OVERHEAT	
276	Z-SERVO SPEED SATURATION FAULT	
277	Z-SERVO FORBID FAULT	

279	Z-SERVO ENCODE FAULT	
280	Z-SERVO ENCODE ZERO TIMEOVER	
281	Z-SERVO IPM FAULT	
282	Z-SERVO OVERCURRENT	
283	Z-SERVO OVERLOAD	
284	Z-SERVO BRAKE FAULT	
286	Z-SERVO POWER OFF	
288	Z-SERVO VERSION ERROR	
289	Z-SERVO ENCODE CRC ERROR	
290	Z-SERVO EEPROM ERROR	

Warning and Precaution

Accident may occur by improper connection and operation！ This system can only be operated by authorized and qualified personnel.

Please read this manual carefully before operation！

Please read this manual and a manual from machine tool builder carefully before installation, programming and operation, and strictly observe the requirements.

This manual includes the precautions for protecting user and machine tool. The precautions are classified into Warning and Caution according to their bearing on safety, and supplementary information is described as Note. Read these Warnings, Caution and Note carefully before operation.

Warning

User may be injured or equipment be damaged if operations instructions and procedures are not observed.

Caution

Equipment may be damaged if operation instructions or procedures are not observed.

Note

It is used to indicate the supplementary information other than Warning and Caution.

Announcement

- This manual describes various possibilities as much as possible. However, operations allowable or unallowable cannot be explained one by one due to so many possibilities that may involve with, so the contents that are not specially stated in this manual shall be considered as unallowable.

Caution

- Functions, technical indexes (such as precision and speed) described in this user manual are only for this System. Actual function deployment and technical performance of a machine tool with this CNC system are determined by machine tool builder's design, so functions and technical indexes are subject to the user manual from machine tool builder.
- Refer to the user manual from machine tool builder for function and meaning of keys on control panel.

Precautions

■ Delivery and storage

- Packing box over 6 layers in pile is unallowed.
- Never climb the packing box, neither stand on it, nor place heavy objects on it.
- Do not move or drag the products by the cables connected to it.
- Forbid collision or scratch to the panel and display screen.
- Avoid dampness, insolation and drenching.

■ Open-package inspection

- Confirm that the products are the required ones.
- Check that the products are not damaged in delivery.
- Confirm that the parts in packing box are in accordance with the order.
- Contact us in time if any inconsistence, shortage or damage is found.

■ Connection

- Only qualified personnel can connect the System or check the connection.
- The System must be earthed, and the earth resistance must be less than 0.1Ω. The earth wire cannot be replaced by zero wire.
- The connection must be correct and firm to avoid any fault or unexpected consequence.
- Connect with surge diode in the specified direction to avoid damage to the System.
- Switch off power supply before plugging out or opening electric cabinet.

■ Troubleshooting

- Only competent personnel are supposed to inspect the System or machine.
- Switch off power supply before troubleshooting or changing components.
- Check for fault when short circuit or overload occurs. Restart can only be done after troubleshooting.
- Frequent switching on/off of the power is forbidden, and the interval time should be at least 1 min.

Safety Responsibility

Manufacturer's Responsibility

——Be responsible for the danger which should be eliminated and/or controlled on design and configuration of the provided CNC systems and accessories.

——Be responsible for the safety of the provided CNC systems and accessories.

——Be responsible for the provided information and advice for the users.

User's Responsibility

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

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

——Be responsible for the failure to observe the provisions for operation, adjustment, maintenance, installation and storage in the manual.

All specifications and designs herein are subject to change without further notice.

This manual is reserved by end user.

Contents

Ⅰ PLC PROGRAMMING

Part 1 Programming

1 Sequence Program Creating Process

1.1 GSK25i PLC specifications

Specification of GSK25i PLC are as follows(see Table 1-1):

Table 1-1

Specification	GSK25i PLC
Programming method language	Ladder, command table
Number of ladder level	2
1st level execution period	8ms
Mean processing time of basic command	0.5(μs/step)
Program capacity	12000 steps
Command	P：10 Functional command：44
Internal relay （R） Data table （D） Meter （C） Timer （T） PLC alarm detection （A） Keep relay （K） Label （L） Subprogram （P）	1100 bytes（R0 to R1099） 1860 bytes （D0 to D1859） 400 bytes （C0 to C399）100PCS 200 bytes （T0 to T199）100PCS 32 bytes(A0 to A31) 32 bytes(K0 to K31) 9999（L1～L9999） 512（P1～P512）
Machine →PLC(X) PLC→machine (Y) CNC→PLC(F) PLC→CNC(G)	128 bytes（X0 to X127） 128 bytes （Y0 to Y127） 256 bytes （F0 to F255） 256 bytes （G0 to G255）

1.2 What ’s a Sequence Program

A sequence program is a program for sequence control of machine tools and other systems.

The program is converted into a format to enable CPU execute encoding and arithmetic processing, and stored into RAM. CPU reads out every instruction stored in the memory at a high-speed and execute the program by arithmetic operation

The sequence program is written firstly from ladder.

1.3 Assignment of interface specifications（step 1）

After deciding the control object specification, calculate the number of input/output signal points, create the interface specification.

For input/output interface signals, see ***Chapter 4.***

1.4 Establishment of ladder diagram（step 2）

Express the control operations decided by 25i ladder diagram. For the timer, meter, etc, which cannot be expressed with the functional instructions.

The edited ladder should be converted into the corresponding PLC instruction i.e. instruction list to store.

1.5 Sequence program debugging（step 3）

The sequence program can be debugged in two ways:

1）Debug by simulator

Instead of the machine, connect a simulator (consisting of lamps and switches). Switch ON/OFF stands for the input signal state of machine, lamp ON/OFF for the output signal state.

2）Actual operation debugging

Debug sequence program through operating the machine. Do measures against the unexpected affairs before debugging.

2 Sequence Program

Since PLC sequence control handled by software and operates on principle difference from a general relay circuit, the sequence control method must be fully understood in order to design PLC sequence program.

2.1 Execution process of sequence program

In general relay control circuit, each relay operates at approximately the same time, in the figure below for example, when relay A operate, the relay D and E operate at approximately the same time(when contacts B and C are off)., In PLC sequence control, each relay of circuit operates sequentially. When relay A operates, relay D operates, then relay E(see Fig.2-1). Thus each relay operates in sequence which can be written as a ladder diagram. (Programmed sequence).

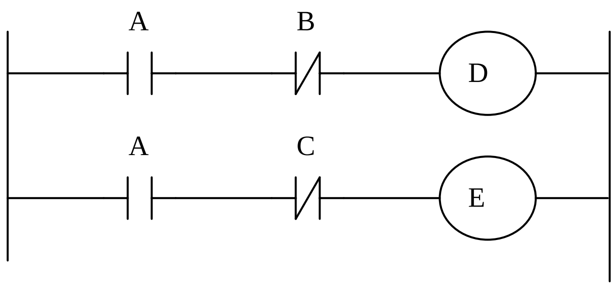


Fig. 2-1(a)

Fig.2.1(b) and (c) illustrate operations varying from the relay circuit to PLC programs.

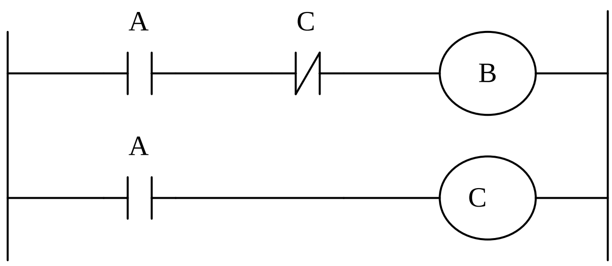


Fig. 2-1(b)

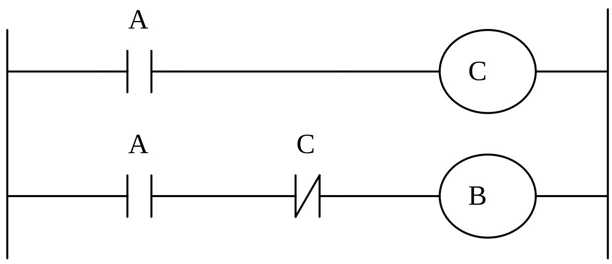


Fig.2-1(c)

(1) Relay circuit

In Fig. 2.1(b) and (c), the operations are the same. Turning on A turns on B and C. Turning on C turns off B.(2) PLC program

In Fig.2.1(b), as in the relay circuit, turning on A turns on B and C, and after one cycle of the PLC sequence, turns off B. But in Fig.2.1(c), turning on A turns on C, but does not turn on

2.2 Repetitive cycle

The PLC executes the ladder diagram from the beginning to the end . When the ladder diagram ends, the program starts over from the beginning. This is called repetitive operation.

The execution time from the beginning to the end of the ladder diagram is called the sequence processing time. The shorter the process time is, the better the signal response becomes.

2.3 Priority of execution(1st level, and 2nd level)

GSK25i PLC consists of two parts: 1st level sequence part, 2nd level sequence part. They have different execution period.

The 1st level sequence part operates every 8 ms, which can deal with the short pulse signal with high-speed response).

The 2nd level sequence part operates every 8∗n ms. Here N is a dividing number for the 2nd level sequence part. The 2nd level sequence part is divided into V part, and every part is executed every 8ms.

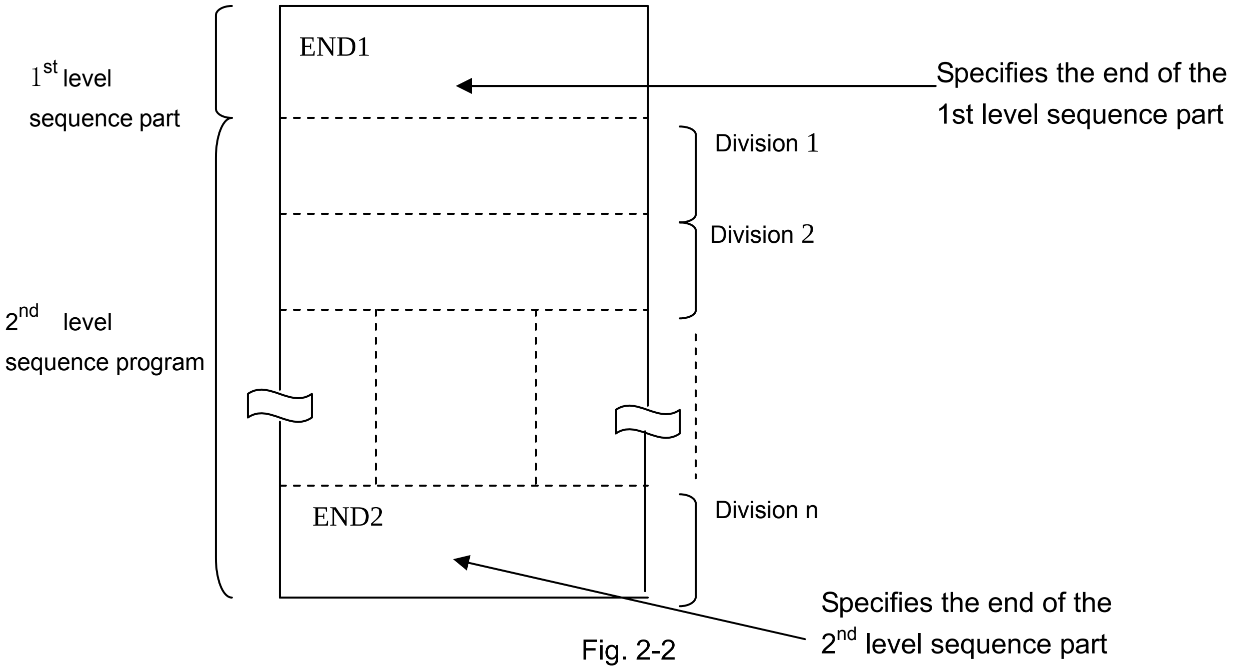


Fig. 2-2

GSK 25i PLC is solely executed in PLC-AVR single chip, and the first 1ms of each 8ms is the communication time of CNC reading or writing PLC data. The fifth 1ms is the time that the PLC receives the system control signal（F, X）and uploads the control result data（G, Y parameter）to the external I/O interface（X, Y）, except for the time responding the interruption to exchange the data, the PLC executes the ladder operation at the rest time.

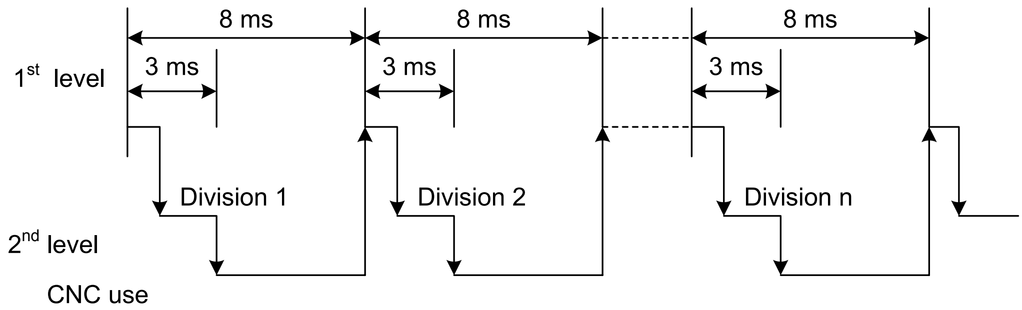


Fig. 2-3

After the last 2nd level sequence part (division n) is executed, the sequence program is executed again from the beginning. Thus, when the dividing number is n, the cycle of execution is 8*n ms. The 1st level sequence operates every 8ms, and the 2nd level sequence every 8*n ms. If the steps of the 1st level sequence is increased, the steps of the 2nd level sequence operating within 4ms becomes less, thereby increasing the dividing number and making the processing time longer. Therefore, it is desirable to program so as to reduce the 1st level sequence to a minimum.

2.4 Sequence program structure

With the conventional PLC, a ladder program is described sequentially. By employing a ladder language that allows structured programming, the following benefits are derived:

1. A program can be understood and developed easily
2. A program error can be found easily.
3. When an operation error occurs, the cause can be found easily.

Three major structured programming capabilities are supported:

1） Subprogram

A subprogram can consist of a ladder sequence as the processing unit.

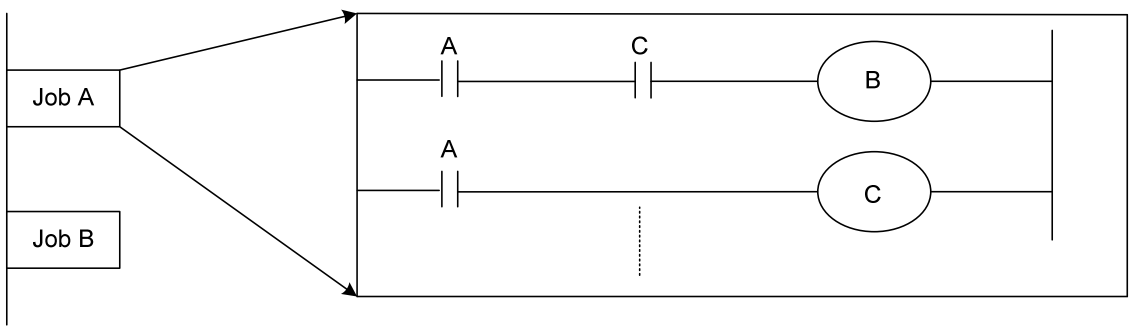


Fig. 2-4

2) Nesting

The Ladder subprograms can call the other ladder subprogram to execute the job.

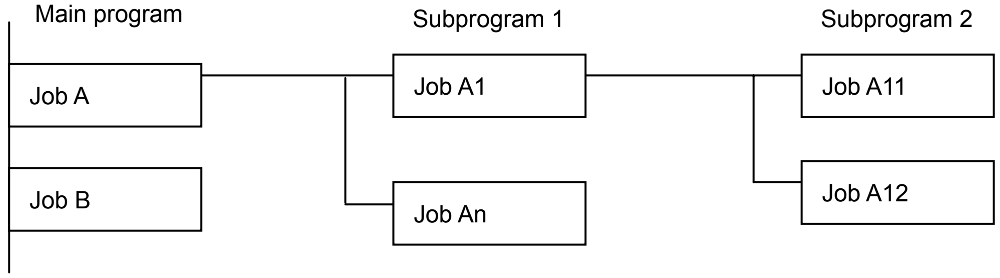


Fig. 2-5

3) Conditional branch

The main program loops and checks whether conditions are satisfied. If a condition is satisfied, the corresponding subprogram is executed. If the condition is not satisfied, the subprogram is jumped.

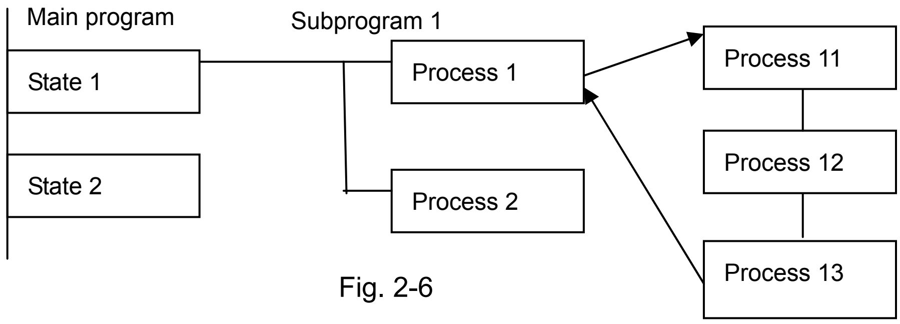


Fig. 2-6

2.5 Processing I/O (input/output) signals

Input signal processing:

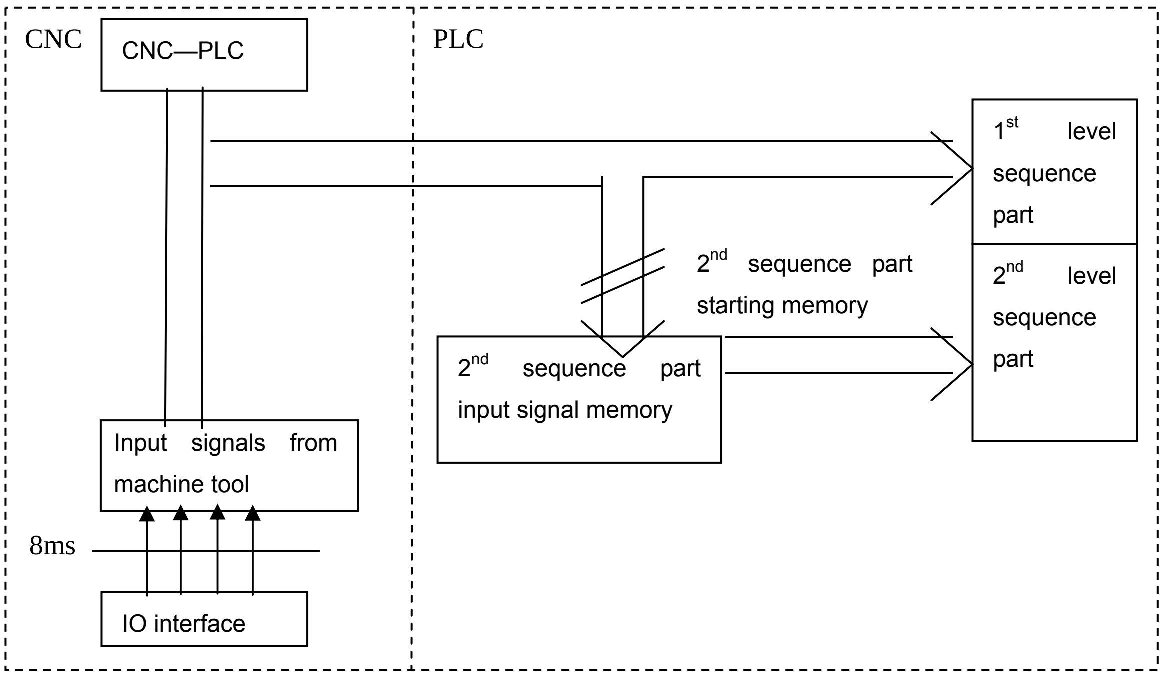


Fig. 2-7

Output signal processing:

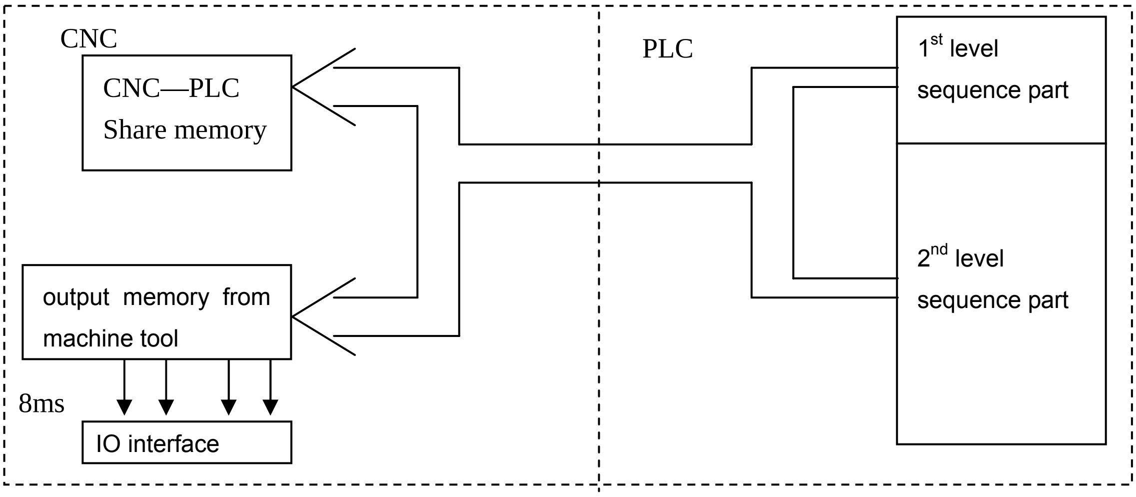


Fig. 2-8

2.5.1 Input signal processing

（1）Input memory of NC

The input signals from NC are loaded in memory of NC and are transferred to the PLC at intervals of 8ms. Since the 1st level sequence part directly refer to these signal and process operations.

（2）Input signal memory to machine tool

The input signal memory stores signals transferred from the machine tool at intervals of 8ms period. Since the 1st level sequence part directly refer to these signal and process operations.

（3）2nd level input signal memory

The 2nd level input signal memory is also called as 2nd level synchronous input signal memory. The stored signals are processed by the 2nd level sequence part. State of the signals set this memory synchronizes with that of 2nd level sequence part.

Input memory Signals from NC and machine tool are transferred to the 2nd level input signal memory only at the beginning of execution of the 2nd level sequence part. Therefore, the state of the 2nd level synchronous input signal memory does not change from the beginning to end of the execution of the 2nd level sequence part.

2.5.2 Output signal processing

（1）NC output memory

The output signals are transferred form the PLC to the NC output memory at intervals of 8ms.

（2）Output signals to machine tool

Output signal to the machine tool from PLC output signal memory to the machine tool at intervals of 8ms.

Note:

The state of the NC input memory, NC output memory, input signals from machine, input/output memory signals to machine can be checked by using the PC self-diagnosis function. The self-diagnosis number specified is the address number used by the sequence program.

2.5.3 Synchronous processing the short pulse signal

1st program can process the short pulse signal. When the short pulse signal change is less than 8ms, i.e.when the system executes the 1st program, the input signal state can change to cause the followings.

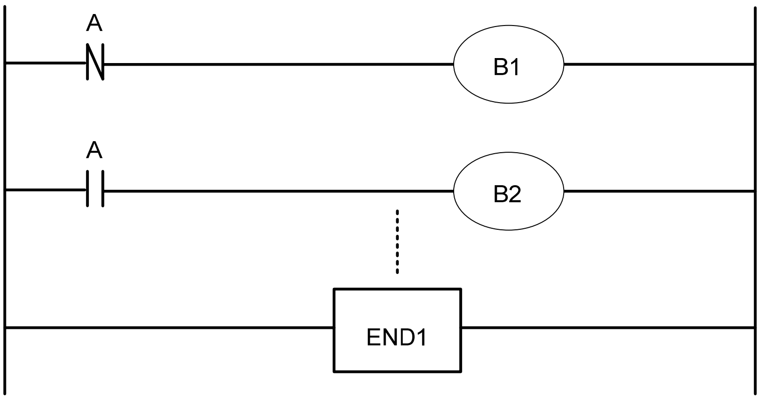


Fig. 2-9

When A＝0 and B1＝1，A becomes 1, at the moment, the system executes the next ladder statement to make B2＝1. so, B1 and B2 become 1.

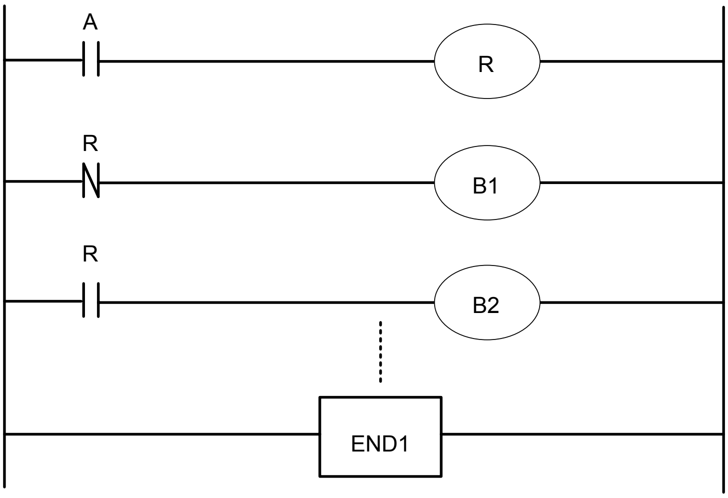


Fig. 2-10

When the medium relay R synchronously processes the signal A, B1, B2 are not 1 at the same time.

2.5.4 Difference state of signals between 1^{st} level and 2^{nd} level

The state of the same input signal may be different in the 1^{st} level and 2^{nd} level sequences. That is, at 1^{st} level, processing is performed using input signal memory and at 2^{nd} level, processing is performed using the 2^{nd} level synchronous input signal memory. Therefore, it is possible for a 2^{nd} level sequence execution at the worst, compared with a 1^{st} level input signal.
This must be kept in mind when writing the sequence program.

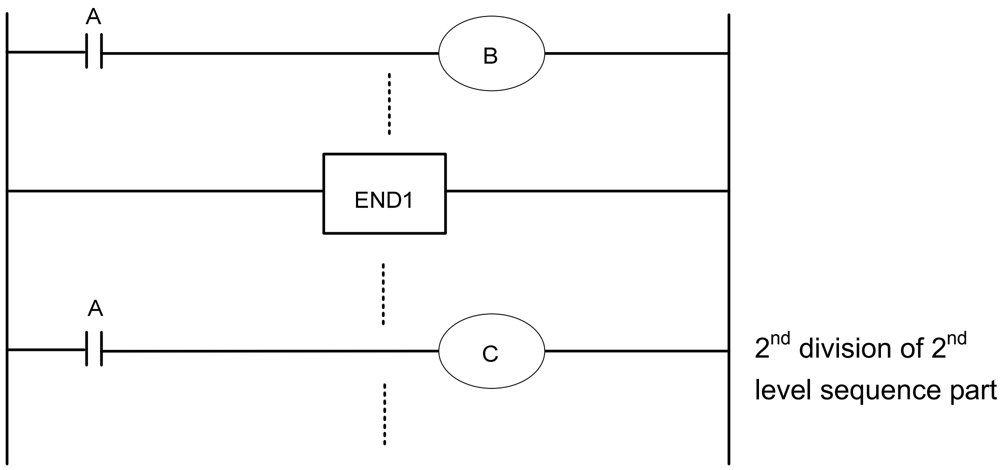


Fig. 2-11

When the processing is 1st 8ms, A=1, and B=1 after 1st sequence part is executed. At the same time, 2nd sequence part is started to execute A=1 is stored to the 2nd sequence part and the 1st division of 2nd sequence part is executed.

When the processing is 2nd 8ms, A=0, and B=0 after 1st sequence part is executed. And then 2nd division of 2nd sequence part is executed, at this time, A is still 1. So C=1.
So, B and C are different.

2.6 Interlocking

Interlocking is externally important in sequence control safety.

Interlocking with the sequence program is necessary. However, interlocking with the end of the electric circuit in the machine tool magnetic cabinet must not be forgotten. Even though logically interlocked with the sequence program (software), the interlock will not work when trouble occurs in the hardware used to execute the sequence program. Therefore, always provide an interlock inside the machine tool magnetic cabinet panel to ensure operator safety and to protect the machine from damage.

3 Address

An address shows a signal location. Addresses include input/output signals with respect to the machine, the input/output signals with respect to the CNC, the internal relays, the meters, the keep relays, and data table. Each address consists of an address number and a bit number. Its serial number regulations are as follows:

Address regulations:

The address comprises the address type, address number and the bit number in the format as shown below:

X	000.	6
Type	Address number	Bit number

Type: including X, Y, R, F, G K, A, D ,C, T

Address number: decimal serial number stands for one byte.

Bit number: octal serial number, 0～7 stands for 0～7 bit of byte of front address number

GSK25i PLC address type is as follows Fig.3-1：

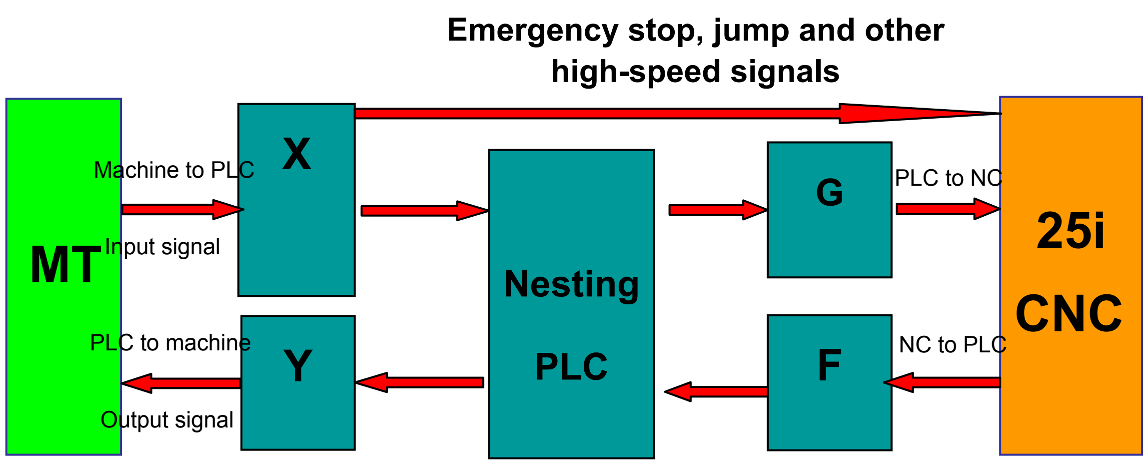


Fig. 3-1

Table 3-1

Address	Address explanation	Address range
X	machine→PLC(128 bytes)	X0～X127
Y	PLC→machine(128 bytes)	Y0～Y127
F	CNC→PLC(256 bytes)	F0～F255
G	PLC→CNC(256 bytes)	G0～G255
R	Internal relay(1100 bytes)	R0～R1099
D	Data register(1860 bytes)	D0～D1859
C	Counter (400 bytes)	C0～C 399
T	Timer (200 bytes)	T0～T199
A	Timer preset data register (32 bytes)	A0～A31
K	Keep relay（32 bytes）	K0～K31

3.1 Machine →PLC address（X）

X addresses of GSK25i PLC are divided into two:

1. X addresses are assigned to IO input interface.
2. X addresses are assigned to the input press keys on MDI panel.
3. X addresses are assigned to other external interfaces, such as the spindle, MPG control signal input.

3.1.1 Assignment of IO module X address

The addresses are from X9 to X119. Its type is INT8U, 111 types.

The signal specification of X addresses can be customized by customer according to the actual operation. X addresses are used to connect the machine tool with the ladder. For the initial definition of input address, see ***Chapter Four Connection.***

3.1.2 Assignment of MDI panel X address

The addresses are from X0 to X8, 9 bytes. They correspond to the press keys on MDI panel. The corresponding relationship between them and the press keys on the standard panel is as Fig. 3-2:

Table 3-2

INPUT KEY ON OPERATION PANEL	PLC ADDRESS	INPUT KEY ON OPERATION PANEL	PLC ADDRESS
Auto mode	X0.0	-Z	X3.5
Edit mode	X0.1	-4	X3.6
MDI mode	X0.2	-5	X3.7
Manual mode	X0.3	Spindle CW	X4.0
MPG mode	X0.4	Spindle stop	X4.1
Zero mode	X0.5	Spindle CCW	X4.2
DNC mode	X0.6	Spindle orientation	X4.3
USER1	X0.7	F0 / 0.001	X4.4
Single block	X1.0	25% / 0.01	X4.5
Jump	X1.1	50% / 0.1	X4.6
Machine lock	X1.2	100% / 1	X4.7
Auxiliary lock	X1.3		
+4	X1.4		
+Z	X1.5		
-Y	X1.6	Tool magazine infeed	X5.3
+5	X1.7	Tool retraction	X5.4
Dry run	X2.0	Tool change manipulator	X5.5
Overtravel release	X2.1	Tool magazine CW	X5.6
Optional stop	X2.2	Tool magazine zero	X5.7
Program restart	X2.3	Clamp/release	X6.0
+X	X2.4	USR2	X6.1
Rapid	X2.5	USR3	X6.2
Step	X2.6	USR4	X6.3
-X	X2.7	Feed hold	X6.4
Cooling	X3.0	Cycle start	X6.5
Lubricating	X3.1	Tool magazine CCW	X6.6
Chip removal	X3.2	Feedrate override, up to 24-gear(no output light)	X7.0-X7.4
Working light	X3.3	Spindle override, up to 16-gear（no output light）	X8.0-X8.3
+Y	X3.4	Emergency stop	X8.4

3.1.3 MPG signal input X address

Table 3-3

MPG signal input	PLC address
HDC0_STP （MPG emergency stop signal）	X121.0
HDC0_MX100（MPG federate override）	X120.0
HDC0_MX10 （MPG federate override）	X120.1
HDC0_MX1 （MPG federate override）	X120.2
HDC0_5 （5^{th} axis）	X120.3
HDC0_4 （4^{th} axis）	X120.4
HDC0_Z （Z axis）	X120.5
HDC0_Y （Y axis）	X120.6
HDC0_X （X axis）	X120.7

3.2 PLC→machine side address（Y）

Y addresses of GSK25i PLC are divided into three:

1. Y addresses are assigned to IO input interface.
2. Y addresses are assigned to the indicators on MDI panel.
3. Y addresses are assigned to the indicators on MPG.

3.2.1 Y address of I/O output interface

The addresses are from Y8 to Y119. Its type is INT8U, 112 types.

The signal specification of Y addresses can be customized by customer according to the actual operation. Y addresses are used to connect the machine tool with the ladder. For the initial definition of input address, see *Chapter Four Connection*.

3.2.2 Assignment of IO module Y address

The addresses are from Y0 to Y7, 8 bytes. They correspond to the indicators on MDI panel, and their signal definitions cannot be changed by user.

Addresses and indicators are as follows Table.3-4:

Table 3-4

OUTPUT KEY ON OPERATION PANEL	PLC ADDRESS	OUTPUT KEY ON OPERATION PANEL	PLC ADDRESS
Auto key indicator	Y0.0	-Z key indicator	Y3.5
Edit key indicator	Y0.1	-4 key indicator	Y3.6
MDI key indicator	Y0.2	-5 key indicator	Y3.7
Manual key indicator	Y0.3	Spindle CW key indicator	Y4.0
MPG key indicator	Y0.4	Spindle stop key indicator	Y4.1
Zero key indicator	Y0.5	Spindle CCW key indicator	Y4.2
DNC key indicaor	Y0.6	Spindle orientation key indicator	Y4.3
USER1 key indicaor	Y0.7	F0 / 0.001 key indicator	Y4.4
Single block key indicaor	Y1.0	25% / 0.01 key indicator	Y4.5
Jump key indicator	Y1.1	50% / 0.1 key indicator	Y4.6
Machine lock indicator	Y1.2	100% / 1 key indicator	Y4.7
Auxiliary lock indicator	Y1.3	Tool magazine infeed key indicator	Y5.3
+4 key indicator	Y1.4	Tool retraction key indicator	Y5.4
+Z key indicator	Y1.5	Tool change key indicator	Y5.5
-Y key indicator	Y1.6	Tool magazine CW key indicator	Y5.6
+5 key indicator	Y1.7	Tool magazine zero key indicator	Y5.7
Dry run key indicator	Y2.0	Clamp/release tool key indicator	Y6.0
Overtravel release key indicator	Y2.1	USR2 key indicator	Y6.1
Optional stop key indicator	Y2.2	USR3 key indicator	Y6.2
Program restart key indicator	Y2.3	USR4 key indicator	Y6.3
+X key indicator	Y2.4	Feed hold key indicator	Y6.4
Rapid key indicator	Y2.5	Cycle start key indicator	Y6.5
Step key indicator	Y2.6	Tool magazine CCW key indicator	Y6.6
-X key indicator	Y2.7	X zero return indicator	Y7.0
Cooling key indicator	Y3.0	Y zero return indicator	Y7.1
Lubricating key indicator	Y3.1	Z zero return indicator	Y7.2
Chip removal key indicator	Y3.2	4^{th} zero return indicator	Y7.3
Working light key indicator	Y3.3	5^{th} zero return indicator	Y7.4
+Y key indicator	Y3.4	System alarms	Y7.6

3.2.3 MPG signal light output

MPG signal light output	Y120.0

3.3 PLC→CNC address（G）

Addresses are from G0 to G255. Type: INT8U,256 bytes. G addresses are the signals from PLC to NC, and these signals have been defined in designing the CNC system and cannot be modified.

The concrete is referred to Appendix 1.

3.4 CNC→PLC address（F）

A ddresses are from F0 to F255. Type: INT8U,256 bytes. F addresses are the signals from NC to PLC, and these signals have been defined in designing the CNC system and cannot be modified. The concrete is referred to Appendix 1.

3.5 Internal relay address（R）

The address area is cleared to zero when the power is turned on.
Type: INT8U, with 1100 bytes.

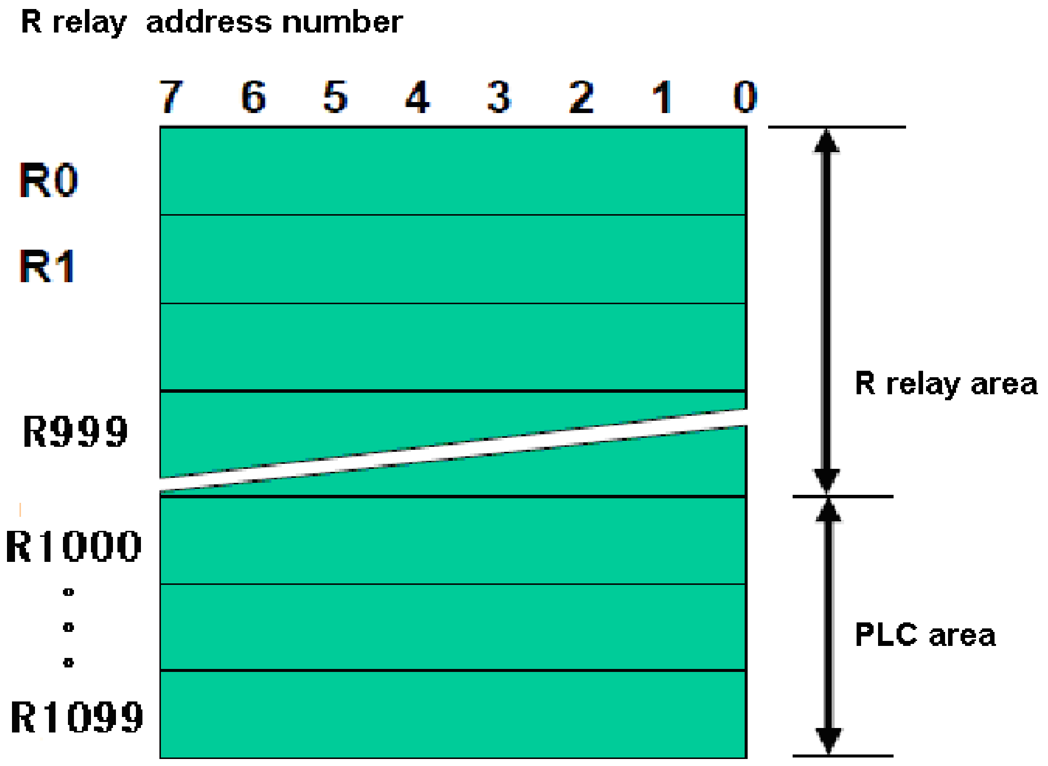


Fig. 3-2

Note: the addresses from R1000 are used by PLC. For example: ADDB, SUBB, COMB functional command operation result are output to the register:

	#7	#6	#5	#4	#3	#2	#1	#0
R1000		Overflow					Negative	Zero

Fig. 3-3

3.6 Address of keep relay（K）

The area is used for the keep relays and PLC parameters. Since this area is nonvolatile, the content of the memory do not disappear even when the power is turned off.

Type: INT8U, with 32 bytes

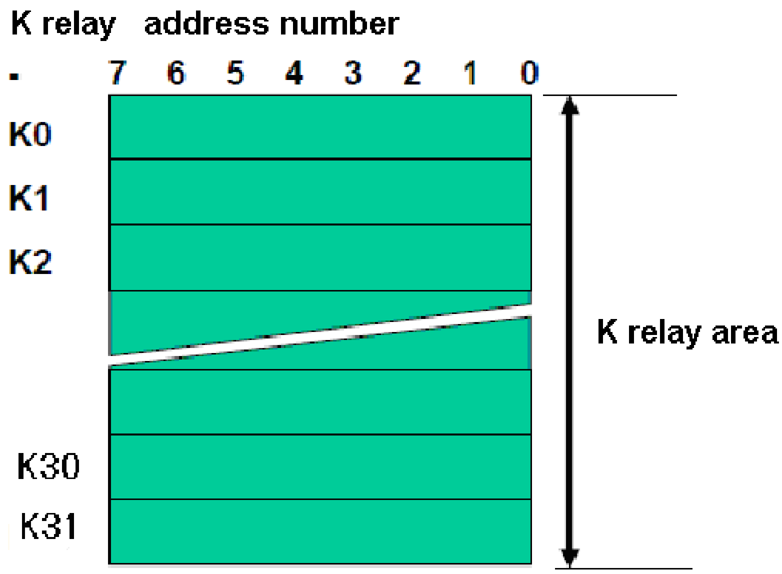


Fig. 3-4

3.7 Addresses(A) for message selection

The address area is cleared to zero when the power is turned on.
Type: INT8U, with 32 bytes.

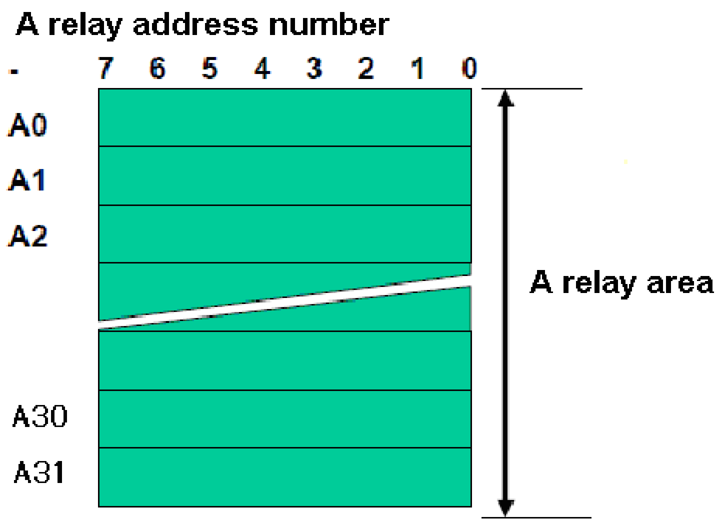


Fig. 3-5

3.8 Address of counter（C）

The area is used as storing current counting value in meter.

Type: 400 bytes.

C1～C100: count range: 0～65535, can set increase/reducing count, and the counting value does not disappear even when the power is turned off.

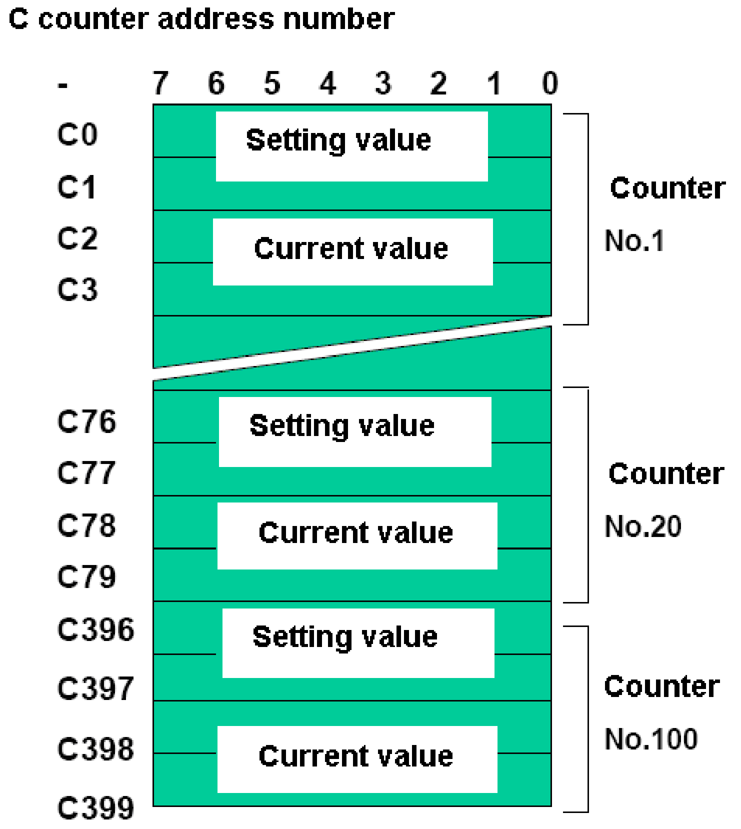


Fig. 3-6

3.9 Address of timer （T）

Type: 200 bytes.

T1～T100，The timing value does not disappear even when the system is turned off.

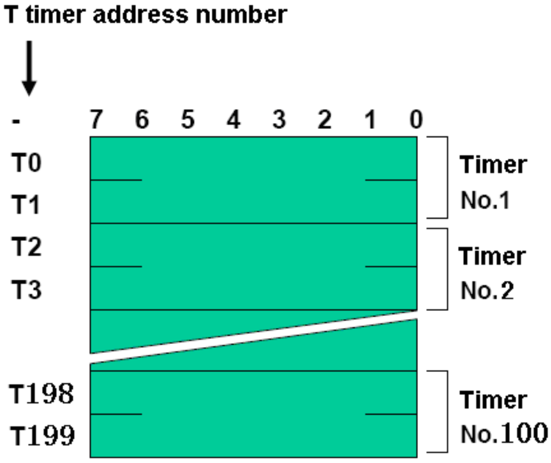


Fig. 3-7

3.10 Address（D）of data table

Each data register has 8-bit, two continuous data registers can store 16-bit data, four continuous data registers can store 32-bit data.

The content of the memory do not disappear even when the power is turned off.

Number of data table：D0～D1859，1860 bytes.

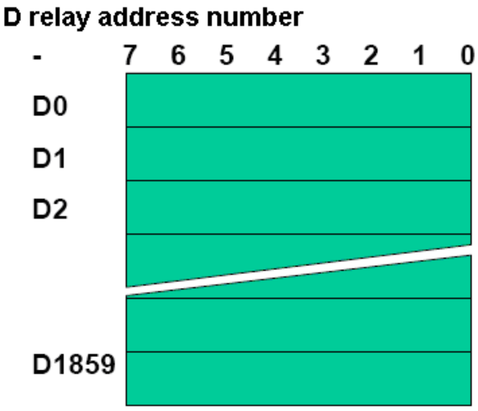


Fig. 3-8

3.11 Label address（L）

Label addresses are used to specify jump destination labels and LBL labels in JMPB instructions.
Range: L0～L9999

3.12 Subprogram numbers（P）

Subprogram numbers are used to specify jump destination subprogram labels and SP instruction subprogram labels in CALL instruction.
Range: P0～P511.

4 PLC Basic Instruction

Designing a sequence program begins with writing a ladder diagram. The ladder diagram is written using relay contact symbols and functional instruction code. Logic written in the ladder diagram is entered as a sequence program in the Programmer. There are two sequence program entry methods. One is the entry method with the mnemonic language (PLC instructions such as RD, AND, OR). The other is the relay symbols of the ladder diagram. When the relay symbol method is used, the ladder diagram format can be used and programming can be performed without understanding the PLC instruction format.

Actually, however, the sequence program entered by the relay symbol method is also internally converted into the instruction corresponding to the PLC instruction.

The basic instructions are often used when the sequence program is designed, and the execute one-bit operation.

GSK25i basic instructions are as follows(see Table 4-1):

Table 4-1

Instruction	Function
LD	Shifts left the content by one bit in register and sets the state of a specified signal in ST0.
LDI	Shifts left the content by one bit in register and sets the logic state of a specified signal in ST0.
OUT	Outputs the results of logic operation to a specified address.
OUTI	Inverts the results of logical operations and output it to a specified address.
AND	Induces a logical product.
ANI	Inverts the state of a specified signal and induces a logical product.
OR	Induces a logical sum.
ORI	Inverts the state of a specified signal and induces a logical sum.
ORB	Sets the logical sum of ST0 and ST1, and shifts the stack register right by one bit.
ANB	Sets the logical product of ST0 and ST1, and shifts the stack register right by one bit.

4.1 LD, LDI, OUT, OUTI command

Instructions and functions（Table 4-2）:

Table 4-2

Instruction	Function
LD	Shifts left the content by one bit in register and sets the state of a specified signal in ST0.
LDI	Shifts left the content by one bit in register and sets the logic state of a specified signal in ST0.
OUT	Outputs the results of logic operation to a specified address.
OUTI	Inverts the results of logical operations and output it to a specified address.

Instruction specifications:

- WRT, WRT. NOT are the output relay, internal relay instructions. They cannot be used to input relay.
- The parallel WRT instruction can be continuously used many times.

Programming

LD X002.1
OUT Y003.7
LDI F100.3
OUT G120.0

4.2 AND, ANI command

Instructions and functions（Table 4-3）:

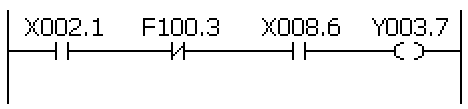
Table 4-3

Instruction	Function
AND	Induces a logical product.
ANI	Inverts the state of a specified signal and induces a logical product.

Instruction specifications:

- AND, ANI can connect with one contact in serial. The serial contact numbers are not limited and they can be used many times.

Programming



LD X002.1
ANI F100.3
AND X008.6
OUT Y003.7

4.3 OR, ORI command

Instructions and functions (Table 4-4)

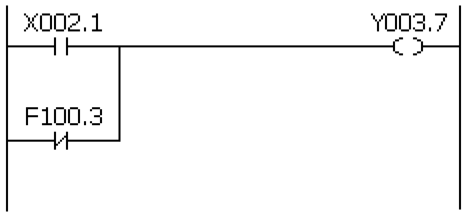
Table 4-4

Instruction	Function
OR	Induces a logical sum.
ORI	Inverts the state of a specified signal and induces a logical sum.

Instruction specification:

- OR, ORI can connect with one contact in parallel.
- OR, ORI begins from their step, which can connect with the mentioned step in parallel.

Programming:



LD X002.1
ORI F100.3
OUT Y003.7

4.4 ORB command

Instruction and function(Table 4-5):

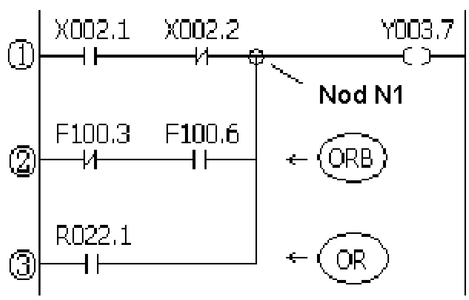
Table 4-5

Instruction	Function
ORB	Sets the logical sum of ST0 and ST1, and shifts the stack register right by one bit.

Instruction specification:

- ORB a sole instruction without other address.

Programming



LDI F100.3
AND F100.6
ORB
OR R022.1
OUT Y003.7
LD X002.1
ANI X002.2

As the above figure, there are three branch circuit ①，②，③ from left bus to the node N1, among which ①，② is circuit block in series; when there is the serial circuit block in the parallel from the bus to node or between nodes, the following branch end uses LD instruction except for the first branch.

The branch ③ is not serial circuit block to use OR instruction.

ORB and ANB are instructions without operation components, indicating the OR, AND relationship between circuit blocks.

4.5 ANB command

Instruction and function（Table 4-6）:

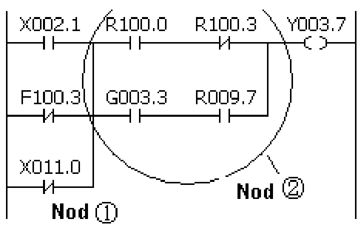
Table 4-6

Instruction	Function
ANB	Sets the logical product of ST0 and ST1, and shifts the stack register right by one bit.

Instruction specification

- When the branch loop (parallel loop block) is connected to the previous loop in series, use ANB instruction. The starting point of branch uses LD, LDI instruction, after the parallel loop block ends, ANB instruction is connected to previous loop in series.
- ANB a sole instruction without other address.

Programming




```
LD    X002.1
ORI   F100.3
ORI   X011.0
LD    R100.0
ANI   R100.3
LD    G003.3
AND   R009.7
ORB                 ←(1)
ANB                 ←(2)
OUT   Y003.7
```

As the above figure and instruction list, ⑴ORB reportS the series circuit block in block ②is connected parallel ⑵ANB reports the block ① and ② are connected in series.

5 PLC Functional Instructions

Basic instructions such as controlling operations of machine tool are difficult to program, therefore, functional instructions are available to facilitate programming.

25i functional instruction as follows(Table 5-1):

Table 5-1

No.	Instruction	Processing
0	END1	End of a 1st level ladder program
1	END2	End of a 2nd level ladder program
2	TMR	Timer processing
3	TMRB	Fixed timer processing
4	TMRC	Timer processing
5	DECB	Binary decoding
6	CTR	Counter processing
7	CTRC	Counter processing
8	ROTB	Binary rotation control
9	CODB	Binary code conversion
10	MOVE	Data transfer after logic AND
11	MOVOR	Data transfer after logic OR
12	MOVB	Transfer of 1 byte
13	MOVW	Transfer of 2 bytes
14	MOVN	Transfer of an arbitrary number of bytes
15	PARI	Parity check
16	DCNVB	Data conversion
17	COMPB	Binary comparison
18	COIN	Coincidence check
19	DSCHB	Binary data search
20	XMOVB	Binary indexed data transfer
21	ADDB	Binary addition
22	SUBB	Binary subtraction
23	MULB	Binary multiplication
24	DIVB	Binary division
25	NUMEB	Binary constant definition
26	DIFU	Edge Up detection

27	DIFD	Failing edge detection
28	SFT	Register shift
29	EOR	Exclusive OR
30	AND	Exclusive AND
31	OR	Exclusive OR
32	NOT	Logic NOT
33	COM	Common line control
34	COME	End of common line control
35	JMP	Jump
36	JMPE	End of a jump
37	CALL	Conditional subprogram call
38	CALLU	Unconditional subprogram call
39	JMPB	Label jump
40	JMPC	Label jump
41	LBL	Label
42	SP	Subprogram
43	SPE	End of a subprogram

5.1 END1（1st level sequence program end）

Function:

It must be specified once in a sequence program, either at the end of the 1st level sequence, or at the beginning of the 2nd level sequence when there is no 1st level sequence. It can write 500 steps.

Format:

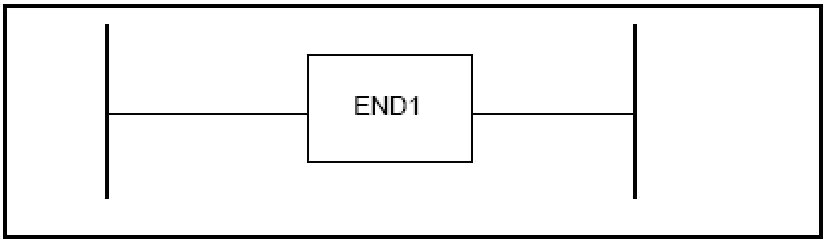


Fig. 5-1

Command table format:

Table 5-2

No.	Command	Operand	Remark
1	FUNC	0	End of 1st level program

5.2 END2（2^{nd} level sequence program end）

Function:

Specify at the end of 2^{nd} level sequence.

Format:

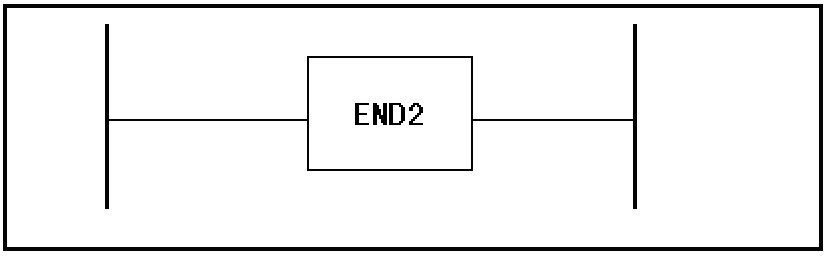


Fig. 5-2

Command table

Table 5-3

No.	Command	Operand	Remark
1	FUNC	1	End of 2^{nd} level program

Note: Only the subprograms of SP head, SPE end are added to the ladder following END2, otherwise, the system prompts the wrong.

5.3 TMR（Timer）

Function:

This is an on-delay timer.

Format:

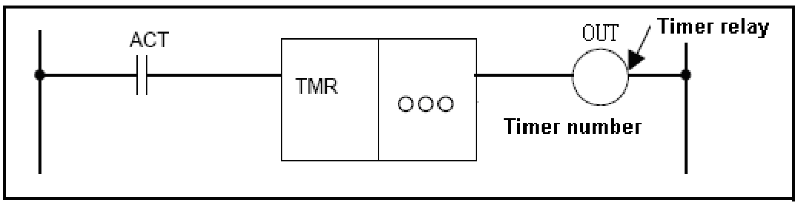


Fig. 5-3

Command table format:

Table 5-4

No.	Command	Operand	Remark
1	LD	○○○○. ○	Exclusive conditions
2	FUNC	2	Timer command TMR
3	PRM	○○○	Timer number
4	OUT	○○○○. ○	Timer relay

Control conditions： ACT＝0，turns off timer relay.

ACT＝1，start TIMER.

Concrete working conditions are as follows:

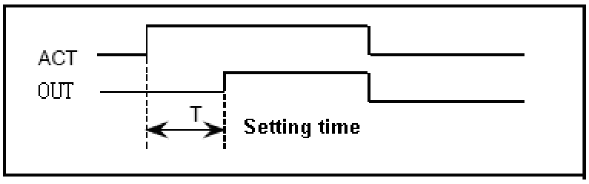


Fig. 5-4

Parameter：

Timer number：reports with ○○○, ○○○ are the number(1～100).

Output：

OUT ： timer relay.

OUT =1 ACT processing is done and reaches the preset time, the timer relay processing is done, OUT =1.

OUT =0 ACT processing is not done or has not reached the preset time, the timer relay is turned off, OUT =0.

Setting timer：

For timer TMR delay time setting value, 1st -20th timer take 48ms as the unit setting, and the maximum setting value is 3145680ms; when the value less than 48ms is omitted; 21st to 100th timer take 8ms as the unit setting and the maximum setting value is 524280ms, and the value less than 8ms is omitted.

For example: when the 1st timer value is 100ms, the set actual value is 96ms, 100=48×2+4 and the remainder 4 is omitted.

5.4 TMRB（fixed timer）

Function：

The timer is used as a fixed on-delay timer.

Format：

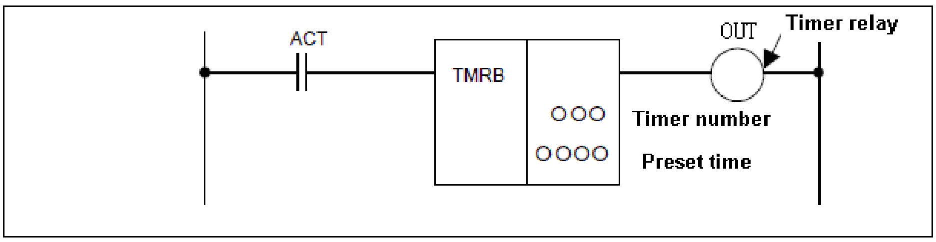


Fig. 5-5

Command table format:

Table 5-5

No.	Command	Operand	Remark
1	LD	○○○○. ○	Exclusive conditions
2	FUNC	3	Fixed timer TMRB
3	PRM	○○○	Timer number
4	PRM	○○○○	Timer time
5	OUT	○○○○. ○	Timer relay

Control condition:

ACT=0: turn off timer relay.

ACT=1: start timer.

Parameter:

Timer number set timer number of the fixed timer (1~100).

Timer time setting preset time (set delay time 8ms~999999ms)

The range of the preset time is 8ms nd the remainder is omitted. For example: the preset is 38ms, 38==8*4+6, and the remainder is discarded and the actual setting time is only 32ms.

Timer relay:

OUT : timer relay.

OUT=1 ACT processing is done and reaches the preset time, the timer relay processing is done, OUT=1.

OUT=0 ACT processing is not done or has not reached the preset time, the timer relay is turned off, OUT=0.

Note: TMR timer number can set the timer parameter to be modified, and it is saved when power-off; the fixed timer number of TMRB is a timer parameter directly processed in the system internal, is saved when power off, and cannot be modified by the user.

5.5 TMRC (timer)

Function:

TMRC is the on-delay timer using the address to set the fixed time. The processing data type is the binary data.

Format:

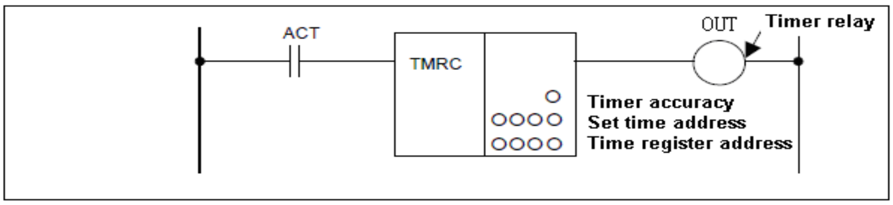


Fig. 5-6

Command table format:

Table 5-6

No.	Command	Operand	Remark
1	LD	○○○○. ○	Exclusive conditions
2	FUNC	4	TMRC command
3	PRM	○	Timer precision
4	PRM	○○○○	Timer time address
5	PRM	○○○○	Time register
6	OUT	○○○○. ○	Timer relay

Control condition:

ACT=0: turns off the timer relay.

ACT=1: starts the timer.

Parameter:

Timer precision: timer precision, parameter setting value, setting time and error are as follows:

Table 5-7

Timer accuracy	Setting value	Setting time	Timer accuracy error
8 ms	0	8 ms to 52428 ms	0 to ±8ms
48 ms	1	48 ms to 3145680 ms	0 to ±8ms
1s	2	1s to 65535 s	0 to ±8ms

Setting time address: the first address of the timer set time filed.

Timer register address: the first address of a specified continuous four-byte R is used as the system working area and is used in timer working.

Timer relay:

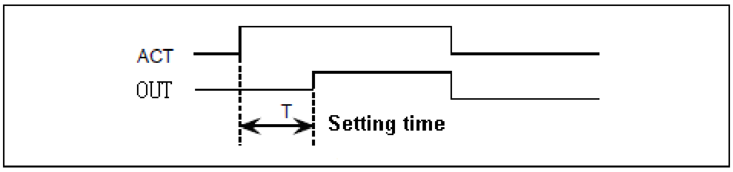


Fig. 5-7

OUT : timer relay.

OUT =1 ACT processing is done and reaches the preset time, the timer relay processing is done,, OUT =1.

OUT =0 ACT processing is not done or has not reached the preset time, the timer relay is turned off, OUT =0.

5.6 DECB（binary decode）

Function：

DECB decodes the binary data with 1, 2, 4 bytes, the corresponding output data is 1 when one of the specified 8-digit continuous data is equal to the code data, and 0 when not.

The command is used to decode M or T function.

Format：

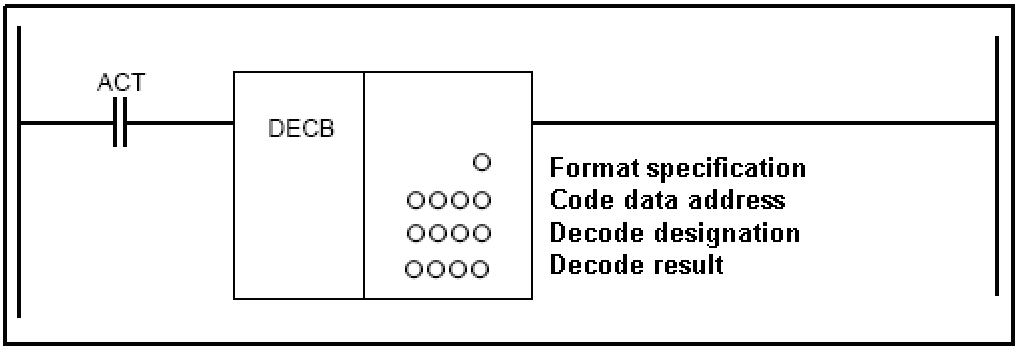


Fig. 5-8

Control condition：

ACT＝0：resets all the output data bits.

ACT＝1：decodes data. Results of processing is set in the output data address.

Command table format：

Table 5-8

No.	Command	Operand	Remark
1	LD	○○○○. ○	Control condition
2	FUNC	5	DECB command
3	PRM	○	Format specification
4	PRM	○○○○	Code data address
5	PRM	○○○○	Decode designation
6	PRM	○○○○	Decode output address

Parameters：

Format specification: Set the size of code data to the 1st digit of the parameter.

0001：code data is in binary format of 1-byte length.

0002：code data is in binary format of 2-byte length.

0004：code data is in binary format of 4-byte length.

Code data address: specify an address of a memory code data.

Decoding designating: designate the first number of the decoding 8 continuous codes.

Decoding result address: designate an address of the output decoding result covering 1-byte. The decoding result of the designated number is output to the 0-digit of the address, and the decoding result of the specified number +1 is output to 1-digit and the continuous 8 numbers are done like this.

Example:

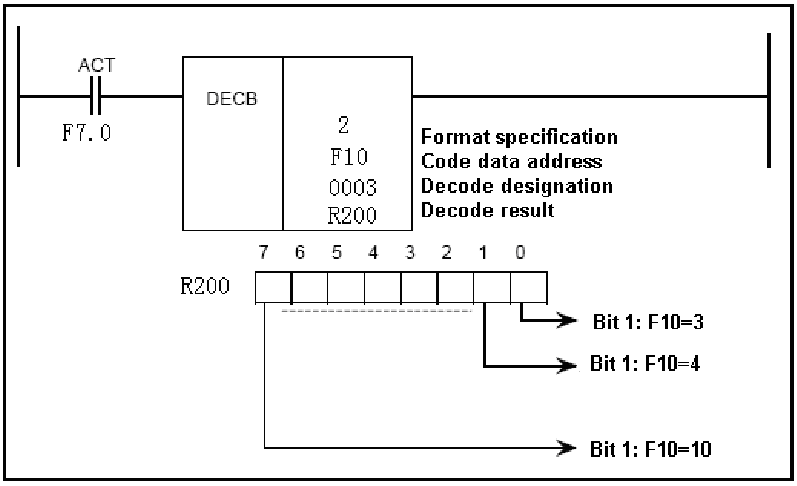


Fig. 5-9

After F7.0 is turned on, 2-byte data of F10～F11 are decoded. When the decoding data is in the range 3～10, the corresponding bit of R200 becomes 1.

5.7 CTR（counter）

Function:

The counter data type is the binary format and has the following functions to meet its application.

1) Preset counter

   Output a signal when the preset count is reached.

2) Ring counter

   Upon reaching the preset count, returns to the initial value by issuing another count signal.

3) Up/down counter

   The count can be either up or down.

4) Selection of initial value

   Select the initial value as either 0 or 1.

Format：

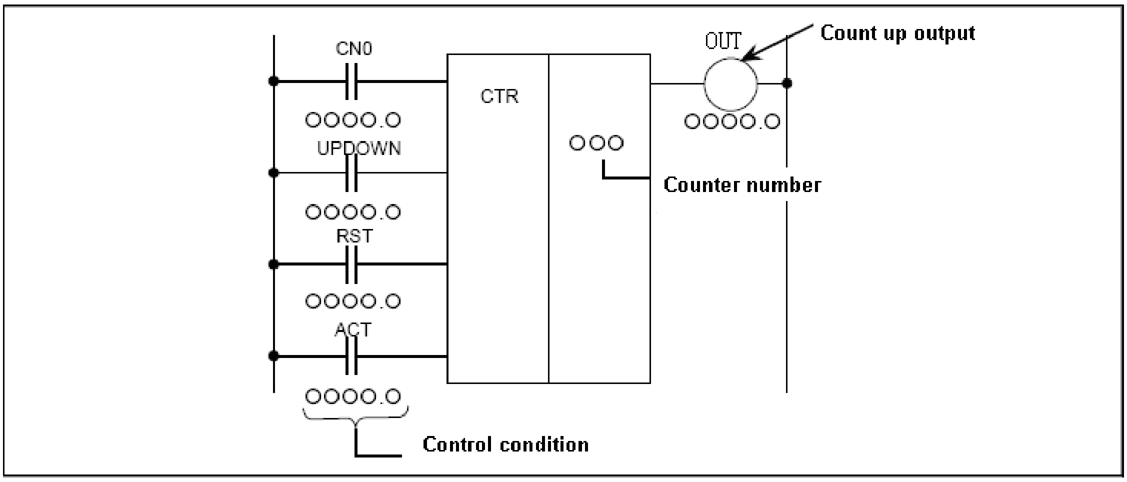


Fig. 5-10

Command table format：

Table 5-9

No.	Command	Operand	Remark
1	LD	○○○○. ○	CN0
2	LD	○○○○. ○	UPDOWN
3	LD	○○○○. ○	RST
4	LD	○○○○. ○	ACT
5	FUNC	6	CTR
6	PRM	○○○○	Counter number
7	OUT	○○○○. ○	Count up output

Control conditions：

CN0： Specify the initial value

CN0=0 begins the value of the counter with 0.

CN0=1 begins the value of the counter with 1.

UPDOWN： specify up or down counter:

UPDOWN=1 Up counter（the initial value is set by CN0）.

UPDOWN=0 Down counter(the counter begins with te preset value).

RST ： reset

RST=0 Releases reset.

RST=1 Enables reset. When OUT is reset to 0, the counter value is reset to the initial value(when the Up counter is done, it is 0 or 1 accoridng to CN0 setting), when it is Down counter, it is the preset value of the counter).

ACT ： Counter signal

ACT=1：counter is madeby catching the rise of ACT.

ACT＝0：counter does not operate. OUT does not change.

Parameter:

Counter number ： specify the counter number and it is 1～100.

Output:

OUT ： when the count is up to a preset value, the Up count reaches the maximum value or the minmum value, OUT = 1.

Note: When the counter is Up edge, the system executes the count. When the count number is repetitive, the operation is unexpected.

The current, preset value of the counter is set in 【Counter】 of 【PLC parameter】 in PLC window.

5.8 CTRC（counter）

Function:

The data in the counter is binary and the counter has the following functions.

1） Preset counter

Preset the count value and if the count reaches this preset value, outputs to show that.

2) Ring counter

This is the ring counter which is reset to the initial value when the count signal is input after the count reaches the preset value.

3) Up/down counbter

This is the reversible counter to be used as both the up counter and down counter.

4） Selection of the initial value

Either 0 or 1 can be selected as the initial value.

Format:

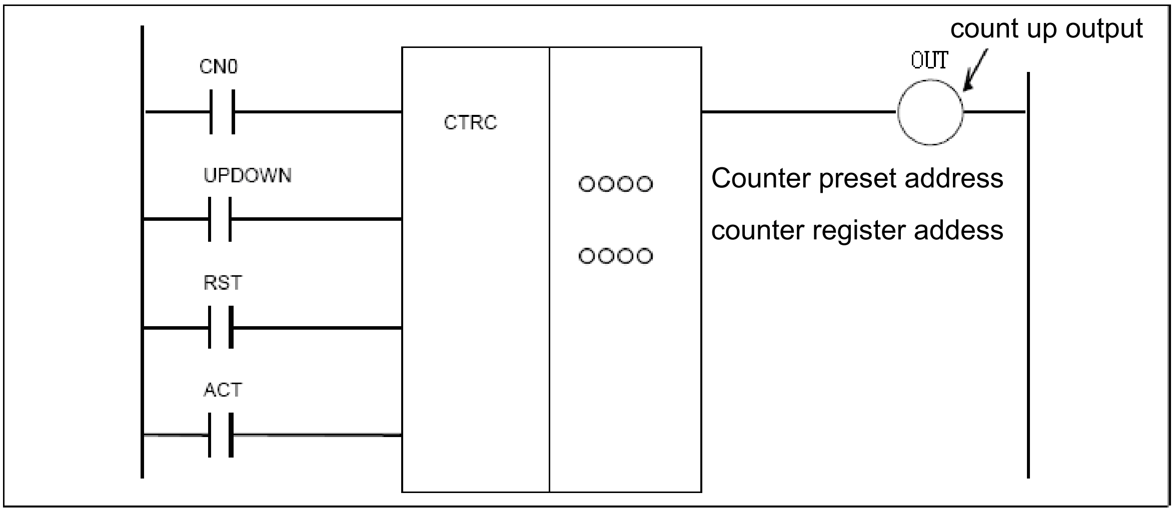


Fig. 5-11

Command table format：

Table 5-10

No.	Command	Operand	Remark
1	LD	○○○○. ○	CN0
2	LD	○○○○. ○	UPDOWN
3	LD	○○○○. ○	RST
4	LD	○○○○. ○	ACT
5	FUNC	7	CTRC command
6	PRM	○○○○	Counter preset address
7	PRM	○○○○	Counter register address
8	OUT	○○○○. ○	Count up output

Control conditions：

CN0 ： Specifying the initial value

CN0=0 the count value starts with 0.

CN0=1 the count value starts with 1.

UPDOWN ：Spcifying up or down counter

UPDOWN=1 Up counter.

UPDOWN=0 Down counter.

RST ： reset

RST=0 release reset.

RST=1 enable reset. When OUT is set to 0 the count value is reset to the initial value.

ACT ： count signal

ACT=1：the counter operates at the rise of this signal.

ACT＝0：the counter does not operate, OUT does not change.

Parameter：

Counter preset value address: the first address of the counter preset value field with 2-byte is set. The continuous 2-byte memory space from the first address is required for this field and the field D is binary and its range is 0～32767.

Counter register address: The first address of the counter register field is set, the continuous 4-byte memory space from the first address is required for this field and the field D is normally used. The first two-byte is accumulated value and the second two –byte is the system working area.

Note: When field R is specified as the counter register address,the counter starts with count value "0" after powered on.

Output：

OUT ：When the count value reaches the preset value, the count reaches the maximum in the Up count or the minimum value in the Down count, OUT = 1.

5.9 ROTB（binary rotation control）

Function：

It is used to control the rotor, such as the tool post, rotary table, etc., and the data processed by ROTB is binary.

Control conditions：

CN0 ： specify the starting number of the rotor.

CNO=0 begins the number of the position of the rotor with 0.

CNO=1 begins the number of the position of the rotor with 1.

DIR ： select the rotation directin via the shorter path or not.

DIR=0 no direction is selected. The direction of rotation is only forward.

DIR=1 selected. The direction of rotation is forward or reverse via the shorter path.

POS ： specify the operating conditions.

POS=0 calculates the Designation position.

POS=1 calculates the position one position before the Designation position.

INC ： specify the position or the number of steps.

INC=0 calculates the number of the position. When the position one position before the Designation position is to be calculated, specify INC=0 and POS=1.

INC=1 calculates the number of steps. When the difference between the current position and the Designation position is to be calculated, specify INC=1 and POS=0.

ACT ： Execution command

ACT= 0：the ROT command is not executed and OUT does not change.

ACT＝1：ROT command is executed.

Format：

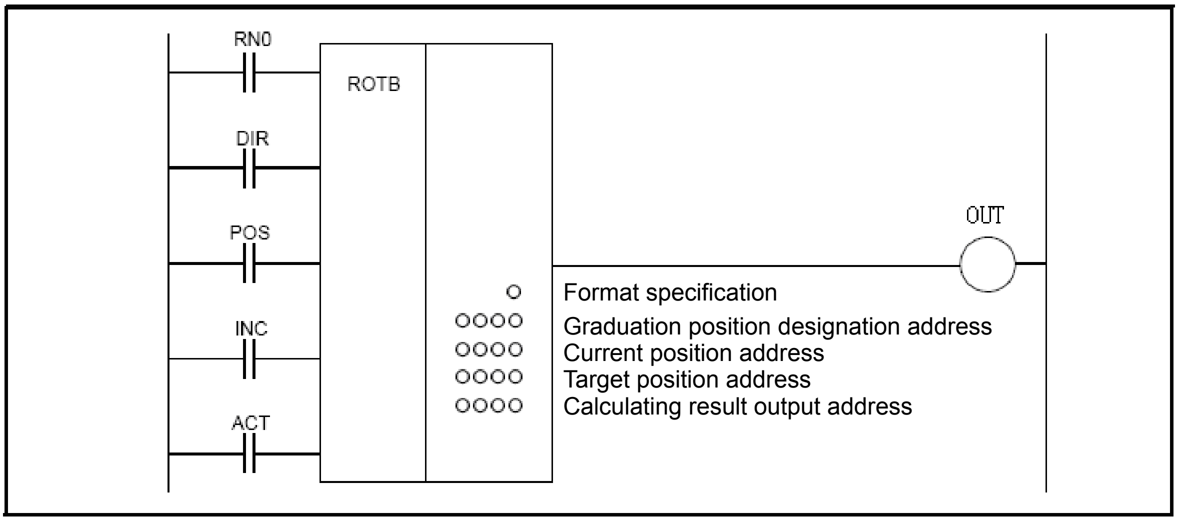


Fig. 5-12

Command table format：

Table 5-11

No.	Command	Operand	Remark
1	LD	○○○○. ○	RN0
2	LD	○○○○. ○	Selection of the shortest path DIR
3	LD	○○○○. ○	Operation condition POS
4	LD	○○○○. ○	Selection of calculation position or number of step INC
5	LD	○○○○. ○	ACT
6	FUNC	8	ROTB
7	PRM	○	Format specification
8	PRM	○○○○	Rotor indexed position address
9	PRM	○○○○	Current position address
10	PRM	○○○○	Target position address
11	PRM	○○○○	Calculating result output address
12	OUT	○○○○. ○	Rotation direction output

Parameter:

Format : specifies data length (1, 2, or 4 bytes).

1：1 byte

2：2 bytes

4：4 bytes

Rotor indexed address: specifies the address containing the number of rotary element positions to be indexed.

Current position address: specifies the address to store the current position.

Designation position address: specifies the address (or command value) to store the Designation position, such as the address of T code is output from CNC.

Calculation result output address: calculate the rotarty steps of rotor and the step to reach the Designation position or the position before the Designation. When the calculated result is used, whether ACT is 1 or not is checked.

Output:

OUT : the rotation direction output. The rotation direction via the short pathis output to OUT. OUT =0: the direction is forward（FOR）; OUT =1: it is reverse（REV）, FOR and REV definitions are as Fig. 5-13，the direction to increase the rotor position number is forward(FOR); to decrease the position number is reverse(REV).

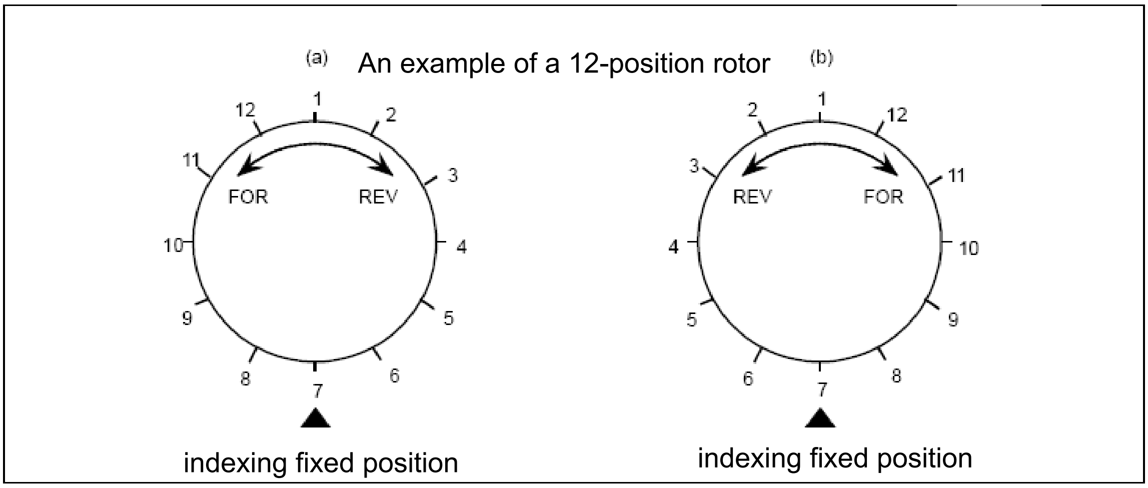


Fig. 5-13

5.10 CODB（binary code conversion）

Function:

The command converts the data in binary format to an optional binary format 1-byte, 2-byte or 4-byte, and the maximum quantity of conversion table is 256.

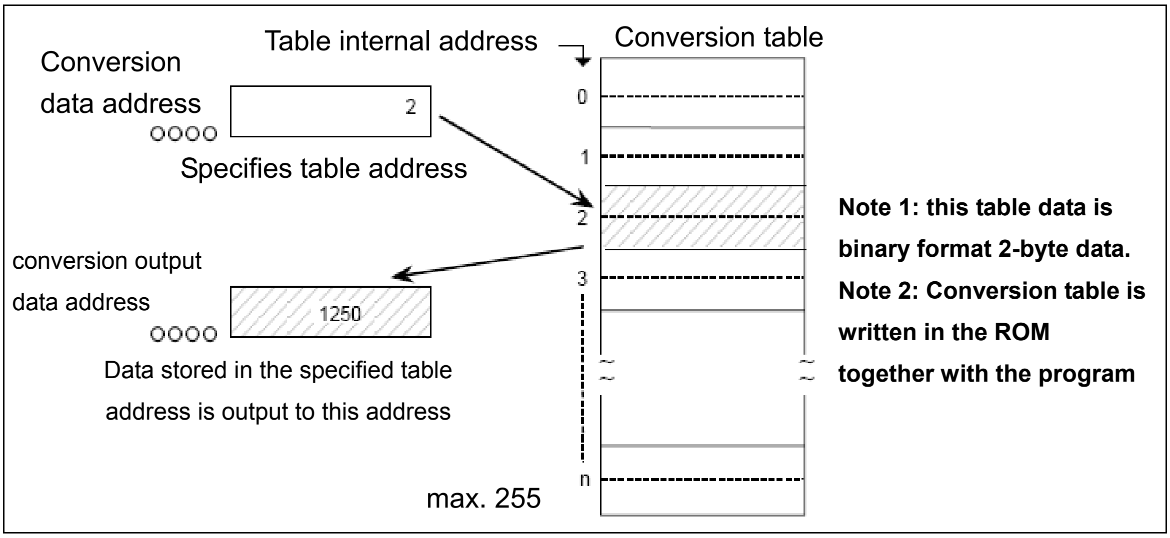


Fig. 5-14

Format:

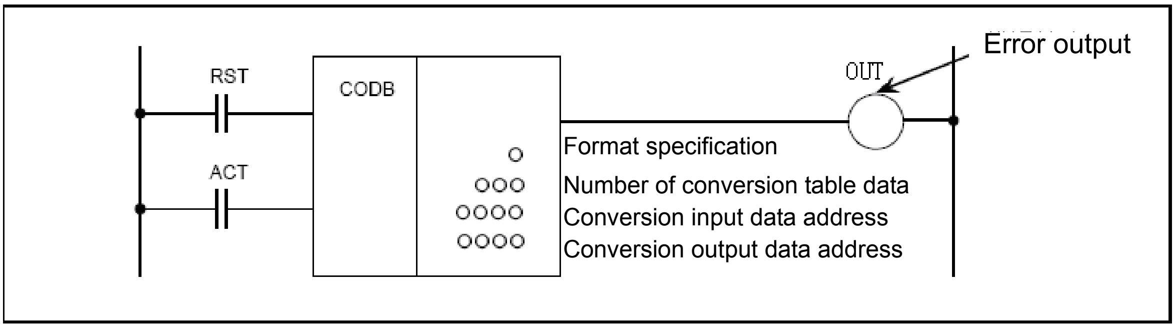


Fig. 5-15

Command table format:

Table 5-12

No.	Command	Operand	Remark
1	LD	○○○○. ○	RST
2	LD	○○○○. ○	ACT
3	FUNC	9	CODB
4	PRM	○	Format specification
5	PRM	○○○○	Number of data table
6	PRM	○○○○	Conversion input data address
7	PRM	○○○○	Conversion output data address
8	TABLE	○○○○	Table address 0 inverts data
9	:	:	
10	:	:	
n	OUT	○○○○. ○	Error output

Control conditions:

RST reset

RST=0 do not reset.

RST=1 reset error output OUT .

ACT activate command

ACT＝0 do not execute COD command.

ACT＝1 execute COD command.

Parameter:

Format specification: designates binary numberical size in the conversion table.

1：numerical data is binary 1-byte data.

2：numerical data is binary 2-byte data.

4: numerical data is binary 4-byte data.

Number of conversion table data ： designates size (1-256) of conversion table data can be made.

Conversion input data address: data in the conversion data table can be taken out by specifying the table number. The address specifying the table number is called conversion input data address, and 1-byte memory is required from the specified address.

Conversion data output address: memory of the byte length specified in the format specification is necessary from the specified address.

Output:

When there are any abnormality when executing the CODB command, OUT=1 and error will be output.

5.11 MOVE（logical product transfer）

Function:

ANDs logical multiplication data and input data, and outputs the results to a specified address. Can also be used to remove unnecessary bits from an eight-bit signal in a specifc address, etc..

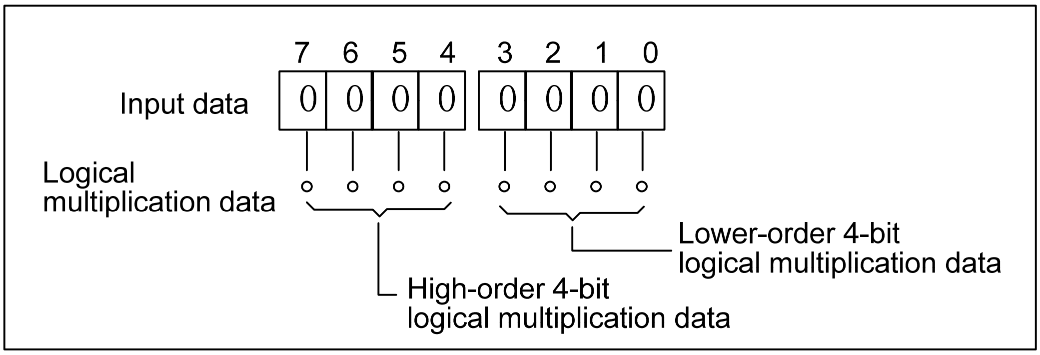


Fig. 5-16

Format:

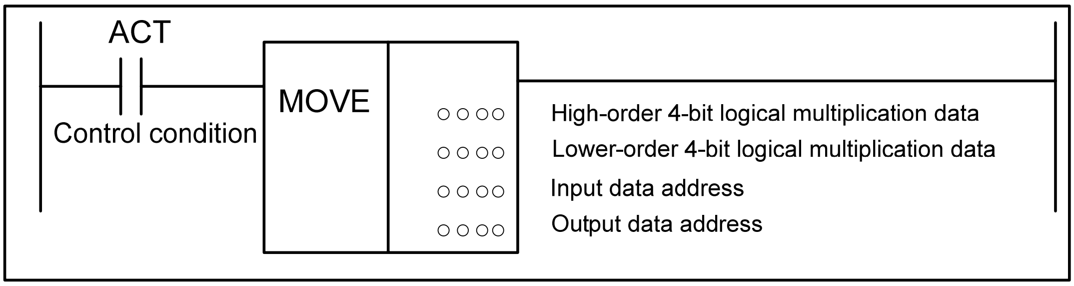


Fig. 5-17

Command table format:

Table 5-13

No.	Command	Operand	Remark
1	LD	○○○○. ○	ACT
2	FUNC	10	MOVE
3	PRM	○○○○	high-order 4-bit logical multiplication data
4	PRM	○○○○	Low-order 4-bit logical multiplication data
5	PRM	○○○○	Input data address
6	PRM	○○○○	Output data address

Control conditions：

ACT＝0：MOVE command is not executed.

ACT＝1：MOVE command is executed.

Using example：

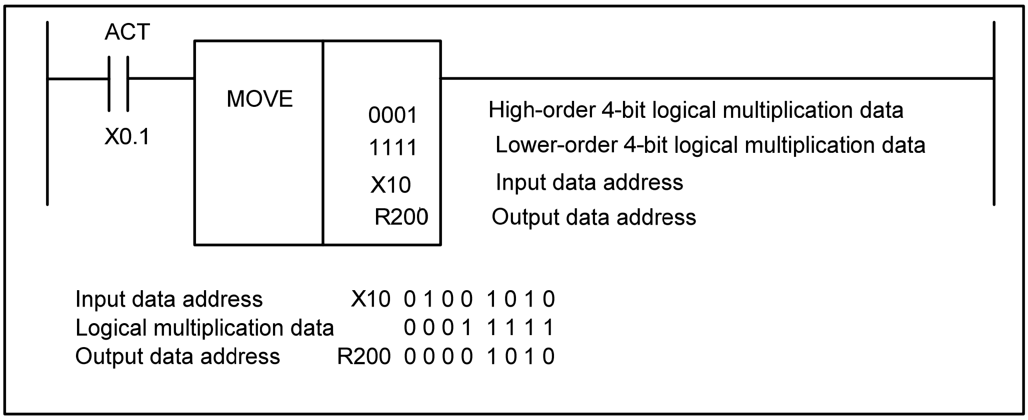


Fig. 5-18

5.12 MOVOR（data transfer after logical sum）

Function：

This command Ors the input data and the logical sum data and transfer the result to the destination.

Format:

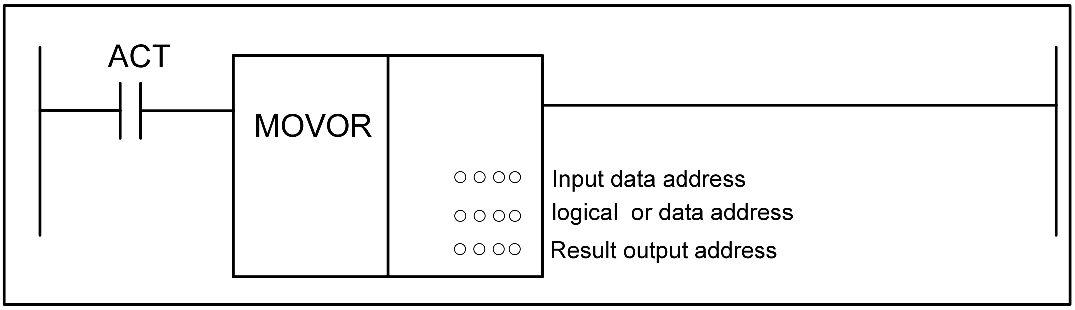


Fig. 5-19

Command table format：

Table 5-14

No.	Command	Operand	Remark
1	LD	○○○○. ○	ACT
2	FUNC	11	MOVOR
3	PRM	○○○○	Input data address
4	PRM	○○○○	Logical sum data
5	PRM	○○○○	Output data

Control conditions:

ACT＝0: do not execute MOVOR command.

ACT＝1: execute MOVOR.

Parameter:

Input data address : specifies the address for the input data.

Logical sum data address : specifies the address of the logical sum data with which to OR the transferred data.

Output address : output the result in the logical sum data address.

5.13 MOVB （transfer of 1 byte）

Function:

The command transfer 1-byte data from a specified source address to a specified destination address.

Format:

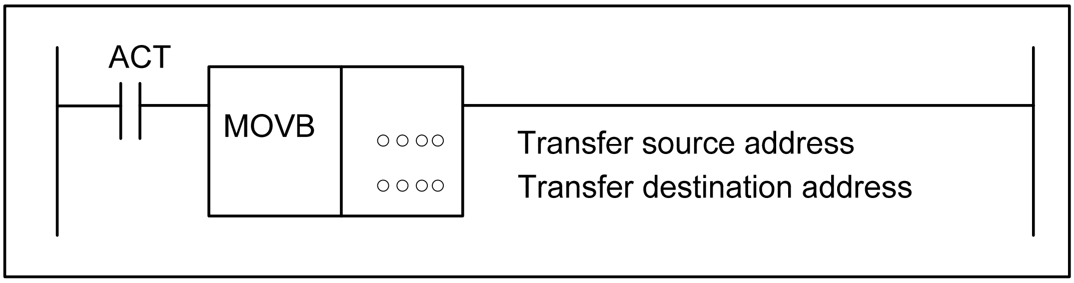


Fig. 5-20

Command table format:

Table 5-15

No.	Command	Operand	Remark
1	LD	○○○○. ○	ACT
2	FUNC	12	MOVB
3	PRM	○○○○	Transfer source address
4	PRM	○○○○	Transfer destination address

Control conditions:

ACT Execution specification

ACT＝0 : do not execute MOVB command and no data is transferred.

ACT＝1 : execute MOVB command and one-byte data is transferred.

Parameter:

Data source address : specifies source address.

Data destination address : specifies destination address.

5.14 MOVW （transfer of 2 bytes）

Function：

The command transfers 2-bytes data from a specified source address to a specified destination address.

Format：

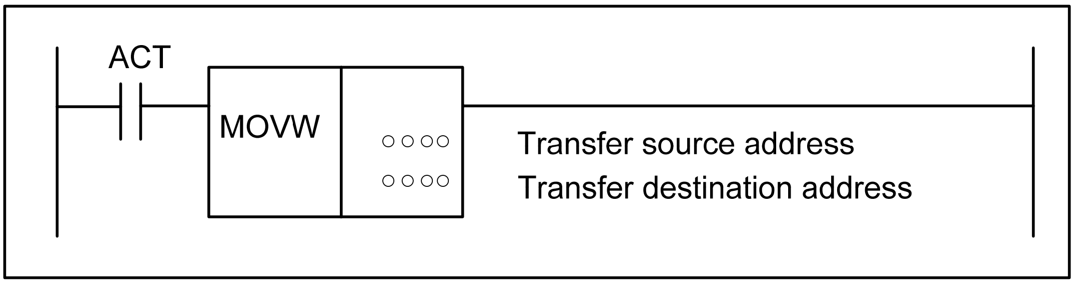


Fig. 5-21

Command table format：

Table 5-16

No.	Command	Operand	Remark
1	LD	○○○○. ○	ACT
2	FUNC	13	MOVW
3	PRM	○○○○	Transfer source address
4	PRM	○○○○	Transfer destination address

Control conditions：

ACT Execution specification

ACT＝0 : do not execute MOVW, no data is transferred.

ACT＝1 : execute MOVW command and two-byte data is transferred.

Parameter：

Data source address： specifies source address.

Data destination address： specifies destination address.

5.15 MOVN（transfer of an arbitrary number of bytes）

Function：

The command transfers data consisting of an arbitrary number of bytes from a specified source address to a specified destination address.

Format:

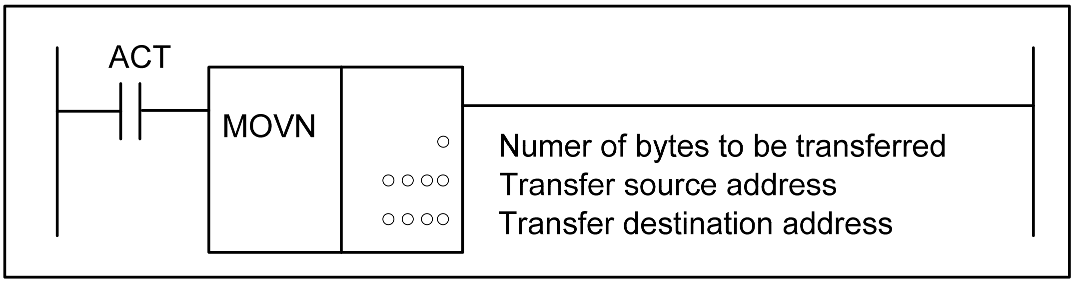


Fig. 5-22

Command table format:

Table 5-17

No.	Command	Operand	Remark
1	LD	○○○○. ○	ACT
2	FUNC	14	MOVN
3	PRM	○	Number of bytes to be transferred
4	PRM	○○○○	Transfer source address
5	PRM	○○○○	Transfer destination address

Control conditions:

ACT execution specification

ACT＝0 : do not execute MOVN command, no data is transferred.

ACT＝1 : execute MOVE command, and a specified number of bytes are transferred.

Parameter:

Number of bytes to be transferred ：specify the number（1～200）of bytes to be transferred.

Data source address: specifies the source address.

Data destination address: specifies the destination address.

5.16 PARI（parity check）

Function:

Checks the parity of code signals, and outputs an error if an abnormality is detected. Specifies either an even-or odd-parity check. Only one-byte (eight bits) of data can be checked.

Format:

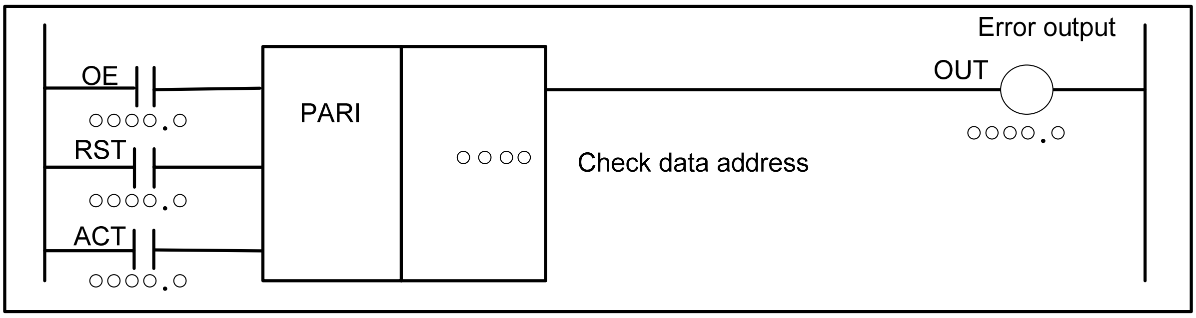


Fig. 5-23

Command table format:

Table 5-18

No.	Commnd	Operand	Remark
1	LD	○○○○. ○	O.E
2	LD	○○○○. ○	RST
3	LD	○○○○. ○	ACT
4	FUNC	15	PARI
5	PRM	○○○○	Check data address
6	OUT	○○○○. ○	Error output

Control conditions:

O.E specify even or odd.

O.E=0: even-parity check.

O.E=1: odd-parity check.

RST reset

RST=0: disables reset.

RST=1: sets error output coil OUT, that is, when OUT =1, RST=1. OUT =0.

ACT execution command

ACT=0: parity checks are not performed and the output does not change.

ACT=1: execute PARI command, performing a parity check.

Output:

If the result of executing the PARI command is abnormal, the check address data has 1-bit even in the odd check or 1-bit odd in the even check, OUT=1.

5. 17 DCNVB（extended data conversion）

Function:

This command converts 1, 2, and –byte binary code into BCD or vice versa.

Format:

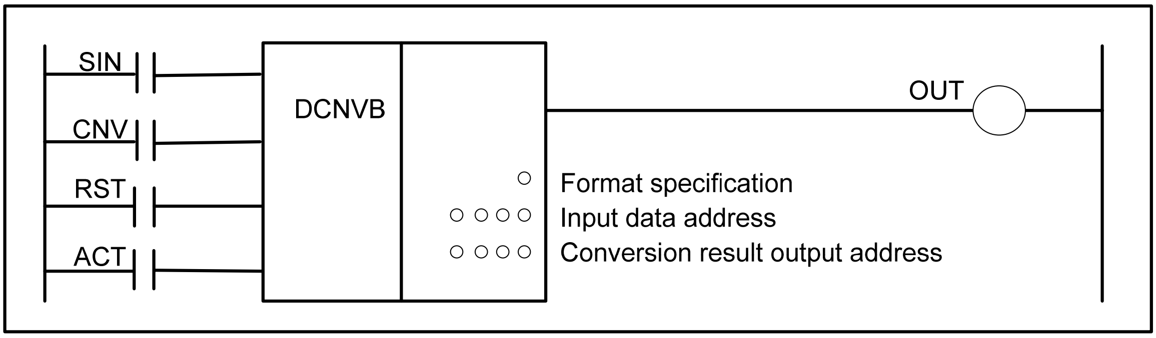


Fig. 5-24

Command table format:

Table 5-19

No.	Command	Operand	Remark
1	LD	○○○○. ○	SIN
2	LD	○○○○. ○	CNV
3	LD	○○○○. ○	RST
4	LD	○○○○. ○	ACT
5	FUNC	16	DCNVB
6	PRM	○	Format specification
7	PRM	○○○○	Input data address
8	PRM	○○○○	Conversion result output address
9	OUT	○○○○. ○	Error output

Control conditions ：

SIN sign of the data to be converted

This parameter is significant only when you are converting BCD data into binary coded data. It gives the sign of the BCD data. Though it is insignificant when you are converting binary into BCD data, you cannot omit it.

SIN=0： BCD code to be input is positive.

SIN=1：BCD code to be input is negative.

CNV type of conversion

CNV=0：convert binary data into BCD data.

CNV=1：convert BCD data into binary data.

RST reset

RST=0：release reset.

RST=1：reset error output coil OUT, that is, when OUT=1 and RST=1, OUT=0.

ACT execution command

ACT=0：data is not converted, and OUT does not change.

ACT=1：data is converted.

Parameter：

Format specification ： specify data length.

1：1 byte.

2：2 bytes.

4：4 bytes.

Input data address conversion: specify the address containing the input data address. The address of the specified table number is called as the input address of the conversion data. The address needs to provide a memory with one byte.

Address for the conversion result output: specify the output address of conversion data. Specify the number of byte of memory in the format starting from the specified address.

Error output（OUT）：

OUT =0：correct conversion.

OUT =1：abnorally.

The data to be converted is specified as BCD data but is found to be binary data, or the specified number of bytes(byte length) cannot contain the BCD data into which a binary data is converted, OUT=1.

Operaton output register R1000

Set the register after the data conversion. When the binary data is converted into BCD data, and definition of each bit is as follows (table 5-25):

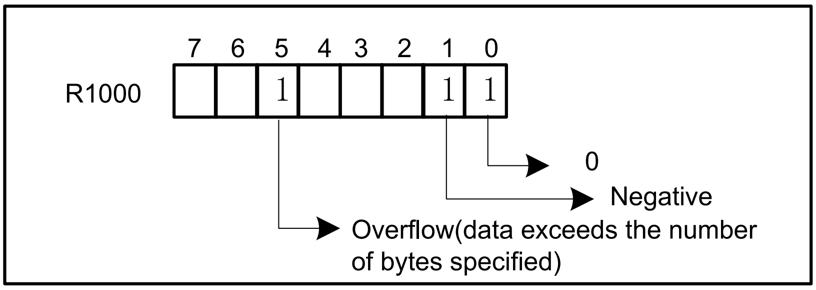


Fig. 5-25

5. 18 COMPB（binary compasion）

Function：

Compare the size fo two binary data and comparison result is stored in the comparison result address. Specify enough byte in memory area when executing COMPB command to memory input and comparison values.

Format:

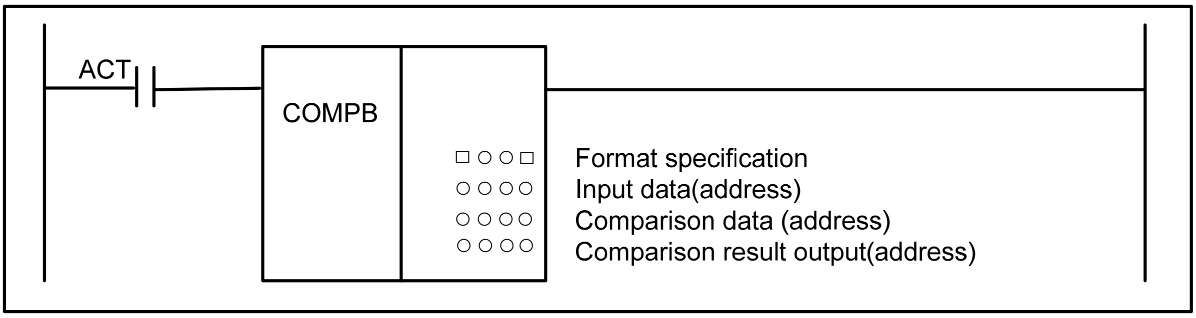


Fig. 5-26

Command table format:

Table 5-20

No.	Command	Operand	Remark
1	LD	○○○○. ○	ACT
2	FUNC	17	COMPB
3	PRM	□○○□	Format specification
4	PRM	○○○○	Input value
5	PRM	○○○○	Comparison data address
6	PRM	○○○○	Comparison result output

Control conditions:

ACT＝0：does not execute COMPB command.

ACT＝1：execute COMPB command.

Parameter:

Format destination: the specified format (constant or address) of input data and specified data length (1, 2 bytes).

Fig. 5-27

Input data: specifies the comparison input data. The input data can be specified as either a constant or the address.

Comparison data: specifies the comparison data address.

Comparison result output: specifies the comparison result output covering one byte.

Comparison result output address:

Comparison result output address bit:	bit5	Bit2	Bit1	Bit0
Input data compared	0	0	0	1
Input data＞data compared	0	0	1	0
Input data ＜data compared	0	1	0	0
data overflow	1	0	0	0

5.19　COIN（coincidence check）

Function：

Checks whether the input value and comparison value conincide and the command is available with the binary data.

Format：

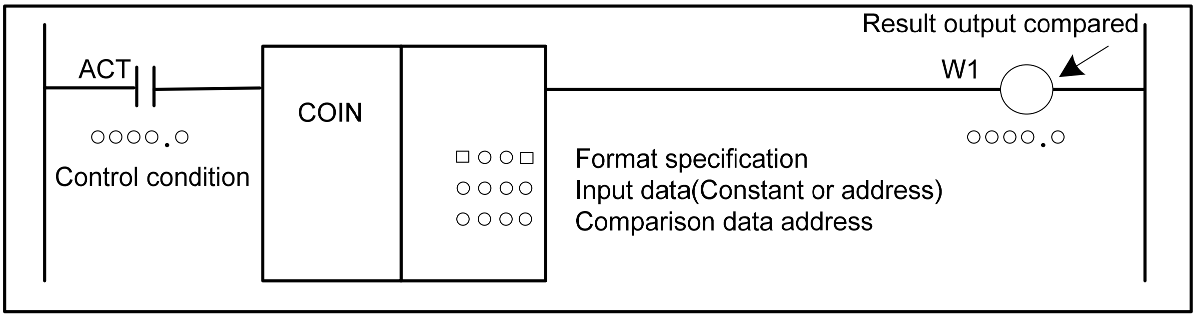


Fig. 5-29

Command table format:

Table 5-21

No.	Command	Operand	Remark
1	LD	○○○○. ○	ACT
2	FUNC	18	COIN
3	PRM	□○○□	Input value format
4	PRM	○○○○	Input value
5	PRM	○○○○	Comparison value address
6	OUT	○○○○. ○	Result output compared

Control conditions:

ACT　execution command

ACT=0：the command is not executed and OUT does not change.

ACT=1：the command is executed and the result is output to OUT.

Parameter：

Input data format: specifies input data format.

0：specifies input data as a constant.

1：specifies input data as an address.

Input data: the input data can be specified as either a constant or an address storing it.

Comparison data address: specifies the address storing the comparison data.

Output：

OUT ： OUT =0： input data ≠ comparison data.

OUT =1： input data = comparison data.

5.20 DSCHB（data search）

Function：

The command is used to searches the data in the data table. Searches the data table for a specified data, outputs an address storing it countig from thebeginning of the data table. If the data cannot be found, OUT=1.

The command is available to the binary data, and the number of data (table capacity) in the data table.

Format：

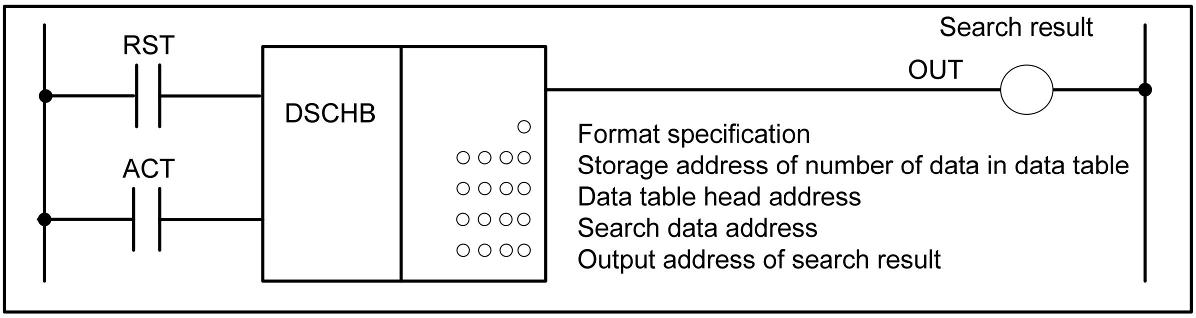


Fig. 5-30

Command table format：

Table 5-22

No.	Command	Operand	Remark
1	LD	○○○○. ○	RST
2	LD	○○○○. ○	ACT
3	FUNC	19	DSCHB
4	PRM	○	Format specification
5	PRM	○○○○	Number of data of the data table
6	PRM	○○○○	Data table head address
7	PRM	○○○○	Data table search address
8	PRM	○○○○	Search result output address
9	OUT	○○○○. ○	Error output

Control conditions：

RST reset

RST=0： release reset.

BYT=1： enable a reset, this is, sets PIT tp 0.

ACT execution command

ACT=0：the command is not executed and OUT does not change.

ACT=1：the command is executed, and the table tinternal number storing the desired data is output, if the data cannot be found, OUT is set to1.

Parameter：

Format specification: specifies the length to search data.

1：1-byte length

2：2-byte length

4：4-byte length

Number of data of the data table: the size of the data table. The byte length specified by the addresss is assigned to the the memory area requiring the byte. The number of data of data table is n+1 (the beginning of the data table is 0 and the end is n)

Data table head address: set the data head address. The head address must D address of D data table.

Search data address: set the address of the data to be searched.

Search result output address: if the data being searched for is found, the internal number of the table storing the data is output to this field. The search result output address field requires memory whose size is the number of bytes conforming to the size of the data specified by byte.

Output：

OUT =0: the data to be searched exists.

OUT =1，the data to be searched does not exist.

5.21 XMOVB（binary indexed modifier data transfer）

Function：

This functional command instructs reading and rewriting of data in the data. The number of data (table capacity) in the ata table can be specified by specifying the address.

Format:

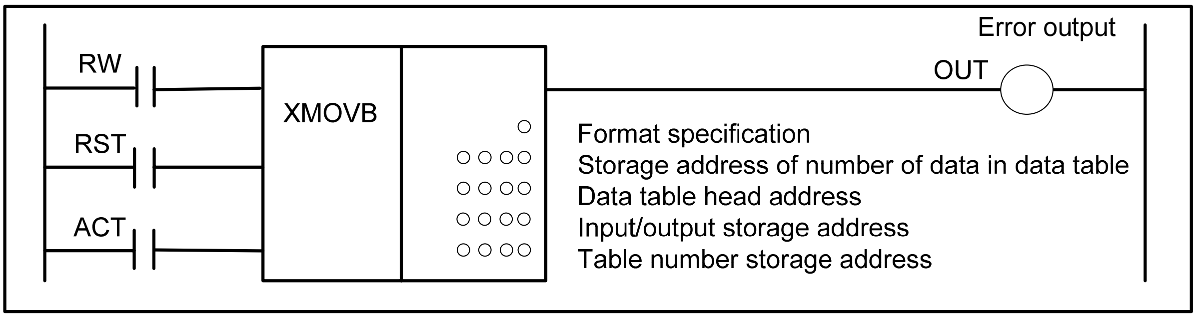


Fig. 5-31

Command table format ：

Table 5-23

No.	Command	Operand	Remark
1	LD	○○○○. ○	RW
2	LD	○○○○. ○	RST
3	LD	○○○○. ○	ACT
4	FUNC	20	XMOVB
3	PRM	○	Format specification
5	PRM	○○○○	Data capcity
6	PRM	○○○○	Data table head address
7	PRM	○○○○	Input/output data storage address
8	PRM	○○○○	Table number storage address
9	OUT	○○○○. ○	Error output

Control conditions：

RW read, write designation

RW=0： read data from data table.

RW=1： write data to data table.

RST reset

RST=0： release reset.

RST=1： reset, OUT =0.

ACT activation command

ACT＝0： do not execute XMOVB command，OUT does not change.

ACT＝1： execute XMOVB command.

Parameter：

Format specification: specifies data length.

1：1-byte length

2：2-byte length

4：4-byte length

Storage address of number of data table: it is used to store the number of data in the data table, the number of byte is as follows with the specified length and the effective range of data is determined by the byte length specified by the format.

1-byte length：1 to 255.

2-byte length：1 to 65535（actually, set a value below the size of the D area）.

4-byte length：1 to 99999999（actually, set a value below the size of the D area）.

Data table head address: sets head address in the data table. The memory area of data table is: the byte length × the number of data table. The head address must be D address in D data table.

Input/output(I/O) data storage address: in case of the reading, set the address of the memory which stores a reading result. In case of the writing, set the address of the memory which stores a writing result.

Index storage address:set the address of the memory in which an index value is stored. The memory with the byte length set in format specification is necessary. When settting an index value above the value to set in storage address of number of data table, it causes error output OUT=1.

Output：

In the case where the index value set in the index storage address exceeds the value set in the storage address of number of data table, OUT=1, and the reading or writing of the data table is not executed.

OUT =0，No error.

OUT =1: Error found.

5.22 ADDB(addition)

Function：

The command is used to the binary addition operation with 1-, 2- or 4-byte length. The addend data and the output data of addion operation result are set with the storage address of the corresponding byte length

Format：

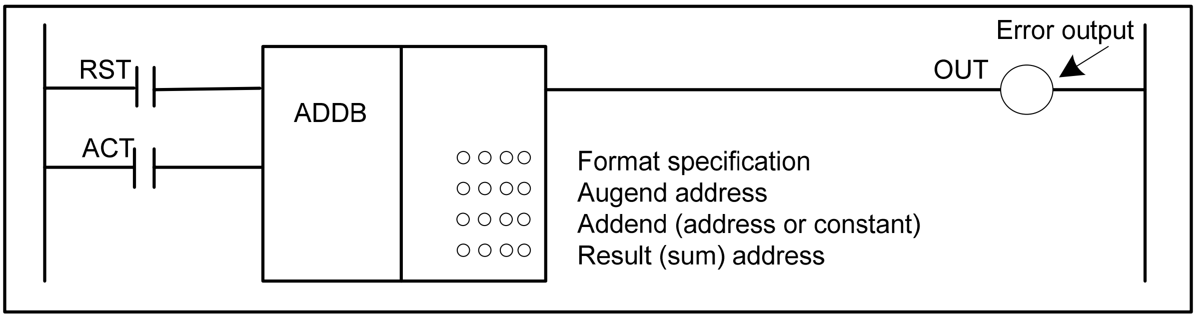


Fig. 5-32

Command table format ：

Table 5-24

No.	Command	Operand	Remark
1	LD	○○○○. ○	RST
2	LD	○○○○. ○	ACT
3	FUNC	21	ADDB
4	PRM	□○○□	Format specification
3	PRM	○○○○	Summand address
5	PRM	○○○○	addend address
6	PRM	○○○○	Sum output storage address
7	OUT	○○○○. ○	Error output

Control conditions：

RST reset

RST=0： release reset.

RST=1： reset OUT =1.

ACT execution command

ACT＝0 ： do not execute ADDB command.

ACT＝1 ： execute ADDB command.

Parameter：

Foramt designation: specifies the data length (1, 2, 4 bytes) and the the specified method of addend (constant or address.

Fig. 5-33

Summand address : specifies the address.

Addend : the specified method of addend is determined by the format specification.

Sum output address: specifies the address to which the sum is to be output.

Output：

OUT =0： operation normability.

OUT =1： operation abnormality.

When the result of addition exceeds the specified data length, OUT=1.

Operation output register**(R1000)**：

Each bit of operation output register:

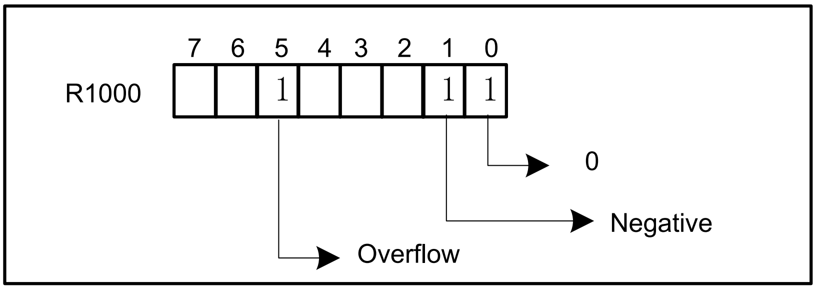


Fig. 5-34

5.23 SUBB（binary subtraction）

Function：

This command is used to the binary subtraction with 1-, 2-, 4-length. The minuend data, the subtraction operation output data need to set the storage address of corresponding byte length.

Format：

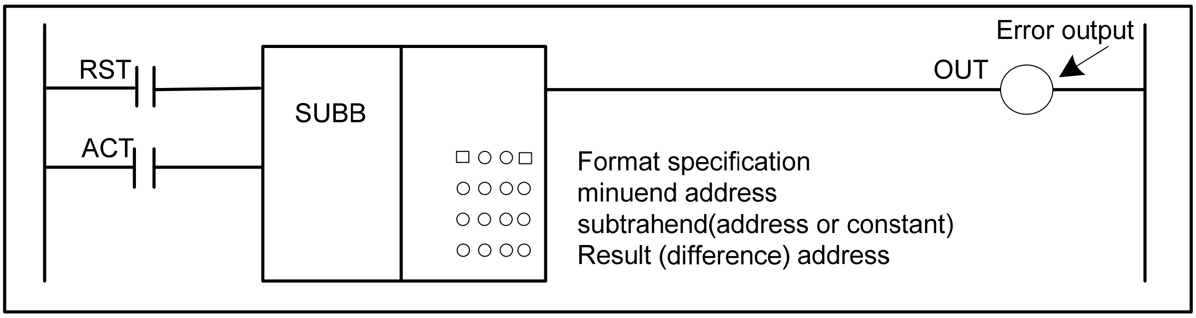


Fig. 5-35

Command table format ：

Table 5-25

No.	Command	Operand	Remark
1	LD	○○○○. ○	RST
2	LD	○○○○. ○	ACT
3	FUNC	22	SUBB
4	PRM	□○○□	Format specification
3	PRM	○○○○	Minuend address
5	PRM	○○○○	subtrahend
6	PRM	○○○○	Operation output storage address
7	OUT	○○○○. ○	Error output

Control conditions：

RST reset

RST=0： release reset.

RST=1： reset OUT =1.

ACT execution command

ACT＝0 ： do not execute SUBB command.

ACT＝1 ： execute SUBB command.

Parameter：

Format specification: specifies the data length (1-, 2-, 4-byte) and the specified method of the subtrahend (constant or address).

Fig. 5-36

Minuend address ： set the address storing the minuend.

Subtrahend ： the specified method of the subtrahend depends on the format specification.

Operation result output address: set the address to which the operation result is output.

Output：

OUT =0： operation normability.

OUT =1： operation abnormality.

When the operation result exceeds the specified data length, OUT=1.

Operation result register (R1000)：

Each bit of operation result register:

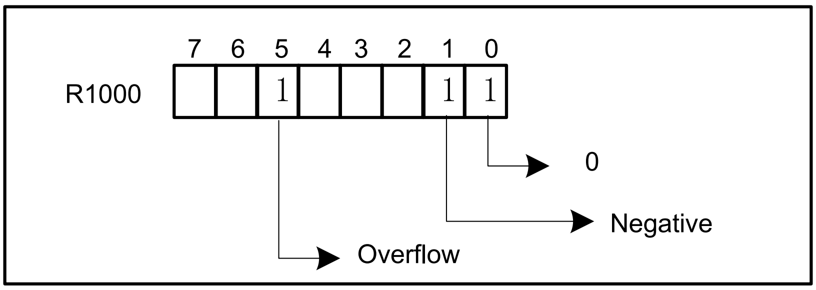


Fig. 5-37

5.24 MULB（binary multiplication）

Function：

This command multiplies 1-, 2-, 4-byte binary data. The operation result is output to the operation result output address. The multiplicand data and the multiplication operation result output data need to set the storage address of corresponding byte length.

Format：

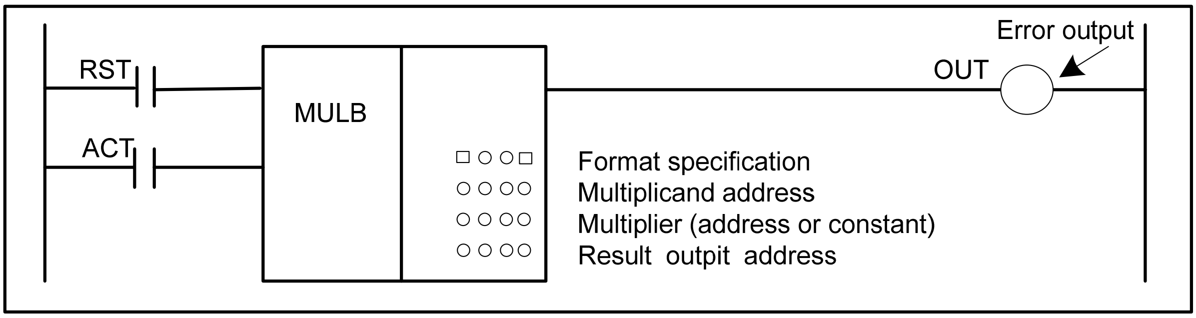


Fig. 5-38

Command table format ：

Table 5-26

No.	Command	Operand	Remark
1	LD	○○○○. ○	RST
2	LD	○○○○. ○	ACT
3	FUNC	23	MULB
4	PRM	□○○□	Format specification
3	PRM	○○○○	Multiplicand address
5	PRM	○○○○	Multiplier
6	PRM	○○○○	Operation result output storage address
7	OUT	○○○○. ○	Error output

Control conditions：

RST reset

RST=0： release reset .

RST=1： reset OUT =1.

ACT execution command

ACT＝0 ： do not execute MULB command.

ACT＝1 ： execute MULB command.

Parameter：

Format specification: specifies the data length (1-, 2-, 4-byte) and the specified method of the multiplication (constant or address).

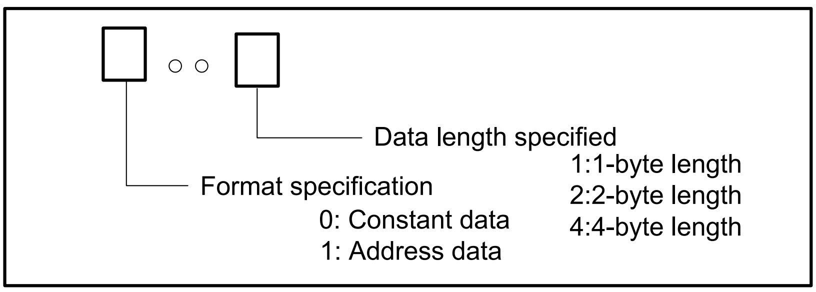


Fig. 5-39

Multiplicand address : address containing the multiplicand.

Multiplier data : the specified method of the multiplier is determined by the format specification.

Operation result output address: specifies the address to contain the operation result.

Output:

OUT =0: operation normability.

OUT =1: operation abnormality.

When the result of multiplication exceeds the specified data length, OUT=1.

Operation result register(R1000):

Each bit of operation result register:

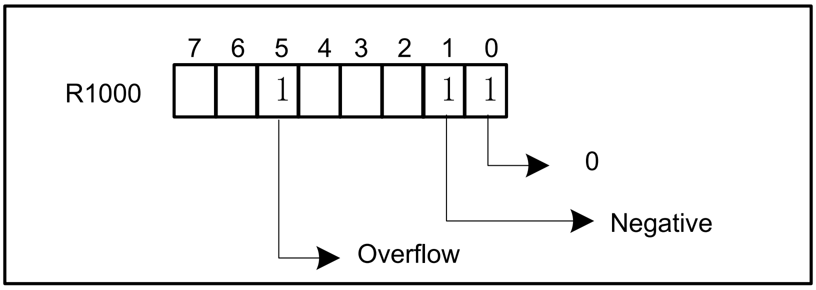


Fig. 5-40

5.25 DIVB（binary division）

Function:

This command divides 1-, 2-, 4-byte binary data. The operation result is output to the operation result output address. The divisor and the dividend and the operation result output data need to set the storage address of corresponding byte length.

Format:

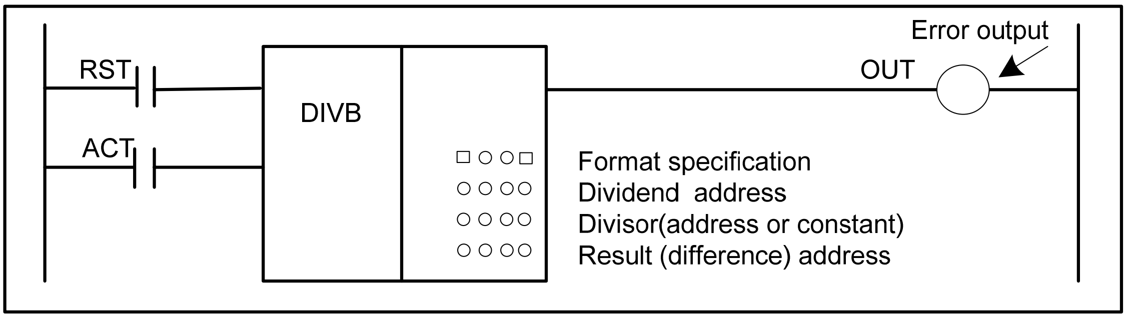


Fig. 5-41

Command table format ：

Table 5-27

No.	Command	Operand	Remark
1	LD	○○○○. ○	RST
2	LD	○○○○. ○	ACT
3	FUNC	24	DIVB
4	PRM	□○○□	Format specification
5	PRM	○○○○	Dividend address
6	PRM	○○○○	Divisor
7	PRM	○○○○	Operation result output storage address
8	OUT	○○○○. ○	Error output

Control conditions：

RST reset

RST=0： release reset .

RST=1： reset OUT =1.

ACT execution command

ACT＝0 ： do not execute DIVB command .

ACT＝1 ： execute DIVB command .

Parameter：

Format specification: specifies the data length (1-, 2-, 4-byte) and the specified method of the divisor data (constant or address).

Fig. 5-42

Dividend address : sets the address storing the dividend.

Divisor : the specified method of the divisor is determined by the format specification.

Operation result output address: specifies the address to which operation result is output.

Output：

OUT =0： operation normality.

OUT =1： operation abnormality.

When the divisor is 0, OUT=1.

Operation result register(R1000)：

Each bit of operation result register:

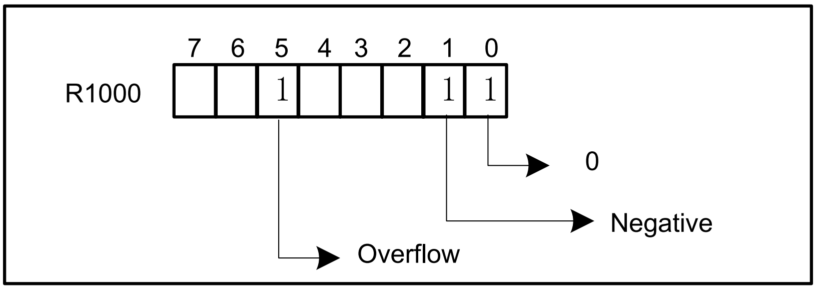


Fig. 5-43

Remainder output register:

The remainder is stored to R1002-R1005 according to the data length when there is the remainder.

5.26 NUMEB（definition of binary constant）

Function：

This command is used to the decimal constant data assign to the specified address. The output data is the binary data and is stored to the specified storage address. The data length can be 1-, 2- or 4- byte length according to the specified.

Format：

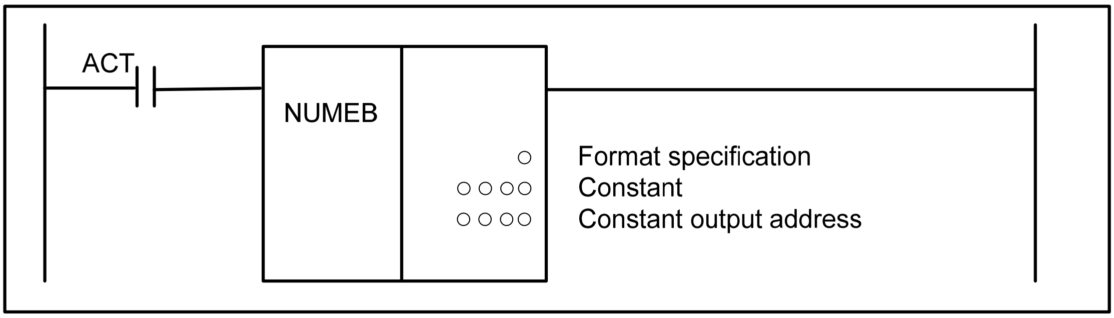


Fig. 5-44

Command table format ：

Table 5-28

No.	Command	Operand	Remark
1	LD	○○○○. ○	ACT
2	FUNC	25	NUMEB
3	PRM	○	Format specification
4	PRM	○○○○	Constant
5	PRM	○○○○	Constant output address

Control conditions：

ACT execute Command

ACT＝0 : do not execute NUMEB command .

ACT＝1 : execute NUMEB command .

Parameter ：

Format specification: specifies the data length.

1：1-byte length.

2：2-byte length.

4：4-byte length.

Constant : specifies the defined constant and its value is the decimal data.

Constant output address: specifies the address to output the operation result.

5.27 DIFU（Edge Up detection）

Function：

The command sets the output relay to 1 for one scanning period on a Edge Up of the output signal.

Format：

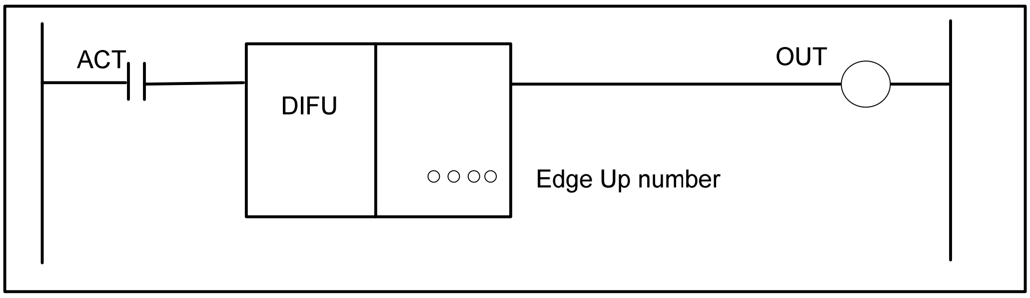


Fig. 5-45

Command table format ：

Table 5-29

No.	Command	Operand	Remark
1	LD	○○○○. ○	ACT
2	FUNC	26	DIFU
3	PRM	○○○○	Edge Up signal
4	OUT	○○○○. ○	Output

Control conditions：

ACT execute Command

ACT＝0 : do not execution command.

ACT＝1 : execution command, output signal sets one scanning period on the ACT Edge Up.

Parameter：

Edge Up number: specifies the Edge Up along the command serial number and its range is 1 to 256.

Warning:

If the same number is used for another DIFU command or a DIFD command in one ladder diagram, operation is not guaranteed.

Output（OUT）:

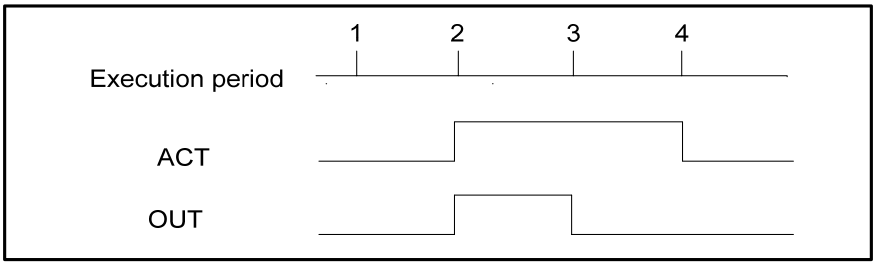


Fig. 5-46

5.28 DIFD（Edge Down detection）

Function:

The command sets the output relay to 1 for one scanning period on a Edge Down of the output signal.

Format:

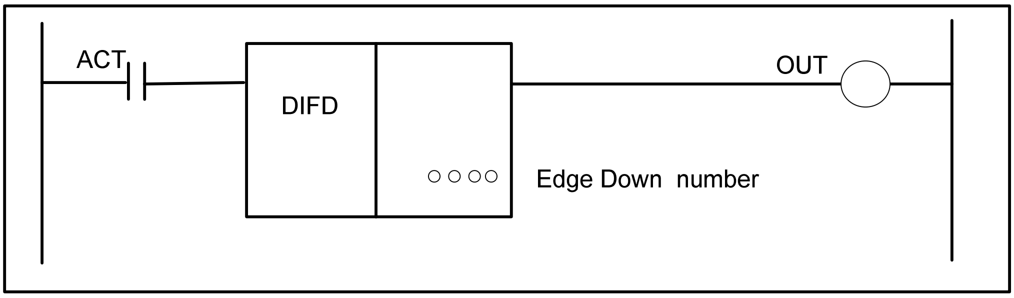


Fig. 5-47

Command table format :

Table 5-30

No.	Command	Operand	Remark
1	LD	○○○○. ○	ACT
2	FUNC	27	DIFD
3	PRM	○○○○	Edge Down signal
4	OUT	○○○○. ○	output

Control conditions:

ACT execution command

ACT＝0 : do not execute command.

ACT＝1 : execution command, output signal sets one scanning period on the ACT Edge Down.

Parameter:

Edge Down number: specifies the Edge Down along the command serial number and its range is 1 to 256.

> Warning:
>
> If the same number is used for another DIFU command or a DIFD command in one ladder diagram, operation is not guaranteed.

Output（OUT）:

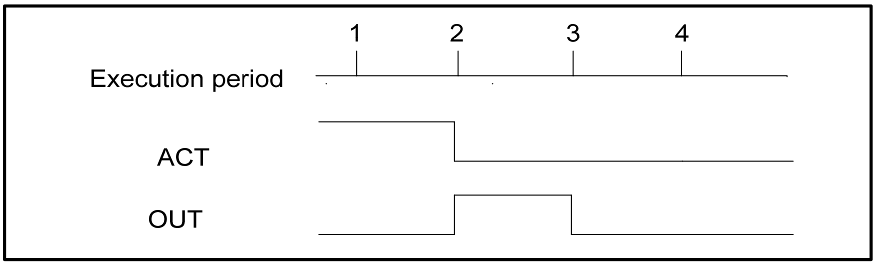


Fig. 5-48

5.29 SFT（shift register）

Function:

The command shifts 2-byte data by a bit to the left or right.

OUT=1 when data “1” is shifted from the left extremity (bit 15) in left shift or from the right extremity (bit 0) in right shift.

Format:

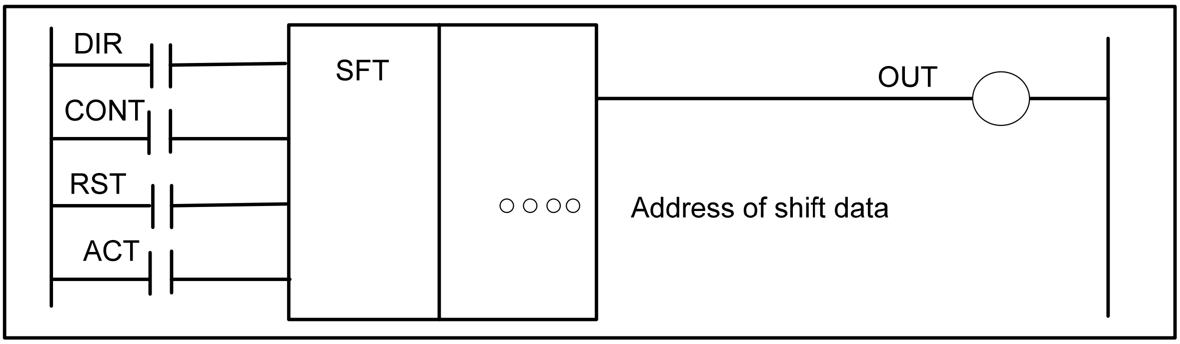


Fig. 5-49

Command table format:

Table 5-31

No.	Command	Operand	Remark
1	LD	○○○○. ○	DIR
2	LD	○○○○. ○	CONT
3	LD	○○○○. ○	RST
4	LD	○○○○. ○	ACT
5	FUNC	28	SFT
6	PRM	○○○○	Shift data
7	OUT	○○○○. ○	output

Control conditions:

DIR specifies shift direction

DIR=0 left shift

DIR=1 right shift

CONT specifies condition

CONT=0 the condition of a data bit is set to the original bit position of the on "0"bit.

CONT=1 the condition of a data bit is set to the original bit position of the on "1"bit..

RST reset

RST=0 OUT is not reset

RST=1 OUT reset （OUT =0）

ACT execution condition

ACT=0 do not execute SFT command

ACT=1 execute shift. When ACT=1, set ACT to 0.

Parameter:

Shift data address: designate addresses which require a continuous 2-byte memory for shift data.

Output:

OUT ： OUT =0 "1" is not shifted out after the shift operation.

OUT =1 "1" is shifted out after the shift operation.

5.30 EOR（EOR）

Function:

The EOR instruction exclusive-Ors the contents of address A with a constant (or the contents of address B), and stores the result at address C.

Format：

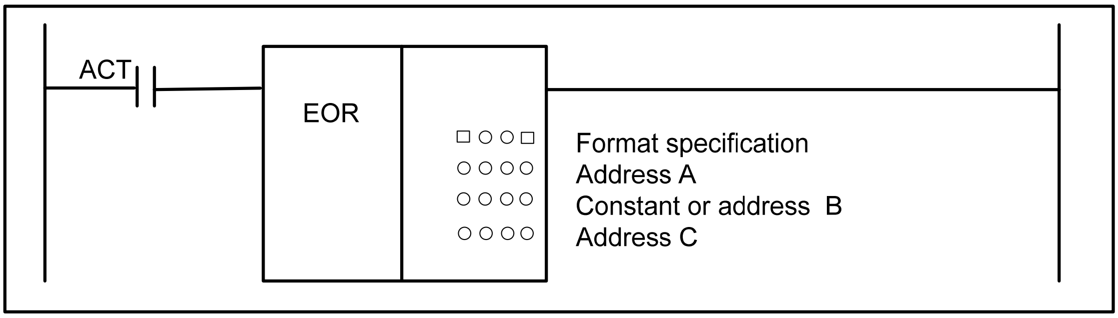


Fig. 5-50

Command table format ：

Table 5-32

No.	Command	Operand	Remark
1	LD	○○○○. ○	ACT
2	FUNC	29	EOR
3	PRM	□○○□	Format specification
4	PRM	○○○○	Address A
5	PRM	○○○○	Constant or address B
6	PRM	○○○○	Address C

Control conditions：

ACT execution condition

ACT＝0 ： do not execute EOR command .

ACT＝1 ： execute EOR command .

Parameter：

Format specification ： Specify a data length (1-, 2-, 4-byte) and an input data format(constant or address).

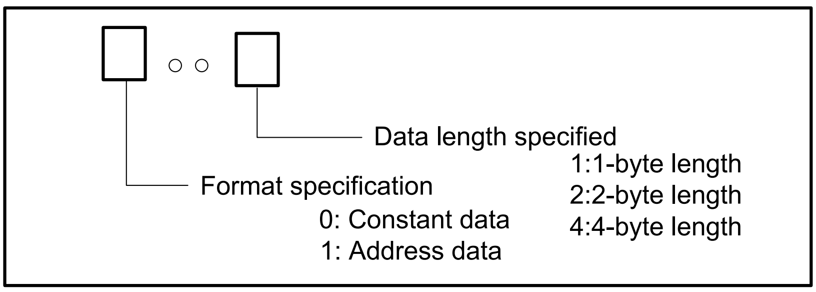


Fig. 5-51

Address A ： the head address of the input data to be exclusive-ORed.

Constant or address B ： Input data to be exclusive-ORed with A. the designation is determined by the format, that is, constant or address.

Address C ： Address used to store the result of an exclusive EOR operation. The result of an exclusive EOR operation is stored starting at this address, and has the data length specified in Length format specification.

Example:

When address A and B hold the following data:

Address A

1	1	1	0	0	0	1	1

Address B

0	1	0	1	0	1	0	1

The result of the exclusive EOR operation is as follows:

Address C

1	0	1	1	0	1	1	0

5.31 AND（logical and）

Function:

The command ANDs the contents of address A with a constant (or the contents of address B), and stores the result at address C.

Format:

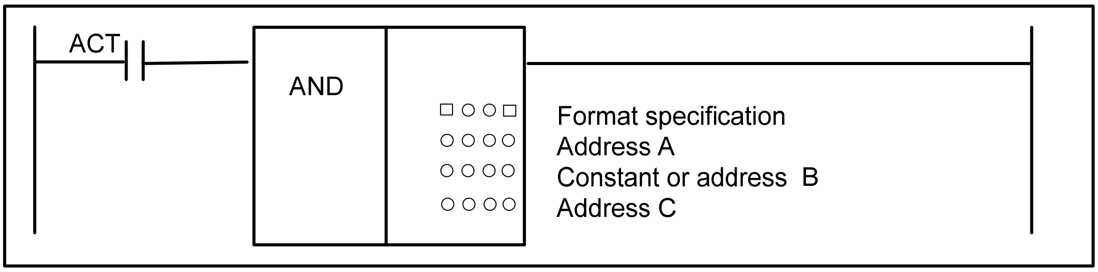


Fig. 5-52

Command table format ：

Table 5-33

No.	Command	Operand	Remark
1	LD	○○○○. ○	ACT
2	FUNC	30	AND
3	PRM	□○○□	Format specification
4	PRM	○○○○	Address A
5	PRM	○○○○	Constant or address B
6	PRM	○○○○	Address C

Control conditions:

ACT execution conditions

ACT＝0 ： do not execute AND command.

ACT＝1 ： execute AND command .

Parameter:

Format specification ： Specify a data length (1-, 2-, 4-byte) and an input data format(constant or address).

Fig. 5-53

Address A : the head address of the input data to be exclusive-ANDed.

Constant or address B : Input data to be exclusive-ANDed with A. the designation is determined by the format, that is, constant or address.

Address C : Address used to store the result of an exclusive AND operation. The result of an exclusive AND operation is stored starting at this address, and has the data length specified in Length format specification.

Example:

When address A and address B has the following data:

Address A	1	1	1	0	0	0	1	1
Address B	0	1	0	1	0	1	0	1

The result of the AND operation is as follows:

Address C	0	1	0	0	0	0	0	1

5.32 OR（logical or）

Function：

The command Ors the contents of address A with a constant (or the contents of address B), and stores the result at address C.

Format：

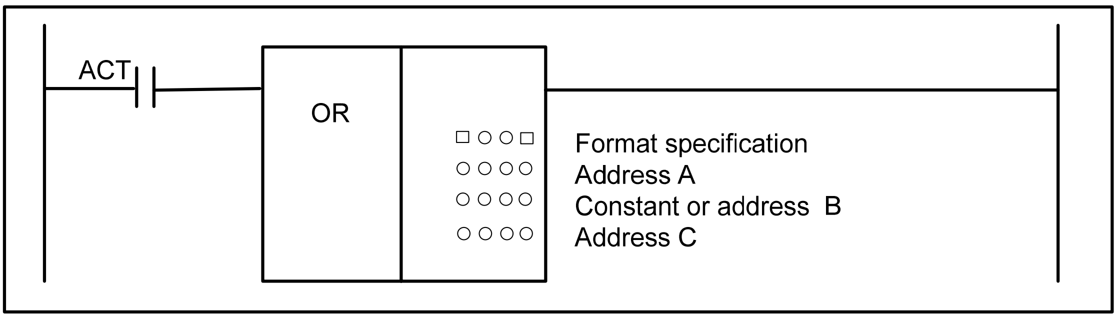


Fig. 5-54

Command table format ：

Table 5-34

No.	Command	Operand	Remark
1	LD	○○○○. ○	ACT
2	FUNC	31	OR
3	PRM	□○○□	Format specification
4	PRM	○○○○	Address A
5	PRM	○○○○	Constant or address B
6	PRM	○○○○	Address C

Control conditions：

ACT execution condition

ACT＝0 ： do not execute OR command .

ACT＝1 ： execute OR command .

Parameter：

Format specification ： Specify a data length (1-, 2-, 4-byte) and an input data format(constant or address).

Fig. 5-55

Address A ： the head address of the input data to be ORed.

Constant or address B ： Input data to be ORed with A. the designation is determined by the format, that is, constant or address.

Address C ： Address used to store the result of an OR operation. The result of an OR operation is stored starting at this address, and has the data length specified in length format specification.

Example：

When address A and address B have the following data:

Address A

1	1	1	0	0	0	1	1

Address B

0	1	0	1	0	1	0	1

The result of the OR operation is as follows:

Address C

1	1	1	1	0	1	1	1

5.33 NOT（logical not）

Function：

The command inverts each bit of the contents of address A, and stores the result at address B.

Format：

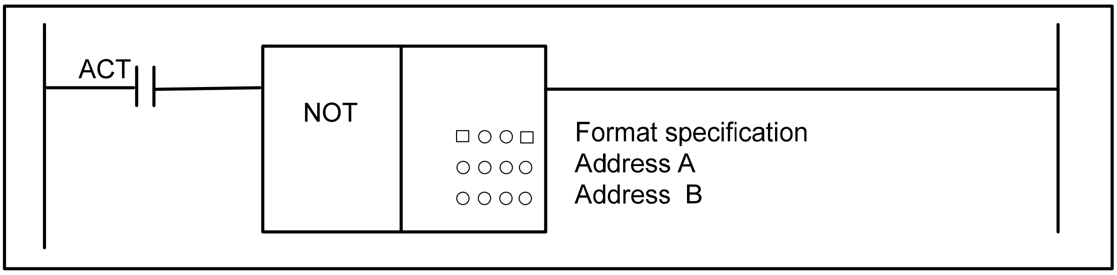


Fig. 5-56

Command table format ：

Table 5-35

No.	Command	Operand	Remark
1	LD	○○○○. ○	ACT
2	FUNC	32	NOT
3	PRM	□○○□	Format specification
4	PRM	○○○○	Address A
5	PRM	○○○○	Address B

Control conditions：

ACT execution condition

ACT＝0，do not execute NOT command .

ACT＝1，execute NOT command .

Parameter：

Format specification: specifies a data length (1-, 2-, 4-byte).

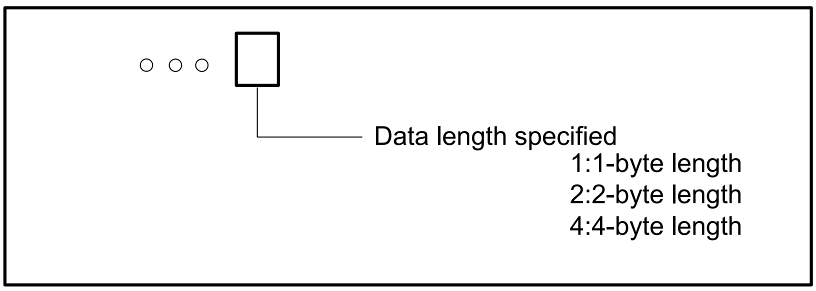


Fig. 5-57

Address A : specifies the head address of the input data to be inverted bit by bit.

Address B : specifies the address used to output the result of a NOT operation. The result of a NOT operation is stored starting at this address, and has the data length specified in format specification.

Example:

When address A and B have the following data:

Address A

1	1	1	0	0	0	1	1

The result of the NOT operation is as follows:

Address B

0	0	0	1	1	1	0	0

5.34 COM（common line control）

Function:

This command can be used to control the coil working from COM to COME (common line end command). The system specifies 0 for the number of coils and uses the common line control end command to use this function. The system alarms when the common line end command is not specified.

Format:

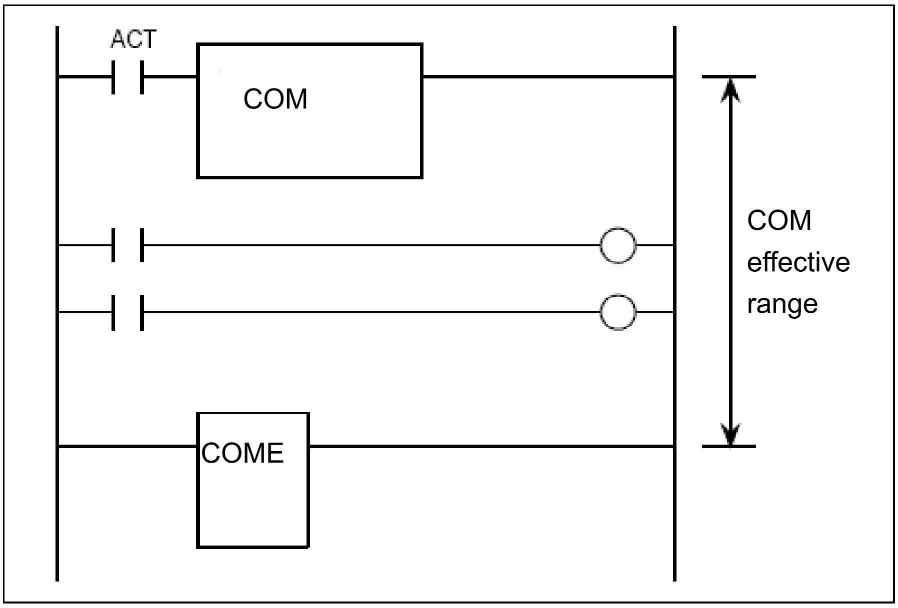


Fig. 5-58

Command table format :

Table 5-36

No.	Command	Operand	Remark
1	LD	○○○○. ○	ACT
2	FUNC	33	COM

Control conditions:

ACT=0: the specified number of coils or the coils within the region specified are unconditionally turned off (set to 0).

ACT=1: not execute.

Parameter：

Specifies the number of coil: specifies to 0 and use COM specifying range.

Note:

1. In the range specified with a COM instruction, no additional COM instruction can be specified.
2. the coil for WRT.NOT in the range specified with a COM instruction is singly set to 1 (OUTN=1) ACT=0.

5.35 COME（common line control end）

Function：

The instruction can be used to specify the control range of the common control line instruction (COM). This instruction cannot be used alone. It must be used together with the COM instruction.

Format：

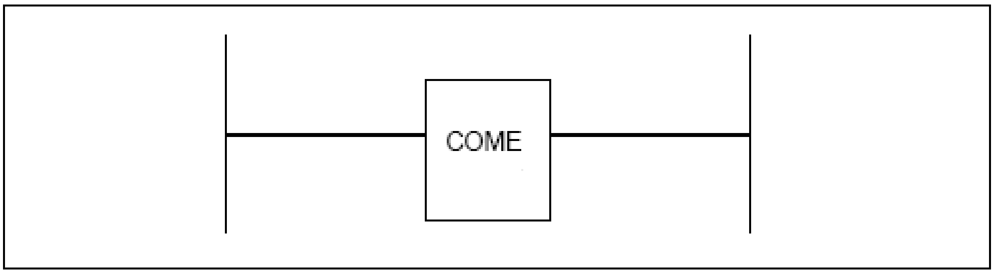


Fig. 5-59

Command table format ：

Table 5-37

No.	Command	Operand	Remark
1	FUNC	34	COME

5.36 JMP（jump）

Function：

The JMP transfers control to a ladder. When the JMP command is executed, the execution process jumps to the jump end command but does not execute the logic command (including functional command) between JMP and JMPE command. The specified coil number is 0. when the system uses JMPE command, it jumps the range. The system prompts the alarms when it does not command the jump end command.

Format：

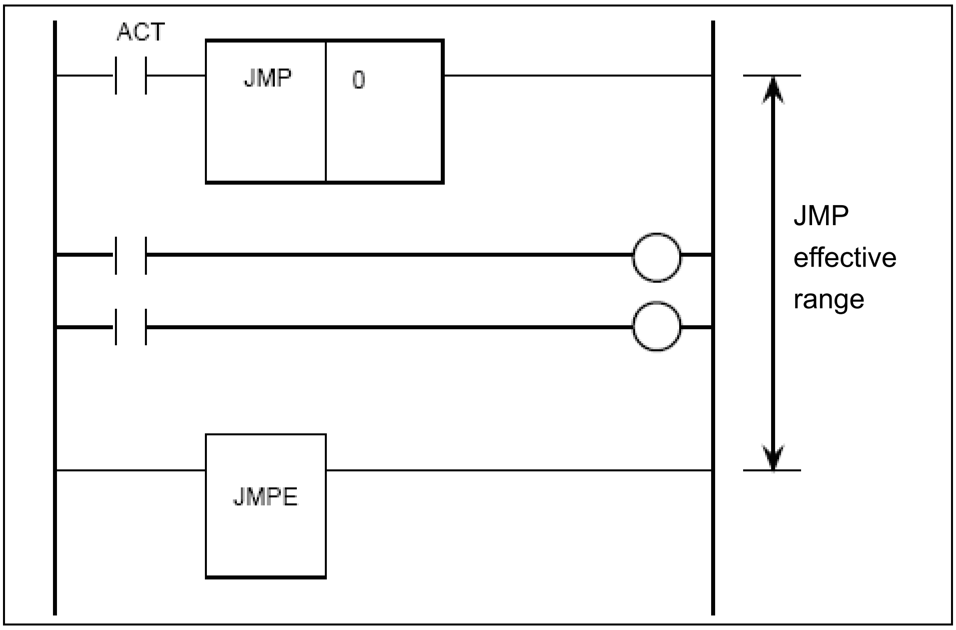


Fig. 5-60

Command table format ：

Table 5-38

No.	Command	Operand	Remark
1	LD	○○○○. ○	ACT
2	FUNC	35	JMP
3	PRM	○	

Control conditions：

ACT=0：do not execute jump. The next command after the JMP command is executed.

ACT=1：jump the logical command (including functional command) in the specified range, and execute the program.

Parameter：

Specifies the number of coil: it is set to 0, use JMPE to specify the range.

Note：

JMP command operation.

ACT=1: the program jumps to the place where the jump end command (JMPE) is. The logical command (including functional command) in the specified range is not executed.

In compileing the program, do not create a program in which a combination of JMP and JMPE command is used to cause a jump to and from a sequence between the COM and COME command. The ladder sequence may not be able to operate normally after the jump.

5.37 JMPE（jump end）

Function：

Specifies the end of JMP(jump command) range. The command must be used together with JMP command.

Format：

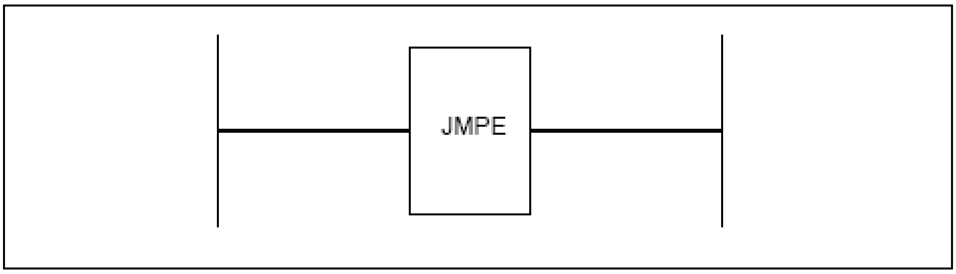


Fig. 5-61

Command table format ：

Table 5-39

No.	Command	Operand	Remark
1	FUNC	36	JMPE

5.38 CALL（conditional subprogram call）

Function：

A jump occurs to the subprogram when a condition is satisfied.

The command has the characteristics and limits as follows:

- ＊ Many call command can call the same one subprogram.
- ＊ The call command can be nested.
- ＊ The subprogram must follow END2 to be compiled.

Format：

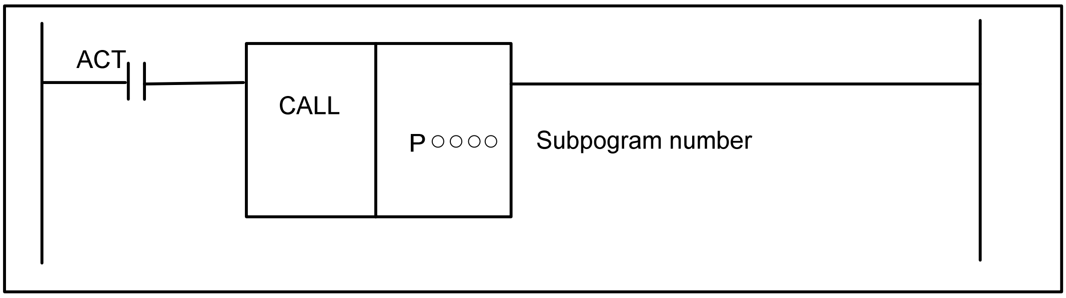


Fig. 5-62

Command table format ：

Table 5-40

No.	Command	Operand	Remark
1	LD	○○○○．○	ACT
2	FUNC	37	CALL
3	PRM	P○○○○	Subprogram number

Control conditions：

ACT execution conditions

ACT＝0: do not execute CALL command .

ACT＝1: execute CALL command，call the subprogram which number is specified.

Parameter：

Subprogram number ： specifies the called subprogram number. The subprogram number range is P1～P512.

5.39 CALLU（uncoditional subprogram call）

Function：

The system unconditionally calls the specified subprogram when it executes the command CALLU.

Format：

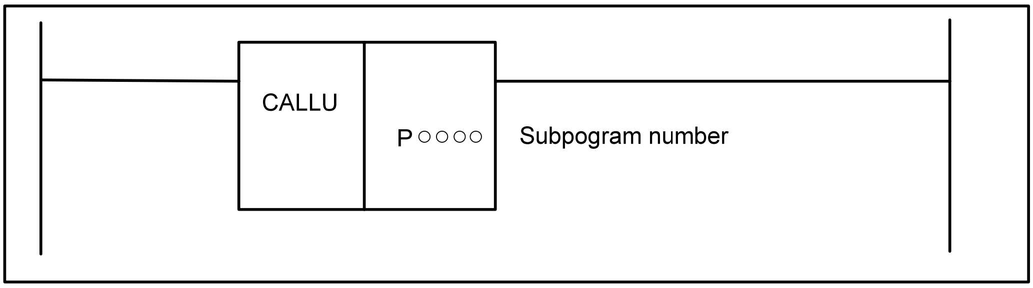


Fig. 5-63

Command table format ：

Table 5-41

No.	Command	Operand	Remark
1	FUNC	38	CALLU
2	PRM	P○○○○	Subprogram number

Parameter：

Subprogram number ： specifies the subprogram number of a subprogram to be called. The subprogram number must be specified in the P address form. A number from P1 to P512 can be specified.

5.40　JMPB（label jump 1）

Function：

The JMPB command transfers control to a ladder after the label set in a ladder pgoram.

The JMPB has the following additional functions:

* More than one jump command can be coded for the same label.
* The jump command can transfer control freely before and after the command within the program unit (main program or subprogram) in which the command is coded.
* Jump commands can be nested.
* Jump END1 and END2 are forbidden.

Format：

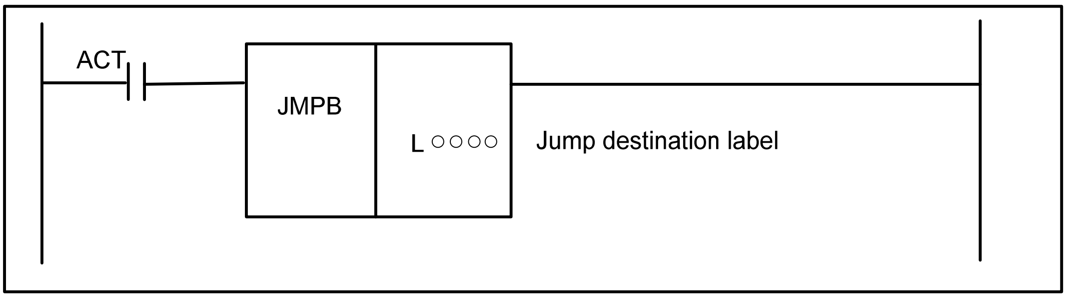


Fig. 5-64

Command table format　：

Table　5-42

No.	Command	Operand	Remark
1	LD	○○○○. ○	ACT
2	FUNC	39	JMPB
3	PRM	L○○○○	Jump destination label number

Control conditions：

ACT　　execution conditions

ACT＝0，do not jump,execute the next command after JMPB command.

ACT＝1，jump to the next after the specified label, execute the next command after the label.

Parameter：

Jump destination label LX: specifies the label of the jump destination. The label number must be specified in the L address head. A value from L1 to L9999 can be specified.

5.41 JMPC（label jump 2）

Function：

The JMPC functional command returns control from a subprogram to the label code position of the main program. The specifications of the JMPC command are the same as those of the JMPB command, except that JMPC always returns control to the main program.

Format：

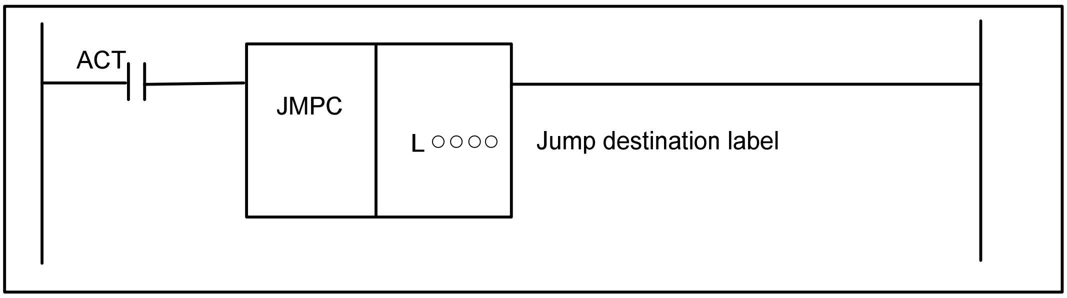


Fig. 5-65

Command table format ：

Table 5-43

No.	Command	Operand	Remark
1	LD	○○○○. ○	ACT
2	FUNC	40	JMPC
3	PRM	L○○○○	Jump destination label

Control conditions：

ACT execution condition

ACT=0：the command after the JMPC command is executed.

ACT=1：control is transferred to the ladder after the specified label.

Parameter：

Jump destination label: specifies the label of the jump destination. The label number must be specified in the L address head. A number from L1 to L9999 can be specified.

Note: when the command is used to jump bac to a previous command, care must be taken not to cause an infinite loop.

5.42 LBL（label）

Function：

The command specifies a label in ladder program for the jump destination of JMPB and JMPC.

Note: one Lx label only use LBL one time, otherwise, the system alarms.

Format:

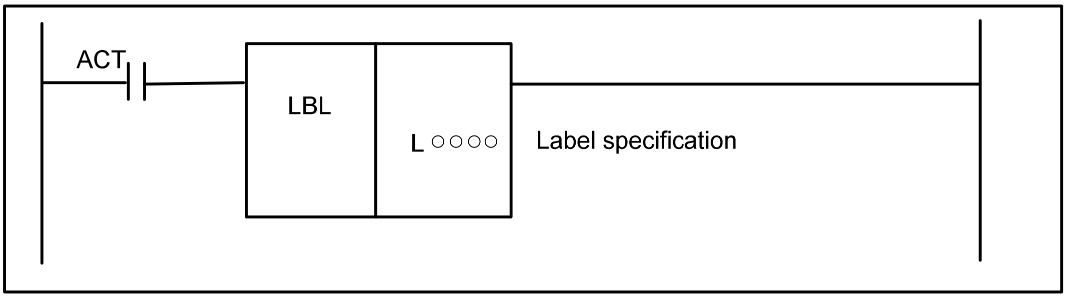


Fig. 5-66

Command table format :

Table 5-44

No.	Command	Operand	Remark
1	LD	○○○○. ○	ACT
2	FUNC	41	LBL
3	PRM	L○○○○	Label specification

Parameter:

Label specification Lx: specifies the jump destination.The label number must be specified in L address head. A label number from L1 to L9999 can be specified.

5.43 SP（subprogram）

Function:

The SP command is used to create a subprogram for CALL and CALLU call, and SP is used with the mentioned later SPE to specify the subprogram range.

Notes:

1. the subprogram must follow END2 to be compiled.
2. can not set another subprogram in one subprogram.

Format:

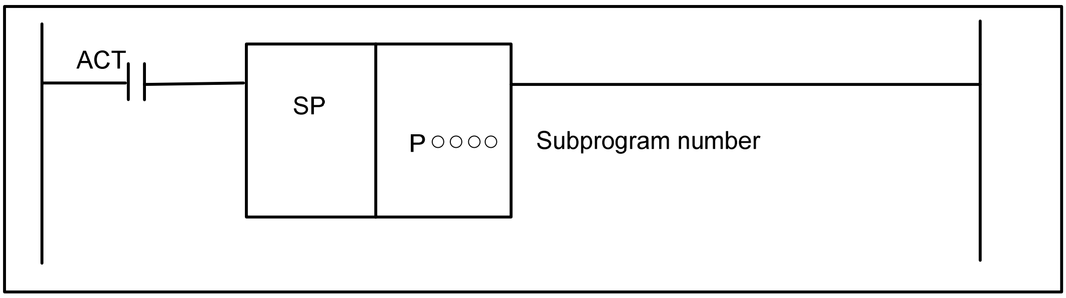


Fig. 5-67

Command table format :

Table 5-45

No.	Command	Operand	Remark
1	LD	○○○○. ○	ACT
2	FUNC	42	SP
3	PRM	P○○○○	Subprogram number

Parameter:

Subprogram number : specifies the called subprogram label number in the P address form. The subprogram number range is P1～P512, and the specified subprogram number must be unique within the sequence program.

5.44 SPE（end of a subprogram）

Function:

* SPE is used with the S P command to specify the subprogram range.

* when the functional command is executed, control is returned to the main program that calls the subprogram.

* the subprogram must follow END2 to be compiled.

Format:

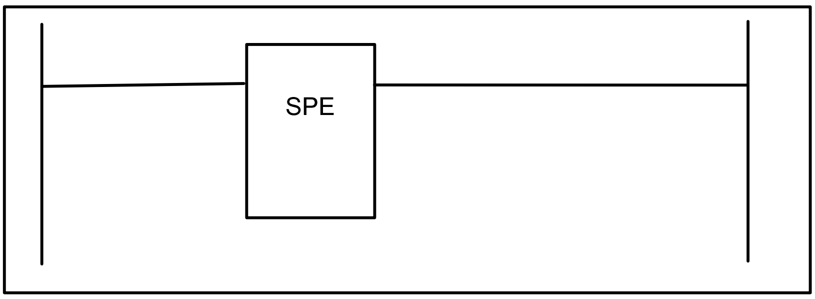


Fig. 5-68

Command table format :

Table 5-46

No.	Command	Operand	Remark
1	FUNC	43	SPE

6 Ladder Writing Limit

1. Sequence program must have END1 and END2 which are taken as the end marks of 1st level and 2nd level sequence part, and END1 must be before END2.
2. They only support the parallel output and do not support the multi-level output.
3. The result output address in all basic instructions and output function instruction are not set the following addresses:
   1) Counter preset address DC, timer preset address DT.
   2)) X address on IO input interface and CNC→PLC F address.

The followings are the phrasing error, and the system will alarm.

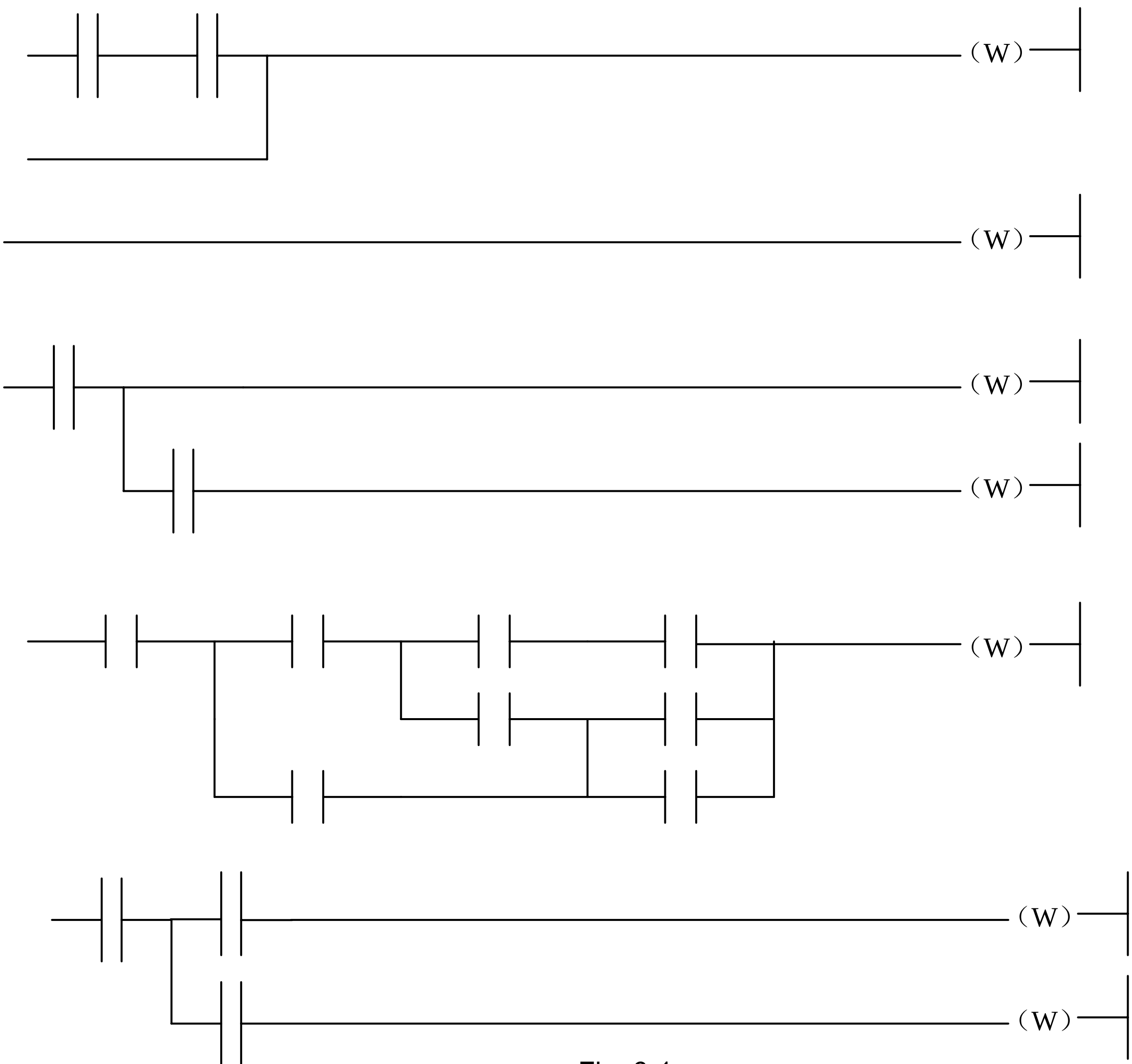


Fig. 6-1

Part 2 Function

1 Preparations for operatoin

1.1 Emergency stop

General When you press Emergency Stop button on the machine operation panel, the machine movement stops in a moment.

Emergency

Fig. 1-1

This button is locked whtn it is pressed. Althouth it varies with the machine tool builder, the button can usually be unlocked by twisting it.

Signal **emergency stop signal**

***ESP （ X008.4 G008.4 ）**

	＃7	＃6	＃5	＃4	＃3	＃2	＃1	＃0
X008				*ESP				
G008				*ESP				

［Classification］ input signal.

［Function］ activating an emergency stop signal stops the machine instantly.

［Operation］ when the emergency stop signal *ESP becomes to “0”, the emergency stop is applied to the machine and the CNC is reset.

1.2 CNC overtral signal

General When the tool tries to move beyond the stroke end set by the machine tool limit switch, the tool decelerates and stops as a result of tripping the limit switch, and an OVERTRAVEL is displayed. The signal can be output with an alarm.

Signal overtravel signal

＋*L1～＋*L5(G114#0~G114#4, X9.6, X10.0,X10.2,X10.4,X10.6)

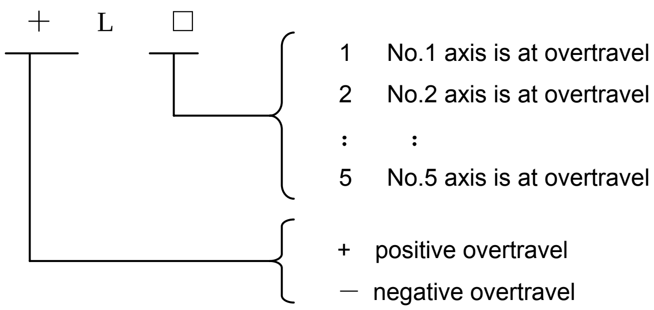
－*L1～－*L5(G116#0~G116#4, X9.7, X10.1, X10.3, X10.5, X10.7)

	#7	#6	#5	#4	#3	#2	#1	#0
G114				＋L5	＋L4	＋L3	＋L2	＋L1
G116				－L5	－L4	－L3	－L2	－L1

	#7	#6	#5	#4	#3	#2	#1	#0
X009	－L1	＋L1						
X010	－L5	＋L5	－L4	＋L4	－L3	＋L3	－L2	＋L2

［Classification］ input signal.

［Function］ indicates that the control axis has reached its stroke limit. There are individual signals for each direction in every control axis. The +/-in the signal name indicates the direction and the number corresponds to the control axis.



［Operation］ When it is “0”， the control unit operates as follows:

*In automatic operation, if even one axis overtravel signal becomes to “0”, all axes are decelerated to stop, an alarm is given and operation is halted.

*In manual operation, only the axis whose overtravel signal has turned to “0” is decelerated to a stop, and the axis can be moved in the opposite direction.

*Once the axis overtravel signal has turned to “0”, the axis direction is registered. Even if the signal returns to “1”, it is not possible to move that axis in that direction until the alarm is cleared.

1.3 Alarm signal

General When an alarm is triggered in the CNC, the alarm is displayed on the screen, and the alarm signal is set to 1.

Signal alarm signal

AL（F001＃0）

	＃7	＃6	＃5	＃4	＃3	＃2	＃1	＃0
F001								AL

［Classification］ output signal.

［Function］ the alarm signal reports that the CNC is in an alarm state:

a) NC alarm

b) Overtravel alarm

c) Servo alarm

［Output condition］ The alarm signal is set to 1 when:

——The CNC is placed in the alarm state.

The alarm signal is set to 0 when:

——The alarm has been released by resetting the CNC.

1.4 Interlock

General These signals disable machine movement along axes. When any of these signals is activated during movement, tool movement along the affected axis is decelerated, then stopped.

All axes interlock signal

***IT (G008＃0)**

［Classification］ input signal.

［Function］ This signal is used to inhibit the machine from moving. When the *IT is “0”, the axis movement is decelerated and stopped. In automatic operation, the system stops in automatic run state(the cycle start signal STL is “1”, the signal SPL is “0”).

Signal address

G008								*IT

Interlock signal for each axis

+MIT1~+MIT5 (G132＃0~G132#4)

-MIT1~-MIT5 (G134#0~G134#4)

［Classification］ input signal.

［Function］ inhibit the specified axis to specify the axis movement.

［Operation］ when the axial interlock signal becomes “1”, the CNC applieds interlock only in the corresponding axial direction. However, during automatic operation, all axes will stop.

Signal address

G132				+MIT5	+MIT4	+MIT3	+MIT2	+MIT1
G134				-MIT5	-MIT4	-MIT3	-MIT2	-MIT1

1.5 Operation mode selection

Signal Mode selection signal

F003#0~F003#7

[Classification**]** Output signal.

[Function**]** The currently selected operation mode is output.

	#7	#6	#5	#4	#3	#2	#1	#0
F003	MZRO	MEDT	MMEM	MRMT	MMDI	MJ	MH	MINC

2 Manual operation

2.1 JOG feed/incremental feed

General

JOG feed In JOG mode, setting a feed axis and direction selection bit to 1 on the machine operator's panel moves the machine along the selected axis in the selected direction.

Incremental feed In incremental feed mode, setting a feed axis and direction selection bit to 1 on the machine operator's panel moves the machine one step along the selected axis in the selected direction. The minimum distance the machine moves, is the least input increment. The step can be 10, 100, or 1000 times the least input increment.

The only difference between JOG feed and incremental feed is the method of selecting the feed distance. In JOG feed, the machine continues to be fed while the following signals selecting the feed axis and direction are 1: ＋J1，－J1，＋J2，－J2，＋J3，－J3，etc. In incremental feed, the machine is fed by one step. Using JOG feedrate override dial can regulate JOG feedrate. The step distance can be selected by MPG feed movement distance G19＃4~G19＃5.

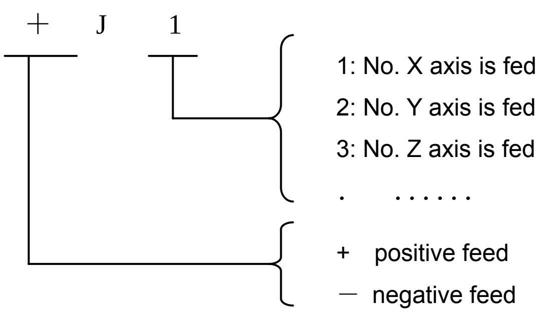
Signal Feed axis and direction selection signal

＋J1~＋J5（G100＃0~G100＃4）

－J1~－J5（G102＃0~G102＃4）

［Classification］ Input signal.

［Function］ In JOG feed or Incremental feed mode, select the required feed axis and direction. +/- in the signal name indicates the feed direction, the number corresponds to the controlled axis.



［Operation］When the signal is set to 1, the control unit operate as follows:

* When JOG feed or incremental feed is allowed, the control unit moves the specified axis in the specified direction.
  When the signal is set to 1 in JOG feed, the control unit continues to move that axis

* In incremental feed, the control unit feeds the requested axis by the step distance which is specified by the manual handle feed move distance selection signal, then the axis stops. Even if the signal is set to 0 while the axis is being fed, the control unit does not stop moving. To feed the axis again set the signal to 0, then to 1 again.

Manual rapid traverse selection signal

RT（G１９#7）

［Classification］ input signal.

［Function］ Select the rapid traverse rate in JOG feed or incremental feed mode.

［Operation］ When the signal becomes 1, the control unit operates as follows:

- The control unit executes the jog feed or incremental feed at a rapid traverse rate. The rapid traverse override is valid.
- When the signal is switched from 1 to 0 or vice versa in jog feed or incremental feed, the feedrate is decelerated until it reaches zero, then increased to the specified value. During acceleration and deceleration, the feed axis and direction selection signal can be kept 1.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G19	RT							
G100				+J5	+J4	+J3	+J2	+J1
G102				−J5	−J4	−J3	−J2	−J1

2.2 MPG / Step feed

General In MPG/Step feed mode, the machine moves by rotating the manual pulse generator(MPG) or Step. Select the axis along which the machine moves with the MPG feed axis selection signal/axis move signal.

Signal （**G018＃0~G018＃3**）

［Classification］ Input signal.

［Function］ Select MPG feed axis.

Signal **MPG/incremental feed selection signal**

（**G019＃4~G19＃5**）

［Classification］ Input signal.

［Function］ The signal selects the movement distance of each pulse of MPG in MPG feed, and also selects the movement distance of each step in the incremental feed.

3 Reference Point Return

3.1 Manual reference point return

General In manual reference point return mode, the machine tool move in the specified direction by setting the position parameter N1004#6 to execute the reference point return. The selected axis on the panel reports the axis to execute the machine zero return, which is not related to the move direction of axis.

The following signals are related to the manual reference point return:

	Manual reference point return
Reference point return selection	MREF
Reference point return deceleration signal	*DEC1~*DEC5
Reference point return completion signal	ZP1~ZP5
Creating reference point signal	ZRF1~ZRF5

Signal Reference point return completion signals

MREF（F004＃5）

［Classification］ Output signal.

［Function］ This signal indicates that manual reference point return has been selected.

［Output condition］ The signal turns to “1” when:

* Manual reference point return has been selected.

The signal turns to “0” when:

* The selection of manual reference point return has terminated.

Reference point return completion signal

ZP1～ZP5(F94＃0~F94＃4)

［Classification］ Output signal.

［Function］ These signals report that the machine tool is at the reference point on a controlled axis.

ZP1	1st axis reference point return completion signal
ZP2	2nd axis reference point return completion signal
ZP3	3rd axis reference point return completion signal
ZP4	4th axis reference point return completion signal
ZP5	5th axis reference point return completion signal

［Output conditions］When these signals becomes 1:

- Manual reference point return is completed and the current position is in the in-position area.
- The automatic reference point return(G28) is completed and the current position is in the in-position area.
- The reference point return check is completed and the current position is in the in-position area.

When the signal becomes 0:

- The machine tool moves from the reference point.
- The emergency stop signal appears.
- The servo alarm appears.

Reference point return deceleration signal

***DEC1~ *DEC5（G122＃0～G122#4, X9＃0~X9＃4）**

［Type］ Input signal.

［Function］These signals decelerate the feedrate for manual reference point return to a low feedrate in order to approach the reference point at the low feedrate.

Reference point establishment signal

ZRF1~ZRF4(F120＃0~F120＃4)

［Classification］ Output signal.

［Function］ Notify the system that the reference point has been established.

ZRF1	1st reference point establishment signal
ZRF2	2nd reference point establishment signal
ZRF3	3rd reference point establishment signal
ZRF4	4th reference point establishment signal
ZRF5	5th reference point establishment signal

［Output condition］ The signal becomes 1 when :

- When the reference point is established after the manual reference point return.
- When the reference point is established using the absolute position detector at initial power-on.

The signal becomes 0 when :

- When the reference point is lost.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
X009				*DEC5	*DEC4	*DEC3	*DEC2	*DEC1
F004			MREF					
F094				ZP5	ZP4	ZP3	ZP2	ZP1
F120				ZRF5	ZRF4	ZRF3	ZRF2	ZRF1
G122				*DEC5	*DEC4	*DEC3	*DEC2	*DEC1

4 Automatic operation

4.1 Cycle start/feed hold

General

* Start of automatic operation(cycle start)

When automatic operation start signal ST is set to 1 then 0 while the CNC is in memory mode, DNC operation mode or MDI mode, the CNC enters the automatic operation start state then starts operating.

The signal ST is ignored as follows:

1. When the CNC is in other modes except for MEM, RMT or MDI mode.
2. When the feed hold signal (SP) is set to 0.
3. The emergency stop signal (ESP) is set to 0.
4. When the reset signal（ERS）is set to 1.
5. When <RESET> on MDI panel is pressed.
6. When CNC is in the state of alarm.
7. When the automatic operation is started.

In automatic operation, the CNC enters the feed hold and stops run as follows:

1. When the feed hold signal (*SP) is set to 0.
2. The operation mode becomes manual operation mode.

In automatic operation, the CNC enters the feed hold and stops run as follows:

1. The single block instruction is end when the single block is running.
2. MDI operation is completed.
3. CNC alarms.
4. The single block instruction is end after the mode is changed to others or Edit mode.

In automatic operation, the CNC enters the reset and stops running as follows:

1. When the emergency stop signal (ESP) is set to 1.
2. When the external reset signal (ERS) is set to 1.
3. When <RESET> on MDI panel is pressed.

* Halt of automatic operation

（Feed hold） When the feed hold signal SP is set to 1 in automatic operation, the CNC enters the feed hold state and stops operation. At the same time, cycle start lamp signal STL is set to 0 and feed hold lamp signal SPL is set to 1. Re-setting signal SP to 0 in itself will not restart automatic operation. To restart automatic operation, first set signal SP to 0, then set signal ST to 1 and to 0.

When signal * SP is set to 0 during the execution of a bloc containing only the M, S, T, or B function, signals STL is immediately set to 0, signal SPL is set to 1, and the CNC enters the feed hold state. If the FIN signal is subsequently setn from the PLC, the CNC executes processing up until the end of the block that has been halted. Upon the completion of that block, signal SPL is set to 0 (signal STL remains set to 0) and the CNC enters the automatic operation stops state.

1. During threading

   When signal SP is set to 0 during threading, the CNC enters the feed hold state after executing a non-thread block after the threading blocks.

   When signal SP is set to 0 during threading with the G92 command (thread cycle), signal SPL is immediately set to 1 but operation continues up until the end of the retraction bloc following thread.

   When signal SP is set to 0 during threading with the G32 command, signal SPL is immediately set to 1 but operation continues until the end of a non-threading block following the threading blocks.

2. During tapping in a canned cycle

   When signal SP is set to 0 during tapping in a canned cycle (G84), signal SPL is immediately set to 1 but operation continues until the tool returns to the initial level or R point level after the completion of tapping.

3. When a macro command is being execued

   Operation stops after the currently executing macro command has been completed.

Signal **Cycle start signal**

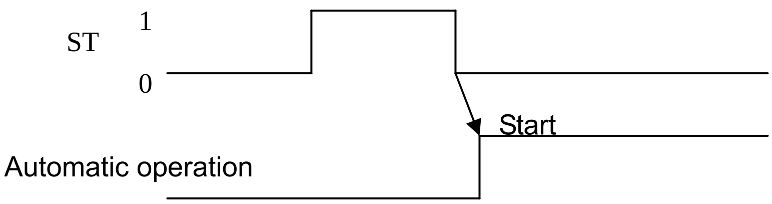
ST（G7＃2）

［Classification］ Input signal.

［Function］ Start the automatic operation.

［Operation］ When signal ST is set to 1 then 0 in automatic operation(Auto), DNC and MDI mode, the CNC enters the cycle start state and starts operations.

In Auto，DNC and MDI mode



Feed hold signal

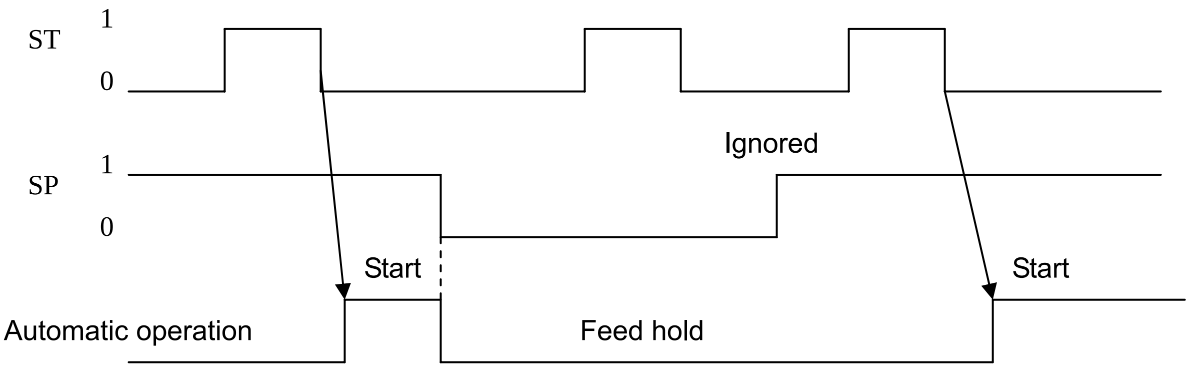
SP（G8＃5）

［Classification］ Input signal.

［Function］ Halt the automatic operation

［Operation］ In Auto mode, SP signal is set 1, CNC enters the feed hold and stops running. When SP signal is set to 0, the automatic operation does not start.

In Auto，DNC and MDI mode



Feed hold lamp signal

STL（F000＃5）

［Type］ Output signal.

［Function］ The signal reports PLC that the feed hold is entered.

［Output conditions］ The signal is set to 1 or 0, which is determined by CNC state as Table 4-1.

Feed hold lamp signal

SPL（F000＃4）

［Classification］ Output signal.

［Function］ The signal reports PLC that the feed hold is entered.

［Output conditions］ The signal is set to 1 or 0, which is determined by CNC state as Table 4-1

OP（F000＃7）

［Classification］ Output signal.

［Function］ The signal reports PLC that the feed hold is entered.

［Output conditions］ The signal is set to 1 or 0, which is determined by CNC state as Table 4-1.

Table 4-1

	Cycle start lamp STL	Feed hold lamp SPL	Automatic operation lamp OP
Cycle start	1	0	1
Feed hold	0	1	1
Automatic operation stopping	0	0	0
Reset	0	0	0

Signal address

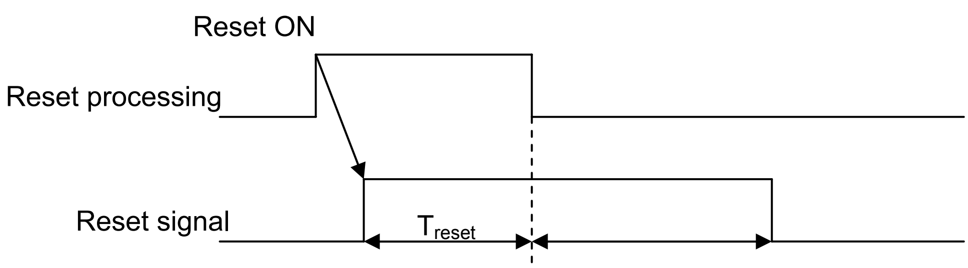
	＃7	＃6	＃5	＃4	＃3	＃2	＃1	＃0
G007						ST		
F000	OP		STL	SPL				

4.2 reset

General CNC is reset and enters the reset state.

1. When the emergency signal (ESP) is set to 1.
2. When the external reset signal（ERS）is set to 1.
3. When <RESET> on MDI panel is pressed.

When the CNC is reset, the resetting signal RST is output to the PLC. The resetting signal RST is set to 0 when the resetting signal output time has elapsed after the above conditions have been released.



When the CNC is reset during automatic operation, automatic operation is stopped and is decelerated to stop. When the CNC is reset during the execution of the MF, SF or TF signal is set to 0 within 16ms.

Signal **External reset signal**

ERS（G8＃7）

［Classification］ Input signal.

［Function］ reset the CNC.

［Operation］ turning the signal ERS to 1 resets the CNC and enters the reset state. While the CNC is reset, the resetting signal RST turns to 1.

Reset signal

RST（F001＃1）

［Classification］ Output signal.

［Function］ Notifies the PLC that the CNC is being reset. This signal is used for reset processing on the PLC.

［Output condition］ The signal is set to 1 when:

1. When the emergency stop signal (ESP) is set to 1.
2. When <RESET> on MDI panel is pressed.
3. <RESET> key on MDI is pressed.

The signal is set to 0 when:

When the reset signal output time set by a parameter is completed after the above are released and CNC is reset.

Signal address

	＃7	＃6	＃5	＃4	＃3	＃2	＃1	＃0
G008	ERS							
F001							RST	

4.3 Testing a program

General Before machining is started, the automatic running check can be executed. It checks whether the established program can operate the machine as desired. This check can be accomplished by running the machine or view the position display change without running the machine.

4.3.1 Machine tool lock

General The change of the position display can be monitored without moving the machine.

When all-axis machine lock signal MMLK is set to 1, output pulses to the servo motors are stopped in manual or automatic operation. The instructions are distributed, however, updating the absolute and relative coordinates. The operator can therefore check if the instructions are correct by monitoring the position display.

Signal **machine lock signal**

MLK G044＃1)

［Classification］ Input signal.

［Function］ The signal reports PLC of the state of all-axis machine tool lock signal.

［Operation］ When this signal is set to 1, pulses are not output to the servo motors for all axes in manual or automatic operation.

All-axis machine lock check signal

MMLK（F004＃1）

［Classification］ Output signal.

［Function］ Notifies the PLC of the state of the all-axis machine lock signal.

［Output condition］When the signal is set to 1, all-axis machine tool lock signal is set to 1.

When the signal is set to 0, all axes machine tool lock signals are set to 0.

Signal address

	＃7	＃6	＃5	＃4	＃3	＃2	＃1	＃0
F004							MMLK	
G044							MLK	

4.3.2 Dry run

General Dry run is valid only for automatic operation. The tool is moved at a constant feedrate regardless of the federate specified in the program. The feedrate is set by the data parameter P1210.

This function is used to check the movement of the tool without a workpiece.

Signal Dry run signal

DRN（G046＃7）

［Classification］ Input signal.

［Function］ Enables dry run.

［Operation］ When the signal is set to 1, the machine tool moves at the feedrate specified for dry run.

When the signal is 0, the machine tool normally moves.

Caution:

When the dry run signal is changed from 0 to 1 or 1 to 0 during the movement of the machine, the feedrate of the machine is first decelerated to 0 before being accelerated to the specified feedrate.

Dry run check signal

MDRN（F002＃7）

［Classification］ Output signal.

［Function］ Notifies the PLC of the state of the dry run signal.

［Operation］ The signal is set to 1 in the following case:

——When the dry run signal DRN is set to 1.

The signal is set to 0 in the following case:

——When the dry run signal DRN is set to 0.

Signal address

	＃7	＃6	＃5	＃4	＃3	＃2	＃1	＃0
G046	DRN							
F002	MDRN							

4.3.3 Single block

General The single block operation is valid in automatic operation mode (Auto mode).

When the single block signal (SBK) is set to 1 during automatic operation, the CNC enters the automatic operation stop state after executing the current block. In subsequent automatic operation, the CNC enters the automatic operation stop state after executing each block in the program. When the single block signal (SBK) is set to 0, normal automatic operation is stored.

Signal **Single block signal SBK（G046＃1）**

［Classification］ Input signal.

［Function］ Enables single block operation.

［Operation］ Execute the single block when the signal is set to1.

Execute the normal operation when the signal is set to 0.

Single block check signal

MSBK（F004＃3）

［Classification］ Output signal.

［Function］ The signal reports PLC of the state of single block signal.

［Operation］ The signal is set to 1 as follows:

——When the single block signal SBK is set to1.

The signal is set to 0 as follows:

——When the single block signal SBK is set to 0.

Caution:

1. Operations in thread cutting

   When the SBK signal becomes 1 in thread cutting, the operation stops after the first non-thread cutting signal after thread cutting instruction.

2. Operation in canned cycle

   When the SBK signal becomes 1 during canned cycle operation, the operation stops at each positioning, approach, drilling and retraction instead of the end of the block. The SPL signal becomes 1 while the STL signal becomes 0, showing that the end of the block has not been reached. When the execution of one block is completed, the STL and SPL signals become 0 and the operation is stopped.

Signal address

	＃7	＃6	＃5	＃4	＃3	＃2	＃1	＃0
G046							SBK	
F004					MSBK			

4.4 Optional block skip

General When a slash followed by a number is specified at the head of a block, and optional block skip signal BDT is set to 1 during automatic operation, the block is ignored**.**

Signal Skip optional block signal

BDT（G044#0）

［Classification］ Input signal.

［Function］ Select whether a block with “/” is neglected.

［Operation］ During automatic operation, when BDT is 1, the block with “/” is neglected.

The program is normally executed when BDT is 0.

Optional block skip check signal

MBDT（F004#0）

［Classification］ Output signal.

［Function］ The signal reports PLC of the state of skip optional block BDT.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G044								BDT
F004								MBDT

4.5 Program restart

General A program may be restarted at a block by specifying the sequence number of the block, after automatic operation is stopped because of a broken tool or for holidays. This function can also be used as a high-speed program check function.

There are two types of restart methods：

P type：restart after a tool is broken down.

Q type：restart after holiday.

Signal Program restart signal

SRN<G006#0>

[Classification**]** Input signal

[Function] Select the program restart

[Operation] When the program restart signal is set to 1 to search for the sequence number of the block to be restarted, the CRT screen changed to the program restart screen. When the program restart signal is set to 0, and automatic operation is activated, the machine moves back to the machining restart point at dry run speed along the axes one by one. When the machine moves to the restart point, machining restarts.

Signal during program restart

SRNMV<F002#4>

[Classification**]** Output signal

[Function**]** Report the program is started.

[Output conditions**]** The signal becomes 1 when:

—The program restart signal is set to 0 after the CRT screen changes to the program restart screen.

The signal is set to “0” when ：

—The program restart sequence ends(the tool has been moved to the restart point on all controlled axes）.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G006								SNR
F002				SRNM				

5 Feedrate Control

5.1 Rapid traverse rate

General A rapid traverse rate is set for each axis by the data parameter P1225, so no rapid traverse rate need be programmed.

The following overrides can be applied to a rapid traverse rate with the rapid traverse override signal:

F0，25％，50％， 100％.

F0 ：it is set by the data parameter P1231.

Signal **rapid traversing signal**

RPDO（F002#1）

［Type］ Output signal.

［Function］ The signal indicates that a move command is executed at rapid traverse.

［Output condition］ "1" indicates that an axis starts moving after rapid traverse has been selected.

"0" indicates that an axis starts moving after a federate other than rapid traverse has been selected. This holds true for both automatic and manual operation modes.

Note：

1. The rapid traverse in automatic operation includes all rapid traverses in cannced cycle positioning, automatic reference point return, etc., as well as the move command G00. The manual rapid traverse also includes the rapid traverse in reference position return.
2. Once rapid traverse has been selected, this signal remains "1", including durig a stop, until another federate has been selected and movement is started.

Signal address

	＃7	＃6	＃5	＃4	＃3	＃2	＃1	＃0
F002							RPDO	

5.2 Override

5.2.1 Rapid traverse override

General An override of four steps (F0，25％，50％，100％) can be applied to the rapid traverse rate. F0 is set by a parameter P1231.

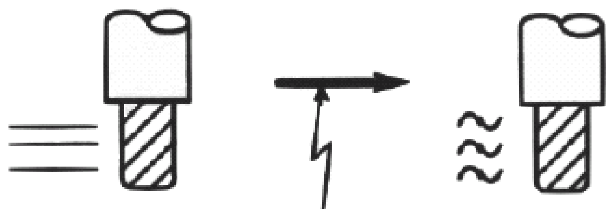


Fig. 5-1

Feedrate Actual feedrate is obtained by multiplying the rapid traverse rate preset by a parameter by the override value determined by this signal (including manual reference point return, program zero return).

F0 rate It is set by the data parameter P1231.

Signal rapid traverse rate override signal ROV1 ROV2<G14.0 G14.1>

Rapid traverse override signal		Override value
ROV2	ROV1	
0	0	F0
0	1	25%
1	0	50%
1	1	100%

Fig. 5-2

［Classification］ Input signal.

［Function］ These signals override the rapid traverse rate.

5.2.2 Override cancel

General The override cancel signal fixes the feerate override to 100%.

Signal Override cancel signal

OVC（G006＃4）

［Classification］ Input signal.

［Function］ The feedrate override is fixed to 100％.

［Operation］ When the signal is 1, CNC operates as follows:

- The feedrate override is fixed to 100% irrespective of the feedrate override signal.
- Rapid traverse override and spindle speed override are not affected.

Signal address

	＃7	＃6	＃5	＃4	＃3	＃2	＃1	＃0
G006				OVC				

6 Auxiliary Function

6.1 Miscellaneous function

General miscellaneous function (M code) When the maximum 3-digit number following the address M is specified, a code signal and a strobe signal are set to the machine. The machine uses these signal to turn on or off its functions.

Basic procedure

The following signals are used for the following functions.

Table 6-1

<table>
<tr><td rowspan="2">Function</td><td rowspan="2">Program address</td><td colspan="3">Output signal</td><td rowspan="2">Completion signal</td></tr>
<tr><td>Code signal</td><td>Strobe signal</td><td>Distribution end signal</td></tr>
<tr><td>Miscellaneous function</td><td>M</td><td>M**</td><td>MF</td><td rowspan="3">DEN</td><td rowspan="3">FIN</td></tr>
<tr><td>Spindle function</td><td>S</td><td>S00～S31</td><td>SF</td></tr>
<tr><td>Tool function</td><td>T</td><td>T00~T31</td><td>TF</td></tr>
</table>

Each function uses different program addresses and different signals, but they all input and output signals in the same way, as described below.(A sample procedure for the miscellaneous function is described below. The procedures for the spindle speed function and the tool function are obtained simply by substituting S, T in place of M.)

（1） Suppose that MXXX is specified during a program:
If XXX is not set，the CNC alarms.

（2） After the code signals M00~M31 is sent, the strobe signal MF is set to 1. The code signal is the binary representation of the programmed value XXX.
If a move, dwell, spindle speed, or other function is specified in the same block as the miscellaneous function, the execution of the other function is started when the code signal of the miscellaneous function is sent.

（3） When the strobe signal is set to 1, the PLC reads the code signal and performs the corresponding operation.

（4） To execute an operation after the completion of the move, dwell or other function specified in the block, wait until distribution end signal DEN is set to 1.

（5） Upon completion of the operation, the PLC sets completion signal FIN to 1. The completion signal is used by the miscellaneous function, spindle speed function, tool function described later, and other functions. If any of these functions are executed simultaneously, the completion signal must be set to 1 upon completion of all the functions.

（6） If the completion signal remains set to 1 for longer than period, the CNC sets the strob signal to 0 and reports that the completion signal has been received.

（7） When the strobe signal is set to0, set the completion signal to 0 in the PLC.

（8） When the completion signal is set to 0, the CNC sets all code signals to 0 and completes all sequences of the miscellaneous function.

（9） Once all other commands in the same block have been completed, the CNC executes the next block.

1. When the spindle speed is executed, the tool function is S code, T code signal is sent.

2. When the spindle speed, the tool function code signal is maintained until a new code for the corresponding function is specified.

The timing diagram is as follows:

One miscellaneous function specified in a block

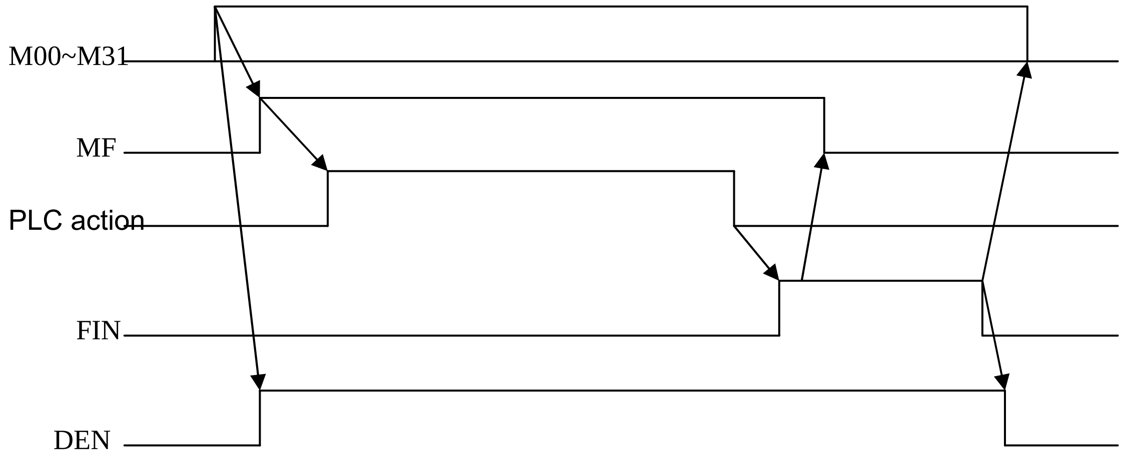


Fig. 6-1

Move command and miscellaneous function in the same block, execute a miscellaneous function with waiting for move command completion:

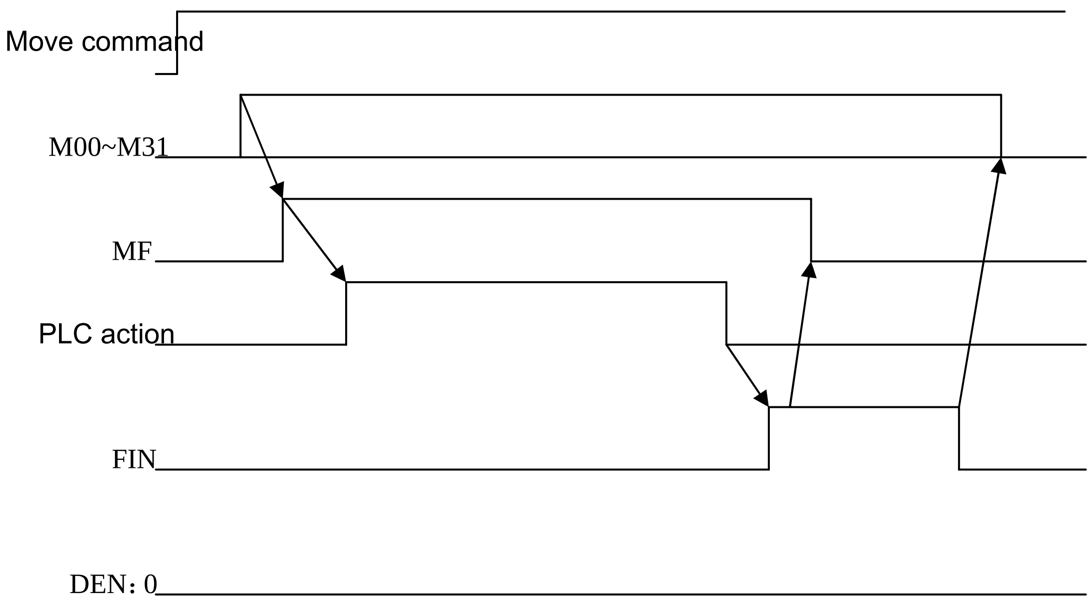


Fig. 6-2

Move command and miscellaneous function in the same block, execute a miscellaneous function with waiting for move command completion:

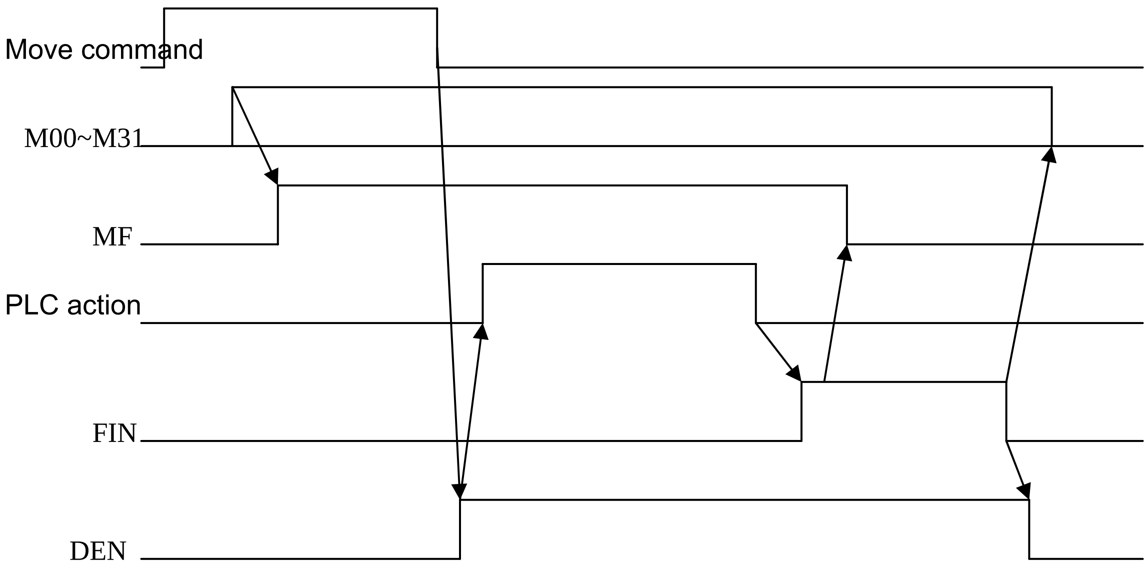


Fig. 6-3

Signal **Miscellaneous function code signals**

M00～M31（F010～F013）

Miscellaneous function strobe signal

MF（F007＃0）

［Classification］ Output signal.

［Function］ These signals report the specification of miscellaneous functions.

［Output condition］ For the output conditions and procedure, see the description of "Basic procedure" above.

Note: 1．The following miscellaneous functions are only processed in the CNC; they are not output to the PLC when programmed:

＊ M98，M99，

＊ M code that calls a subprogram

＊ M code that calls a custom macro

2．Decode signals as well as the code signals and strobe signal are output for the miscellaneous functions listed below..

M00，M01，M02，M30

3． M00～M31 are output to M code in the binary BCD format.

For example: M5 corresponds to 00000000, 00000000, 00000000, 00000101.

M decoding signal

DM00（F009＃7）

DM01（F009＃6）

DM02（F009＃5）

DM30（F009＃4）

［Classification］ Output signal.

［Function］ These signals report particular miscellaneous function are specified. The miscellaneous functions in a command program correspond to output signals as indicated below.（Table 6-2）:

Table 6-2

Program command	Output signal
M00	DM00
M01	DM01
M02	DM02
M30	DM30

［Output condition］ A decode M signal goes “1” when :

- The corresponding miscellaneous function is specified, and any move commands and dwell commands specified in the same block are completed. These signals are not output when the end signal of the miscellaneous function is returned before completion of such move command and dwell commands.

A decode M signal goes “0” when :

- FIN signal goes “1”.
- Reset occurs.

Spindle speed code signals

S00～S31（F022～F025）

Spindle speed strobe signal

SF（F007＃2）

［Classification］ Output signal.

［Function］ These signals report that spindle speed function have been specified.

［Output condition］For the output conditions and procedure, see the description of “Basic procedure” above.

Use S code output of the analog spindle.

Note: S00～S31 in the binary BCD format is output to S code.

For example: S4 corresponds to 00000000, 00000000, 00000000, 00000100.

Tool function code signal

T00～T31（F026～F029）

Tool function strobe signal

TF（F007＃3）

［Classification］ Output signal.

［Function］ These signals indicates the actually specified tool function.

［Output condition］For the output conditions and procedure, see the description of "Basic procedure" above.

Note： T00～T31 in the binary BCD format is output to T code.

For example: T3 corresponds to 00000000, 00000000, 00000000, 00000011.

Miscellaneous function end signal

FIN（G004＃3）

［Classification］ Input signal.

［Function］ These signals report that the end of the miscellaneous function, the spindle speed function, the tool function.

［Operation］ When the signal goes "1", for the operation and procedure, see the description of "Basic procedure".

Warning

Only one end signal is used for all functions above, the end signal must go "1" after all functions are completed.

Distribution end signal

DEN（F001＃3）

［Classification］ Output signal.

［Function］ These signals report that all commands (such as move commands and dwell) are completed except those miscellaneous functions, spindle speed function, tool functions and other commands (move command and pause command) are contained in the same block and have been set to the PLC FIN signal.

［Output condition］ The DEN signal turns to "1" when:

Waiting for the completion of miscellaneous functions, spindle speed function, tool functions and other commands in the same block are completed, and the current position is in the in-position.

The DEN signal turns to "0" when:

The execution of one block is completed.

Signal address:

	#7	#6	#5	#4	#3	#2	#1	#0
G004					FIN			
F001					DEN			
F007					TF	SF		MF
F009	DM00	DM01	DM02	DM30				
F030	M07	M06	M05	M04	M03	M02	M01	M00
F031	M15	M14	M13	M12	M11	M10	M09	M08
F032	M23	M22	M21	M20	M19	M18	M17	M16
F033	M31	M30	M29	M28	M27	M26	M25	M24
F022	S07	S06	S05	S04	S03	S02	S01	S00
F023	S15	S14	S13	S12	S11	S10	S09	S08
F024	S23	S22	S21	S20	S19	S18	S17	S16
F025	S31	S30	S29	S28	S27	S26	S25	S24
F026	T07	T06	T05	T04	T03	T02	T01	T00
F027	T15	T14	T13	T12	T11	T10	T09	T08
F028	T23	T22	T21	T20	T19	T18	T17	T16
F029	T31	T30	T29	T28	T27	T26	T25	T24

6.2 Auxiliary function lock

General Inhibits execution of a specified M, S, and T function. That is, code signals and strobe signals are not issued. This function is used to check a program.

Signal Auxiliary function lock signal

AFL（G05＃6）

［Classification］ Input signal.

［Function］ The signal selects the auxiliary function lock, i.e., the signal disables the execution of the specified M, S, T function.

［Operation］ When the signal becomes 1, the control unit functions are as follows:

1 . The control unit does execute M, S, T functions specified for memory operation, DNC operation, or MDI operation. That is, the control unit stops the output of code signals and strobe signals.

2. If this signal turns to “1” after code signal output, the output operation is executed in the ordinary manner until its completion (that is, until the FIN signal is received, and the strobe signal turns to “0”) .

3. Among the miscellaneous functions, M00,M01， M02 and M30 are executed even when this signal is “1” .

4. Among the miscellaneous functions, even when this signal is “1”, those functions(M98 and M99) that are executed in the control unit without outputting their execution results are executed in the ordinary manner.

Warning Even when this signal is “1”, spindle analog output or spindle serial output is executed.

Auxiliary function lock check signal

MAFL（F004＃4）

［Classification］ Output signal.

［Function］ The signal reports the state of auxiliary function lock signal AFL.

［Output conditions］ When the signal is 1, the auxiliary function lock signal AFL is1.

When the signal is 0, the auxiliary function lock signal AFL is 0.

Signal address

	＃7	＃6	＃5	＃4	＃3	＃2	＃1	＃0
G005		AFL						
F004				MAFL				

7 Spindle Speed Function

7.1 Spindle speed control mode

General **For 25i CNC System, the spindle is divided into gear spindle and analog spindle:**

1. In gear spindle mode, CNC changes S code to switch value to output to the spindle to control the spindle speed.
2. During analog spindle, changes S code to analog value to output to the spindle to control the spindle speed.

Spindle enabling signal

ENB<F001#4>

Another output related to spindle control is the spindle enable signal ENB.

The ENB signal is logical 1 when a nonzero command output is sent to the spindle. If the command is logical 0, the ENB signal becomes logical 0.

When the analog spindle is being used, an offset voltage in the spindle motor speed amplifier may cause the spindle motor to rotate at low speed even if the command output(in this case, analog voltage) to the spindle is zero. The ENB signal can be used to stop the motor in such a case.

	＃7	＃6	＃5	＃4	＃3	＃2	＃1	＃0
F004				ENB				

7.1.1 Gear spindle

General Gear spindle is defined that the actual spindle speed is controlled by the machine gear. So, the CNC outputs the switching inverted by S code to the machine gear to control the spindle speed.

Signal Spindle speed code signal

S00～S31（F022～F025）

Spindle speed strobe signal

SF（F007＃2）

［Classification］ Output signal.

［Function］ These signals report the actually specified the spindle speed function.

［Output condition］For the output condition and the procedure, see “Basic procedure” .

Use S code ouput of analog spindle.

Note: S00～S31 is output to S code in the binary BCD format.

S4 corresponds to 00000000, 00000000, 00000000, 00000100.

7.1.2 Analog spindle

General The analog spindle is defined that the spindle speed is controlled by the analog voltage value from CNC. So, CNC changes S code into the analog voltage value to output to the spindle of machine tool to control the spindle speed.

1. The actual output analog voltage value equals to the S value controlled by the spindle multiplying the spindle override.
2. CNC still reports the speed by S00~S31 signal but SF signal does not output.

Gear change processing：

Although S instructs the spindle speed, the actual is to control the spindle motor. So, CNC needs to confirm the corresponding relation between the spindle motor and gear. Like S instruction selection，CNC selects the gear according to the previously defined gear speed range by parameter to report PLC to select the corresponding the gear by using the gear change select signal (GR3, GR2, GR1). At the same time, CNC outputs the spindle motor speed according to the selected gear. CNC outputs the instruction corresponded to the spindle (GR1, GR2, GR3 output) speed by specifying S0～S99999 during MDI mode. 2 or 3 speed gear (GR1, GR2, GR3) is set simultaneously output to the gear select signal. When the gear select signal is changed, CNC simultaneously output SF signal).

Specification of gear change signal is as follows:（Table 7-1）：

Table 7-1

	No. 2 gear	No. 3 gear	Remark
GR1	Low	Low	Low: low gear
GR2	High	Medium	Medium: middle gear
GR3		High	High: high gear

Signal: Gear select signal

GR1,GR2,GR3

<F034#0～#2>

[Classification] Output signal.

[Function] These signals report PLC the selected gear.

[Output conditions] For the definition of these signals, see Gear change Mode.

Gear change select signal (input)

GR1,GR2<G028#1～#2>

[Classification] Input signal.

[Function] These signals report CNC the current selected gear.

[Output condition] For the definition of these signals, see Gear change Mode.

7.2 Spindle speed arrival signal

General The spindle speed arrival signal SAR is an input signal used as a condition for the CNC to start cutting feed. This signal is used generally when cutting feed should be started after the spindle reaches the specified speed. In this case, a sensor is used to check the spindle speed. The detected speed is sent to the CNC via the PLC. When the avoe operation is performed continuously using the PLC ladder, however, cutting feed may be started based on the SAR signal indicating the previous spindle start, if the spindle speed change command and the cutting feed command are issued at the same time.

Signal **Spindle speed arrival signal**

SAR（G029＃4）

［Classification］ Input signal.

［Function］ SAR signal controls the start of the cutting feed. When the signal is set to 0, the CNC starts the cutting feed.

［Operation］ Notifies that the CNC spindle has reached the specified spindle speed, so, the signal is set to 1 after the actual spinde speed reaches the specified value.

CNC checks SAR signal in the following states:

1. Before starting distribution of the first feed(move command) block after shifting from the rapid traverse mode to the cutting feed mode. This checking is performed after the time set by parameter No. 5113 has elapsed after the feed block is read.
2. Before starting distribution of the first feed command block after an S code is command. The wait time for checking is the same as in item 1.
3. When an S code and feed are programmed in the same block, the S code (or command output to the spindle) is output, and the SAR signal is checked after a fixed time elapses. If the SAR signal is set to “1”, feed begins.

Signal address

	＃7	＃6	＃5	＃4	＃3	＃2	＃1	＃0
G029				SAR				

7.3 Rigid tapping

General During a tapping cycle, synchronous control is applied to the tapping operation of a tapping axis and the operation of the spindle.

Namely, during rigid tapping (G74, G84), CNC needs to detect the rotation direction signal of spindle to confirm the cutting feed direction and machining process.

Procedure：

Spindle rotating→ Z tool infeed tapping→ transmit M05 to spindle→ wait for spindle to completely stop→ transmit CCW instruction→ starting point of Z tool retraction→ spindle stops rotating

So, to realize the rigid tapping, the corresponding ladder must be written to report the rotation direction of CNC external spindle.

Signal **rigid tapping signal RGTAP（G61＃0）**

［Classification］ Output signal.

［Function］ Reports to the servo to enter the rigid tappind mode.

［Output condition］ After the system executes the rigid tappind command, the system sends the signal to the servo that the CNC has entered the rigid tapping command. .

RGTAP 1：the current CNC is during the rigid tapping mode.

0：the current CNC is not during the rigid tapping

Signal address

	＃7	＃6	＃5	＃4	＃3	＃2	＃1	＃0
G061								RGTAP

Signal **spindle drive unit speed/position switch completion signal VPO 〈F076#3〉**

〔Classification〕Output signal

〔Function〕reports the PLC confirmation signal after the spindle drive unit completes entering the rigid tapping state.

〔Output condition〕when the system executes the rigid tapping command, PLC sends to the spindle drive unit to enter the rigid tapping state. After the spindle drive unit completes the rigid tapping switch to enter the rigid tapping state, the signal notifies the PLC that the spindle has completed the control switch to enter the rigid tapping state.

Signal address

	＃7	＃6	＃5	＃4	＃3	＃2	＃1	＃0
F076					VPO			

8 Tool function

8.1 T command tool change

The custom specifies T command to execute the tool change in automatic run and MDI mode. After the CNC explains the T command, sends the tool number and the strobe signal specified by T command and waits PLC to complete the tool change.

9 Programming command

9.1 Custom macro program

General Although subprograms are useful for repeating the same operation, the custom macro function also allows use of variables, arithmetic and logic operations, and conditional branches for easy development of general programs. A machining program can call a custom macro with a simple instruction, just like a subprogram.

This means that a functions of general use can be formed when programming a certain function as custom macro. That is, programs can be written using variables for data that might change or be unkown. This can be further applied to group technology.

Signal **User macro program input signal**

UI000~UI013（G054，G055, G056, G057）
UI100~UI113（G226，G227, G228, G229）
UI200~UI213（G230，G231, G232, G233）
UI300~UI313（G234，G235, G236, G237）

［Classification］ Input signal

［Function］ The signals do not provide any functions for the control unit. These signals which are taken as one of system variable is read by macro program, used for the interface signal between macro program and PLC.

The system variable corresponding to these signals are as follows:（Table 9-1）:

Table 9-1

Signals	Address	Q'ty	Variables
UI000	G54#0	1	#1000
UI001	G54#1	1	#1001
UI002	G54#2	1	#1002
UI003	G54#3	1	#1003
UI004	G54#4	1	#1004
UI005	G54#5	1	#1005
UI006	G54#6	1	#1006
UI007	G54#7	1	#1007
...	...		...
...	...	1	...
...	...	1	...
UI029	G57#5	1	#1029
UI030	G57#6	1	#1030
UI031	G57#7	1	#1031
UI000~UI031	G54~G57	32	#1032
UI100~UI131	G226~G229	32	#1033
UI200~UI231	G230~G233	32	#1034
UI300~UI331	G234~G237	32	#1035

Note: #1032 is variable with 32-bit as follows:

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
#1032	UI007	UI006	UI005	UI004	UI003	UI002	UI001	UI000
#1032	UI015	UI014	UI013	UI012	UI011	UI010	UI009	UI008
#1032	UI023	UI022	UI021	UI020	UI019	UI018	UI017	UI016
#1032	UI031	UI030	UI029	UI028	UI027	UI026	UI025	UI024

Custom macro program output signal

UO000～UO031（F054～F057）

UO100～UO131（F226～F229）

UO200～UO231（F230～F233）

UO300～UO331（F234～F237）

［Type］ Output signal.

［Function］ The signals do not provide any functions for the control unit. These signals which are taken as one of system variable are read/written by macro program, used for the interface signal between macro program and PLC.

The system variable corresponding to these signals are as follows (Table 9-2):

Table 9-2

Signals	Address	Q'ty	Variables
UO000	F54#0	1	#1100
UO001	F54#1	1	#1101
UO002	F54#2	1	#1102
UO003	F54#3	1	#1103
UO004	F54#4	1	#1104
UO005	F54#5	1	#1105
UO006	F54#6	1	#1106
UO007	F54#7	1	#1107
. . .	. . .		. . .
. . .	. . .	1	. . .
. . .	. . .	1	. . .
UO029	F57#5	1	#1129
UO030	F57#6	1	#1130
UO031	F57#7	1	#1131
UO000~UO031	F54~F57	32	#1132
UO100~UO131	F226~F229	32	#1133
UO200~UO231	F230~F233	32	#1134
UO300~UO331	F234~F237	32	#1135

Note: #1132 is variable with 32-bit variable as follows:

	＃7	＃6	＃5	＃4	＃3	＃2	＃1	＃0
#1132	UO007	UO006	UO005	UO004	UO003	UO002	UO001	UO000
#1132	UO015	UO014	UO013	UO012	UO011	UO010	UO009	UO008
#1132	UO023	UO022	UO021	UO020	UO019	UO018	UO017	UO016
#1132	UO031	UO030	UO029	UO028	UO027	UO026	UO025	UO024

Ⅱ CONNECTION

Notes

1. Machine electric box requirements

The machine electric boxes of the installation system and the drive unit use the fully closed dust-proof design to effectively protect the dust, the lubrication and the coolant from entering any internal components, and the temperature difference tween the inner and the outer of the electric box cannot exceed 10℃.

2. System installation position

CNC system is the control core of the whole CNC machine, and it is prior to be placed in the position where there is the small temperature increasing and the less electromagnetic radiation interference. The spindle drive unit with strong power and the feed axis drive unit should be installed on the upper because their much heat. I/O should be placed in the below.

3. Protective ground

Machine electric box should be grounded, the consecutive of the protective grounding should be meet with GB 5226.1-2008 requirements. It is necessary with the stable ground for the system stably running, each grounding wire of all components of the system cannot be series each other, and grounding bar (thickness ≥3mm copper) should be installed in the electric box, the grounding resistance of the ground connected with the grounding bar should be equal to or less than 0.1Ω, and the protective grounding terminal of each component should be separately connected with the grounding bar with the stubby yellow-green wiring.

4. Suppressing interference

Although the system uses the anti-interference in design to avoid the external interference influence, the following measures in the installation and connection should be executed to get the stable and reliable run.

a) use the insolated transformer to CNC power supply;

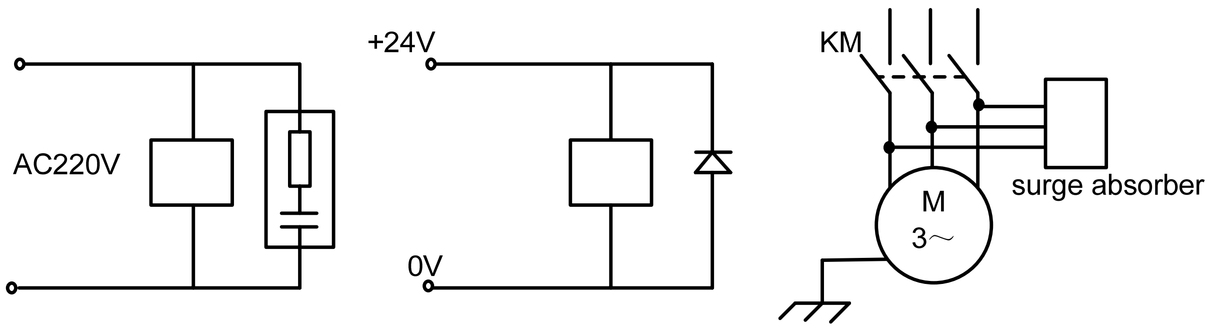
b) the installation of the CNC system should be far away from the ones bringing inference;

c) CNC signal should use the shield cable which should be far away from the power electromagnetic interferenece, and which should be straight, otherwise, which causes the interference signals;

d) Parallel RC circuit in AC coil, and the RC circuit should approach the inductive load;

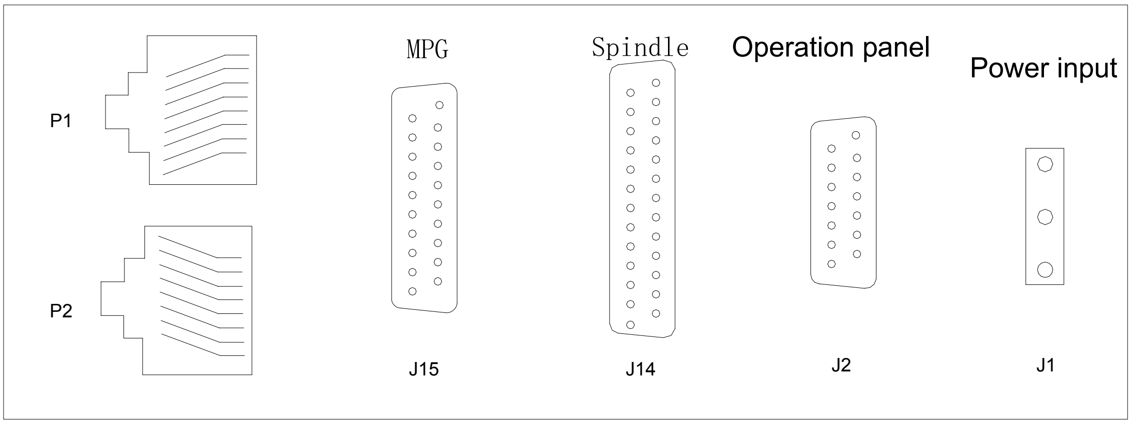
e) Inversely parallel freewheeling diode in the two terminals of DC coil;

f) Parallel surge absorber in AC motor winding terminal.



1 GSK25i System Box Interface

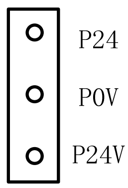
GSK 25i system box interface is as follows:



P1: Ethernet interface one	
pin explanation of one terminal of crystal plug	
pin No.	pin explanation
1	TX1+
2	TX1-
3	RX1+
4	NC
5	NC
6	RX1-
7	NC
8	NC

J2 :Operation panel interface			
1	P24V	2	
3	P0V	4	
5		6	RXD-
7	RXD+	8	
9		10	
11	0V	12	
13	TXD+	14	TXD-

J1:Power interface



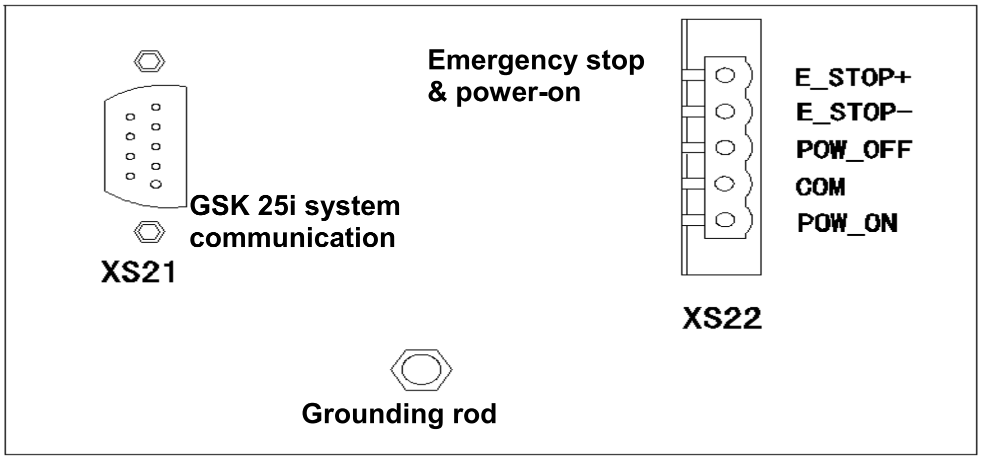
P2: Ethernet interface two	
pin explanation of another terminal of crystal plug	
pin No.	pin explanation
1	TX2+
2	TX2-
3	RX2+
4	NC
5	NC
6	RX2-
7	NC
8	NC

J15: MPG interface			
1	+5V	11	P_24V
2		12	
3	STP	13	
4	LED	14	PB-
5	HX	15	PB+
6	HY	16	PA+
7	HZ	17	PA-
8	H4	18	X100
9	H5	19	X1
10	P_0V	20	X10

J14:Spindle interface			
1	SVC+	14	
2		15	
3	SVC-	16	PB+
4	CP+	17	PB-
5	CP-	18	PA+
6	DIR-	19	PA-
7	DIR+	20	
8	ALM	21	P_0V
9	COIN	22	VP
10	ZSP	23	EN
11	VPO	24	STAO
12	SAR	25	ZSL
13	P_24V	26	ARST

2 Operation panel interface

2.1 Sketch map of machine operation panel interface



2.2 GSK 25i CNC system communication interface XS21

1	P24V	2	
3	P0V	4	
5		6	RXD-
7	RXD+	8	
9		10	
11	0V	12	
13	TXD+	14	TXD-

*TXD+, TXD-, RXD+, RXD- ：RS485 difference communication signal;

*0V：reference ground of difference signal；

*P24V，P0V：24V input

2.3 Emergency stop power-on interface

XS22 (5-male)

Emergency stop & power-on

1	E_STOP+
2	E_STOP-
3	POW_OFF
4	COM
5	POW_ON

3 I/O Interface

Sketch map of I/O interface

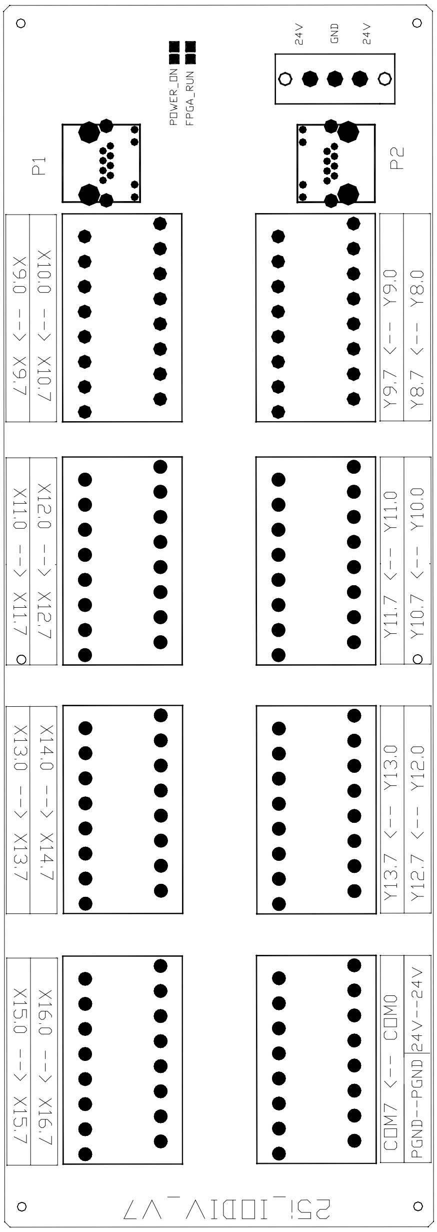


Fig. 3-1

①I/O power interface

XS34(3-male)

1	24v
2	0v
3	24v

Fig. 3-2

*0V：share with the corresponding groundi of the machine.

②P1, P2 are the industrial Ethernet interface

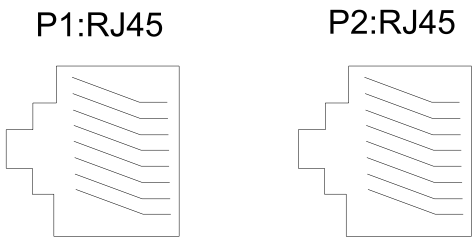


Fig. 3-3

③0V～24V terminal is the level selection one of the input signal COM, which determines whether the group of input signal is HIGH or LOW is valid：

（1）Marking COM0～COM7，orderly corresponds the common terminal of the input point X9～X16;

（2）When COM is connected with 24V，the corresponding input point being connected with LOW (0V) is valid;

（3）When COM is connected with 0V，the corresponding input point being connected with HIGH (24V) is valid.

Example:（1）COM0 is the common terminal of X9, i.e. input signal X9.0～X9.7are taken COM0 as the common terminal.

（2）When COM0 is connected with LOW（0V），the input point X9.0～X9.7being connected with HIGH（24V）is valid.

（3）When COM0 is connected with HIGH（24V），the input point X9.0～X9.7 being connected with LOW（0V）is valid

4 Interconnection Graph

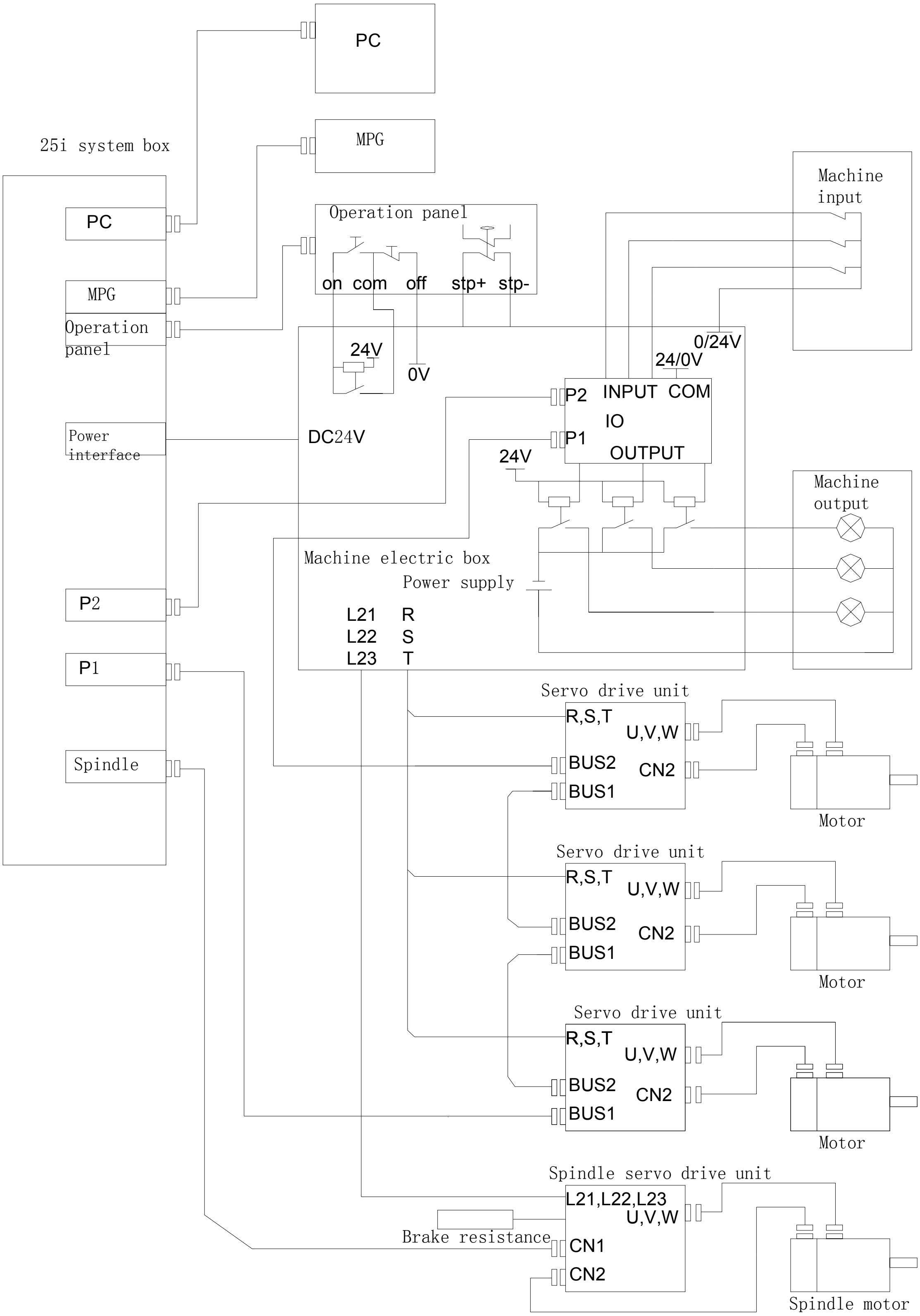


Fig. 4-1

5 PC serial communication wire

Communication connection between the system and PC RS232 is as Fig. 5-1.

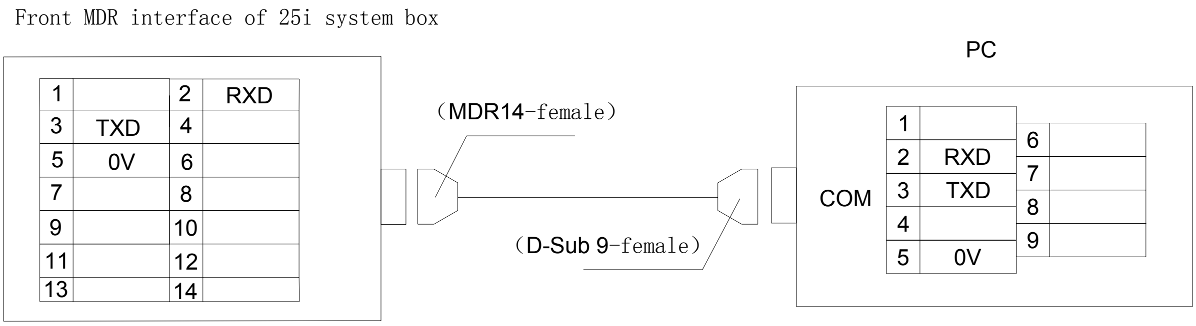


Fig. 5-1

PC communicatin cable connection is as Fig. 5-2.

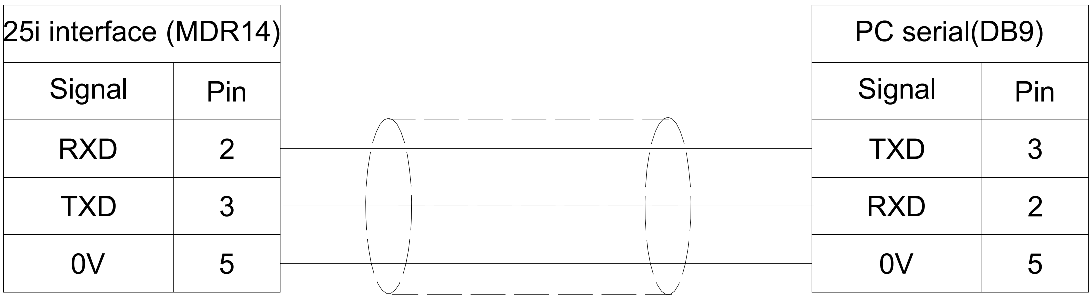


Fig. 5-2

6 MPG Wiring

25i system box

J15 MPG interface

1	+5V	11	P_24V
2		12	
3	STP	13	
4	LED	14	PB-
5	HX	15	PB+
6	HY	16	PA+
7	HZ	17	PA-
8	H4	18	X100
9	H5	19	X1
10	P_0V	20	X10

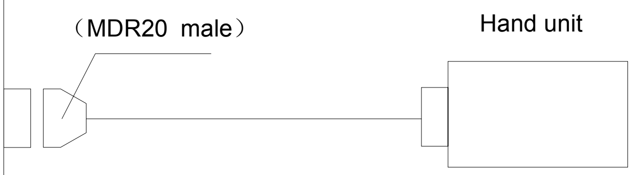
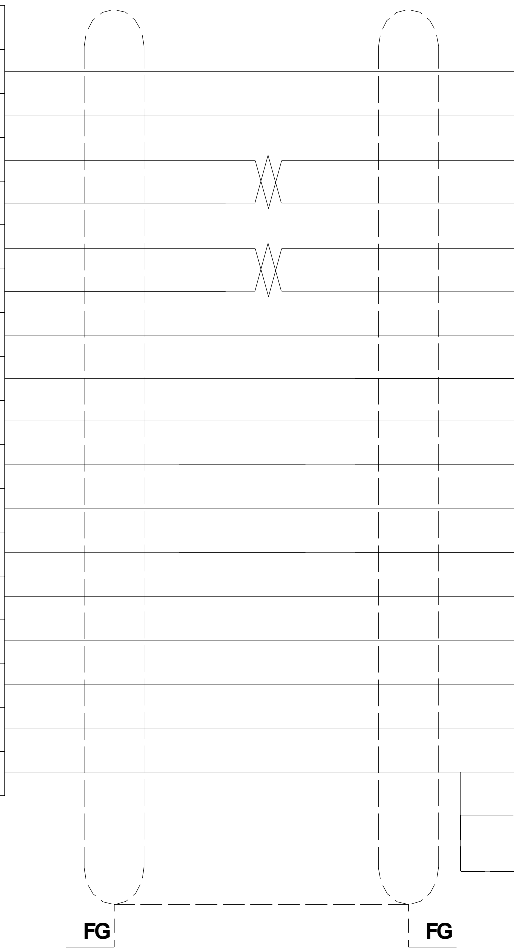


Fig. 6-1

External MPG signal connection is as Fig. 6-2.

MPG J15(MDR20)

	Signal	Pin
5V input	+5V	1
Emergency stop signal input	STP	3
A-phase differnce signal	PA+	16
A-phase differnce signal	PA-	17
B-phase differnce signal	PB+	15
B-phase differnce signal	PB-	14
24V output	P_24V	11
LED control signal	LED	4
X-axis selection signal	HX	5
Y-axis selection signal	HY	6
Z-axis selection signal	HZ	7
A-axis selection signal	H4	8
B-axis selection signal	H5	9
X1 override signal	X1	19
X10 override signal	X10	20
X100 override signal	X100	18
5V & 24V ground	P_0V	10



MPG

No.	Wiring color	Signal
1	Red	+5V
16	Blue-black	CN+
3	Green	A+
3*	Purple	A-
4	White	B+
4*	Purple-black	B-
5	Green-black	LED+
6	White-black	LED-
7	Yellow	X
8	Yellow-black	Y
9	Brown	Z
10	Brownish black	4
9*	Pink	5
11	Grey	X1
12	Greyish black	X10
13	Orange	X100
14	Orange-black	COM
15	Light blue	CN-
2	Black	0V

Fig. 6-2

MPG signal point definition.

Table 6-1

Signal name	PLC address	Signal function	I / O
HX	X120.7	X axis selection signal input	I
HY	X120.6	Y axis selection signal input	I
HZ	X120.5	Z axis selection signal input	I
H4	X120.4	4 axis selection signal input	I
H5	X120.3	5 axis selection signal input	I
X1	X120.2	X1 override signal input	I
X10	X120.1	X10 override signal input	I
X100	X120.0	X100 override signal input	I
STP	X121.0	Emergency stop signal input	I
LED	Y120.0	LED lamp output	O

7 Operation Panel Signal Line

GSK25I CNC system communicates with the operation panel by RS485 serial interface as Fig. 7-1.

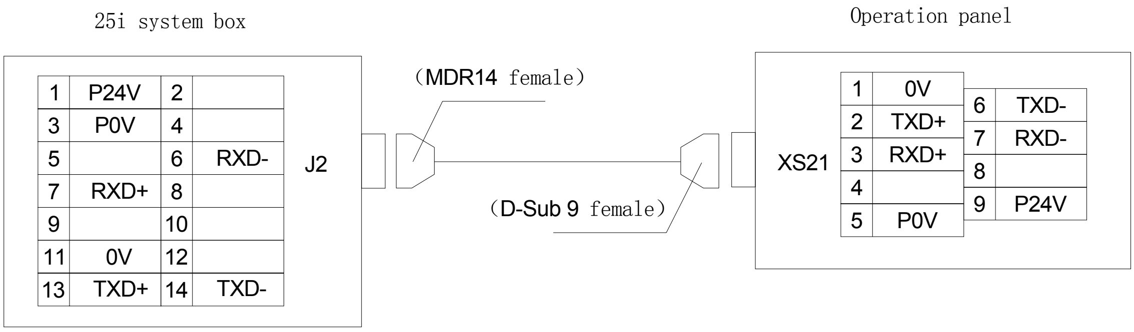


Fig. 7-1

Cable connection of operation panel is as Fig. 7-2.

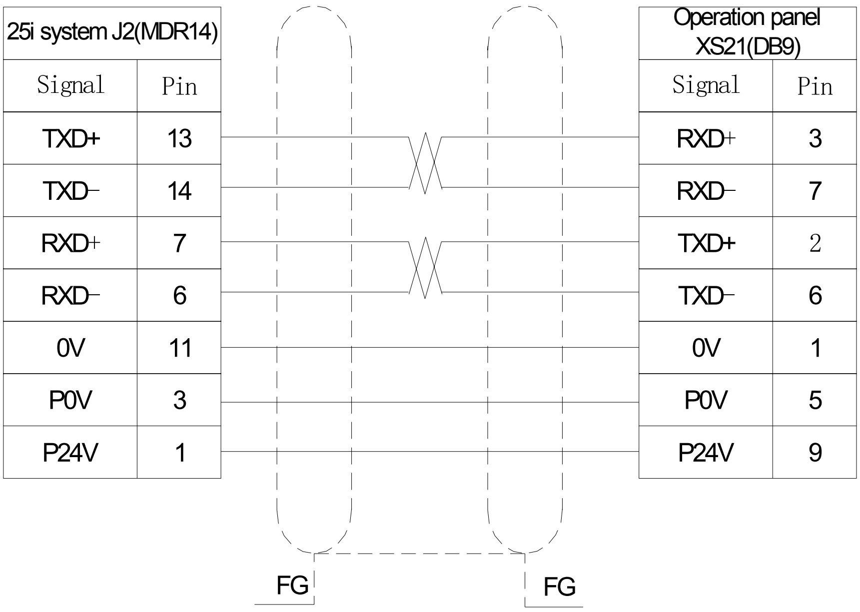


Fig. 7-2

8 Ethernet Communication Connection

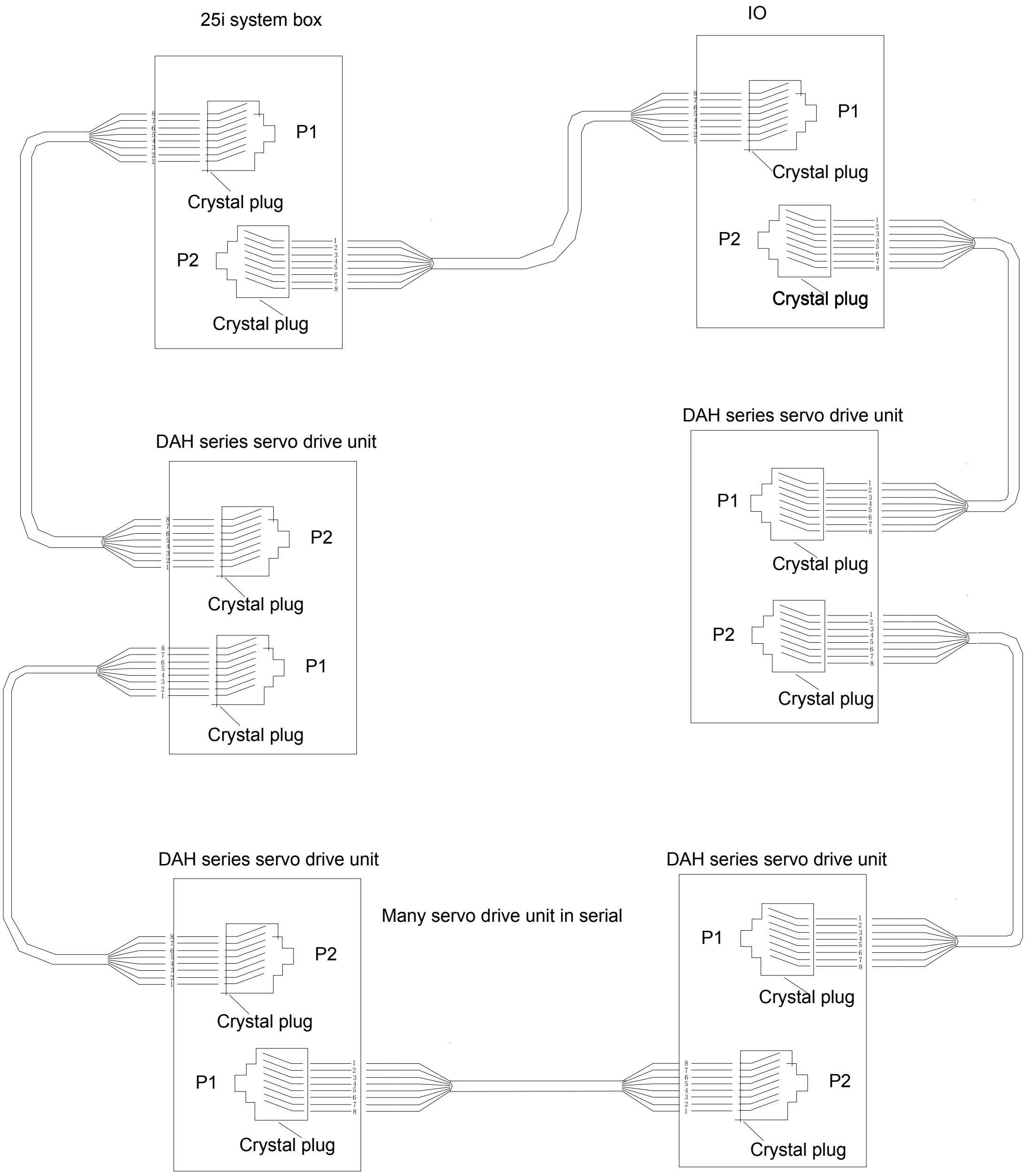
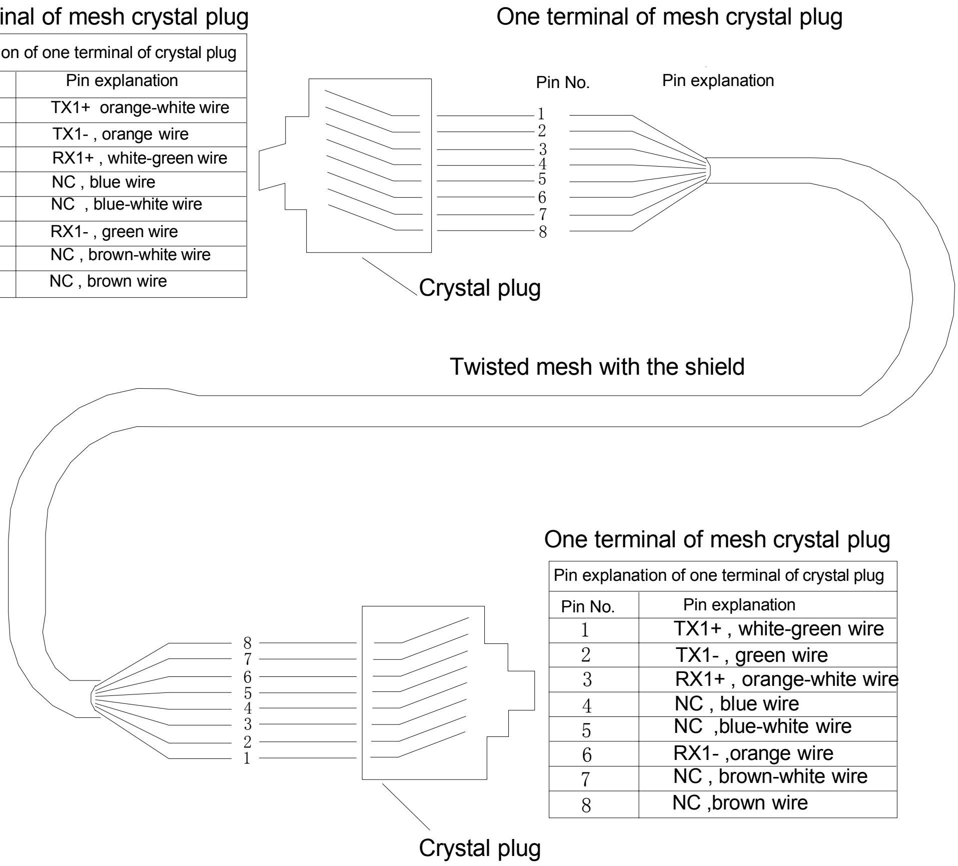


Fig. 8-1

Cable connection drawing of Ethernet

One terminal of mesh crystal plug

Pin explanation of one terminal of crystal plug	
Pin No.	Pin explanation
1	TX1+ orange-white wire
2	TX1- , orange wire
3	RX1+ , white-green wire
4	NC , blue wire
5	NC , blue-white wire
6	RX1- , green wire
7	NC , brown-white wire
8	NC , brown wire



One terminal of mesh crystal plug

Pin explanation of one terminal of crystal plug	
Pin No.	Pin explanation
1	TX1+ , white-green wire
2	TX1- , green wire
3	RX1+ , orange-white wire
4	NC , blue wire
5	NC ,blue-white wire
6	RX1- ,orange wire
7	NC , brown-white wire
8	NC ,brown wire

Fig. 8-2

9 Connected with the Spindle Servo

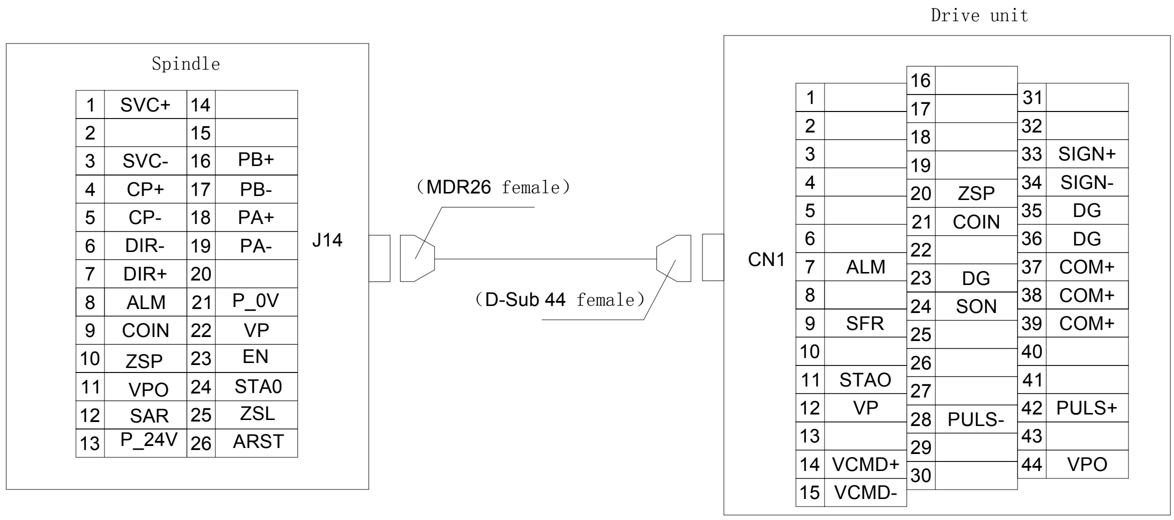


Fig. 9-1

Cable connection drawing of spindle.

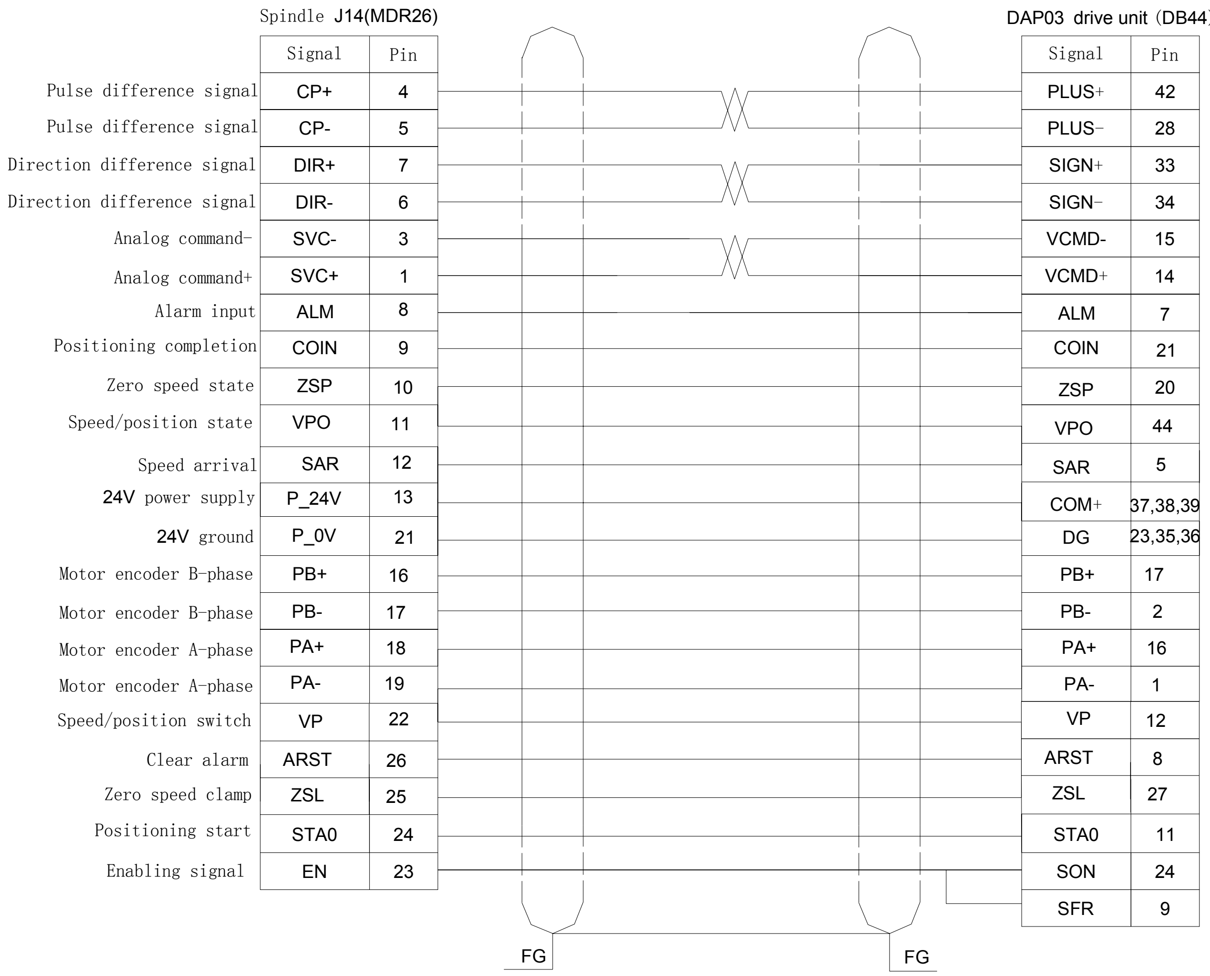


Fig. 9-2

10 Connected with the Spindle Converter

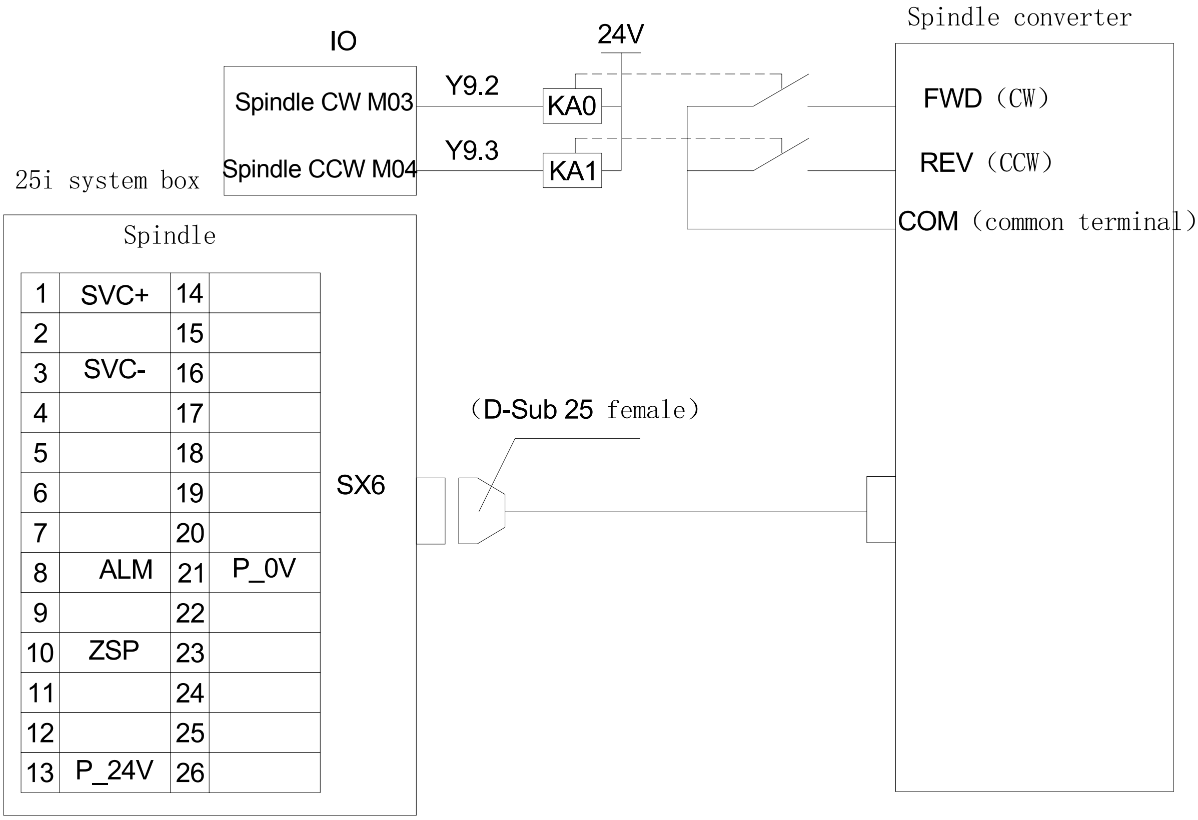


Fig. 10-1

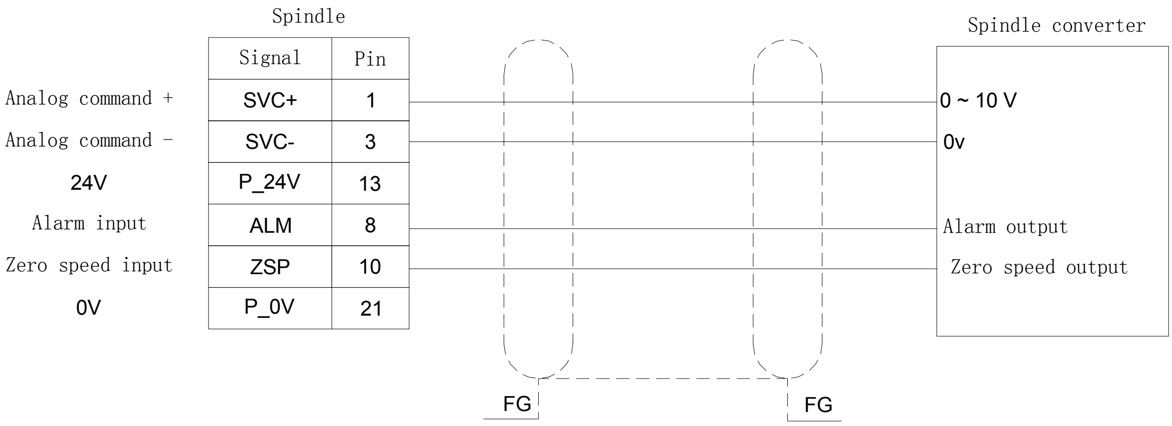


Fig. 10-2

11 Connection Method of Z Brake, System Power-on Control

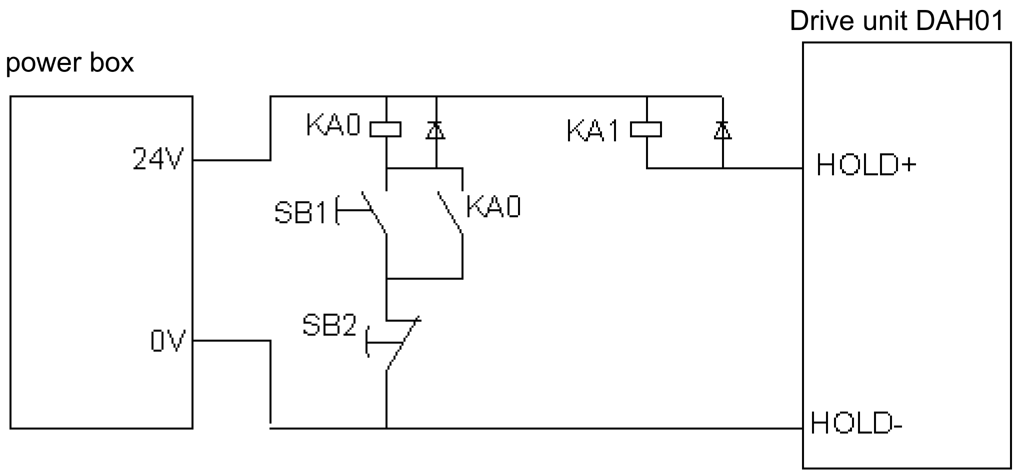


Fig. 11-1

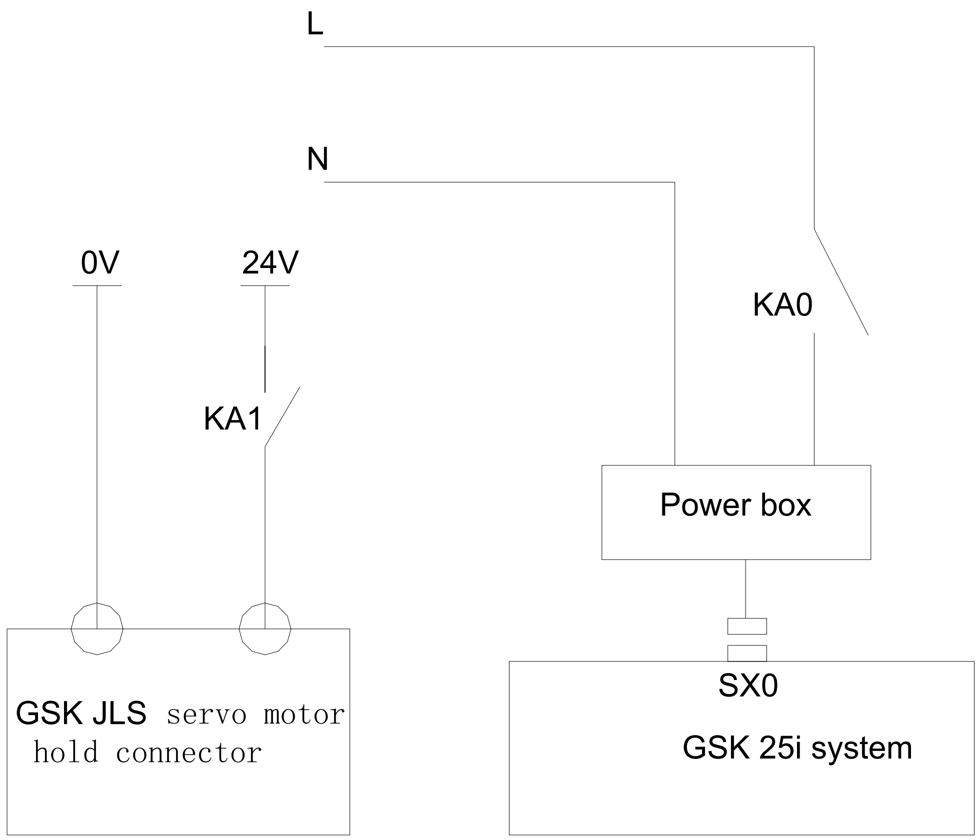


Fig. 11-2

12 I/O Input, Output Signal

12.1 Connection method of input signal

COM terminal of each group of address determines whether HIGH or LOW input is valid:

（1）When COM is connected with 24V, each input point connected with LOW（0V）is valid;

（2）When COM is connected with 0V, each input point connected with HIGH（24V）is valid.

connection method when LOW is valid **connection method when HIGH is valid**

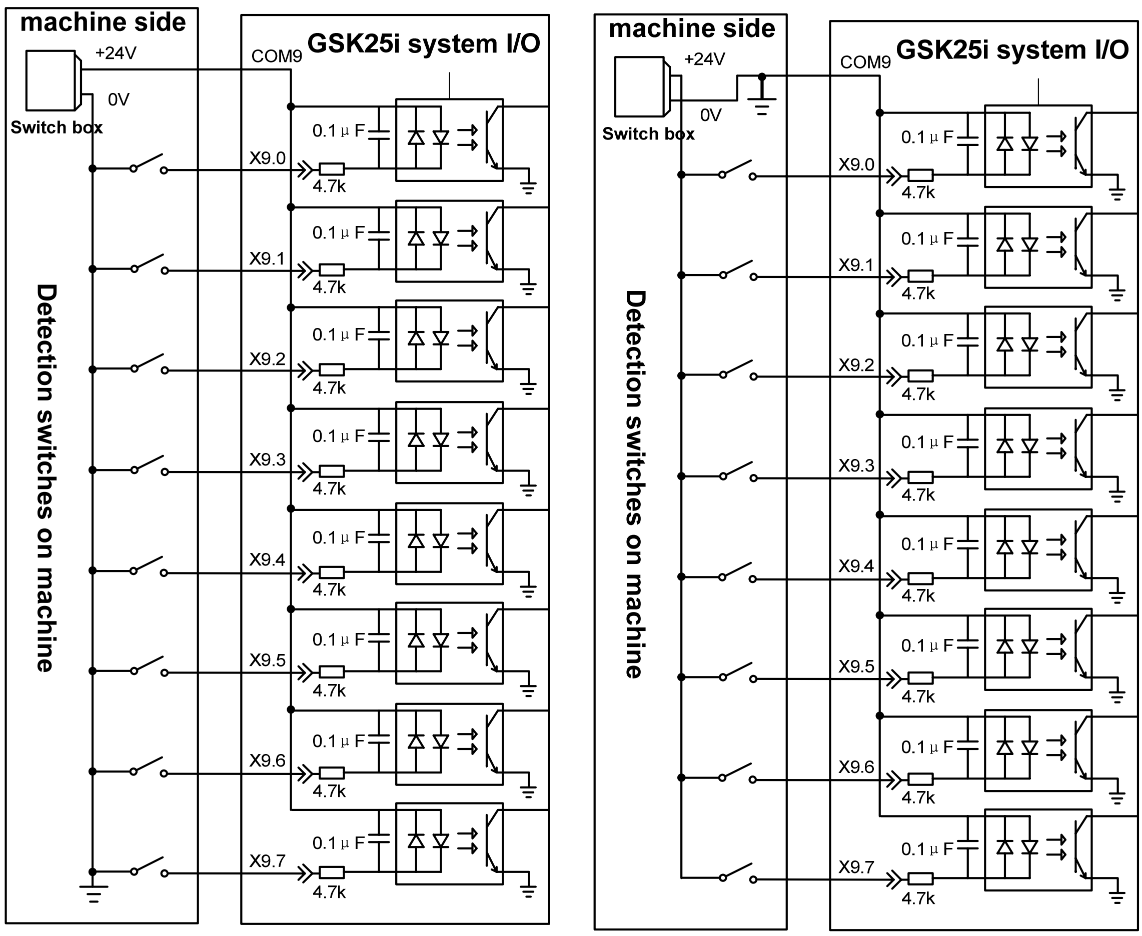


Fig. 12-1

***Note**：An input point has 8 groups including 64 points, the above figure takes the example of the group of X9.0—X9.7，and the connection methods of other groups are the same.

12.2 Connection method of output signal

An output signal has 48 points using the output ULN280-3，max. flowing current of each point is 200mA.

Output signal connection method

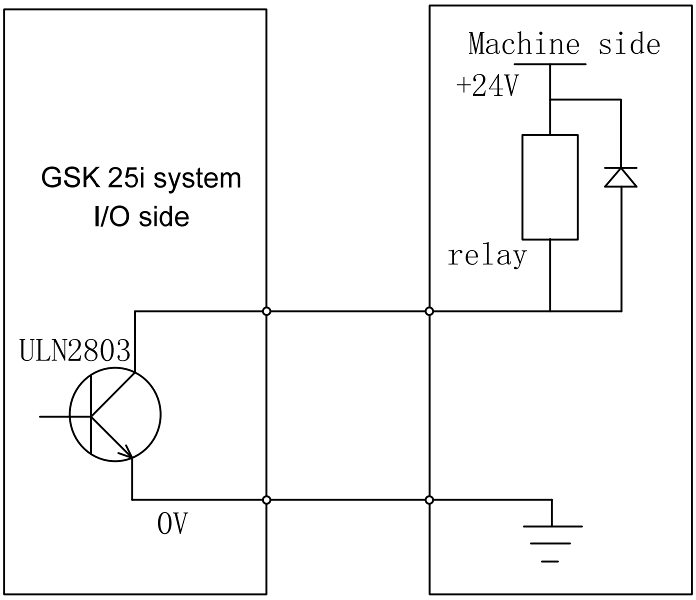


Fig. 12-2

12.3 Definition of input signal point

Table 12-1

Terminal No.	PLC address	Signal name	Signal function	I / O
X9.0	X9.0	*DECX（fixed）	X zero return deceleration input signal，normally-closed contact, power-off is valid	I
X9.1	X9.1	*DECY（fixed）	Y zero return deceleration input signal，normally- closed contact, power-off is valid	I
X9.2	X9.2	*DECZ（fixed）	Z zero return deceleration input signal，normally-closed contact, power-off is valid	I
X9.3	X9.3	*DEC4（fixed）	4th zero return deceleration input signal，normally- closed contact, power-off is valid	I
X9.4	X9.4	*DEC5（fixed）	5th zero return deceleration input signal，normally- closed contact, power-off is valid	I
X9.5	X9.5			
X9.6	X9.6	*+LX（fixed）	X positive limit(short circuit when not be used) normally- closed contact, power-off is valid	I
X9.7	X9.7	*-LX（fixed）	X negative limit(short circuit when not be used) normally- closed contact, power-off is valid	I
X10.0	X10.0	*+LY（fixed）	Y positive limit(short circuit when not be used) normally- closed contact, power-off is valid	I
X10.1	X10.1	*-LY（fixed）	Y negative limit(short circuit when not be used) normally- closed contact, power-off is valid	I
X10.2	X10.2	*+LZ（fixed）	Z positive limit(short circuit when not be used) normally- closed contact, power-off is valid	I
X10.3	X10.3	*-LZ（fixed）	Z negative limit(short circuit when not be used) normally- closed contact, power-off is valid	I
X10.4	X10.4	*+L4（fixed）	4th positive limit(short circuit when not be used) normally- closed contact, power-off is valid	I
X10.5	X10.5	*-L4（fixed）	4th negative limit(short circuit when not be used) normally- closed contact, power-off is valid	I
X10.6	X10.6	*+L5（fixed）	5th positive limit(short circuit when not be used) normally- closed contact, power-off is valid	I
X10.7	X10.7	*-L5（fixed）	5th negative limit(short circuit when not be used) normally- closed contact, power-off is valid	I
X11.0	X11.0	LUB.ALM	Lubricating pump alarm input	I
X11.1	X11.1	DOOR	Safe door input	I
X11.2	X11.2	HYPUP.ALM	Hydraulic pump overload input signal	I
X11.3	X11.3	AIRPRE.ALM	Air pressure check alarm input signal	I
X11.4	X11.4	CLNM.ALM	Cooling pump motor overload alarm input signal	I
X11.5	X11.5	CHIPM.ALM	Chip removal motor overload input signal	I
X11.6	X11.6	MGPLA.ALM	Tool pot motor overload input signal	I
X11.7	X11.7	USER.ALM1	Custom alarm 1 input terminal	I

Terminal No.	PLC address	Signal name	Signal function	I/O
X12.0	X12.0	GR1.M	Spindle No. 1 gear(in-position check)	I
X12.1	X12.1	GR2.M	Spindle No. 2 gear(in-position check)	I
X12.2	X12.2	GR3.M	Spindle No. 3 gear(in-position check)	I
X12.3	X12.3			I
X12.4	X12.4			I
X12.5	X12.5	TRLCK.I	Release tool (in-position check)	I
X12.6	X12.6	TCLCK.I	Clamp tool(in-position check)	I
X12.7	X12.7	CKST	Release/clamp tool button	I
X13.0	X13.0	4UCLPI	4th axis release in-position check	I
X13.1	X13.1	4CLPI	4th axis clamp in-position check	I
X13.2	X13.2	4CLPI.JOG	4th axis clamp button button input	I
X13.3	X13.3	4UCLPI.JOG	4th axis release button input	I
X13.4	X13.4	5UCLPI	5th axis release in-position check	I
X13.5	X13.5	5CLPI	5th axis clamp in-position check	I
X13.6	X13.6	5CLPI.JOG	5th axis clamp button button input	I
X13.7	X13.7	5UCLPI.JOG	5th axis release button input	I
X14.0	X14.0	T-BARE	Current position empty tool check of tool magazine	I
X14.1	X14.1	TZER.I	Tool magazine zero return signal	
X14.2	X14.2	TCN.I	Tool count signal	I
X14.3	X14.3	TFN.I	Tool magazine forward in-position	I
X14.4	X14.4	TBK.I	Tool magazine backward in-position	I
X14.5	X14.5			
X14.6	X14.6			
X14.7	X14.7			

Note: **X15.0—X15.7, X16.0—X16.7 together have 16 input signal interfaces to the user.**

12.4 Definition of output signal point

Terminal No.	PLC address	Signal name	Signal function	I/O
Y8.0	Y8.0	CLN.O	Cooling （coolant）pump output	O
Y8.1	Y8.1	MGFR.O	Tool magazine forward（Output signal）	O
Y8.2	Y8.2	MGBK.O	Tool magazine backward（Output signal）	O
Y8.3	Y8.3			O
Y8.4	Y8.4	TRL.M	Release tool（Output signal）	O
Y8.5	Y8.5	MGCW.O	Tool magazine CW（Output signal）	O
Y8.6	Y8.6	MGCCW.O	Tool magazine CCW（Output signal）	O
Y8.7	Y8.7	HYPR.O	Hydraulic oil pump output	O
Y9.0	Y9.0	LUB.O	Lubricating pump output	O
Y9.1	Y9.1	OR.T	Overtravel release	O
Y9.2	Y9.2	M03	Spindle CW（Output signal）	O
Y9.3	Y9.3	M04	Spindle CCW（Output signal）	O
Y9.4	Y9.4	RED.ALL	Red lamp alarm output	O
Y9.5	Y9.5	YEL.ALL	Yellow lamp output（normally wait）	O
Y9.6	Y9.6	GRE.ALL	Green lamp output（machine normally runs）	O
Y9.7	Y9.7			O
Y10.0	Y10.0	GR1.O	Spindle No.1 gear output	O
Y10.1	Y10.1	GR2.O	Spindle No.2 gear output	O
Y10.2	Y10.2	GR3.O	Spindle No.3 gear output	O
Y10.3	Y10.3			O
Y10.4	Y10.4			
Y10.5	Y10.5			
Y10.6	Y10.6			
Y10.7	Y10.7			
Y11.0	Y11.0	LAMP.L	Machine working lamp	O
Y11.1	Y11.1	CLN2.O	chip water valve output	O
Y11.2	Y11.2	CFN.O	Spindle blowing output	O
Y11.3	Y11.3	CLN-2.O	Workpiece blowing output	O
Y11.4	Y11.4	CHIP1.CW	Chip removal 1 CW output	O
Y11.5	Y11.5	CHIP1.CCW	Chip removal 1 CCW output	O
Y11.6	Y11.6	CHIP2.CW	Chip removal 2 output	O
Y11.7	Y11.7			O

Y12.0	Y12.0	4UCLPO	4th axis release output	O
Y12.1	Y12.1	4-CLPO	4th axis clamp output	O
Y12.2	Y12.2	5UCLPO	5th release output	O
Y12.3	Y12.3	5-CLPO	5th clamp output	O
Y12.4	Y12.4			
Y12.5	Y12.5			
Y12.6	Y12.6			
Y12.7	Y12.7			
Y13.0	Y13.0			
Y13.1	Y13.1			
Y13.2	Y13.2			
Y13.3	Y13.3			
Y13.4	Y13.4			
Y13.5	Y13.5			
Y13.6	Y13.6			
Y13.7	Y13.7			

Appendix

1 Signal table(address order)

F code		
Address	Signal name	Symbol
F000#4	Automatic run pause signal	SPL
F000#5	Automatic run start signal	STL
F000#6	Servo ready signal	SA
F000#7	Automatic run signal	OP
F001#0	Alarm signal	AL
F001#1	reset signal	RST
F001#3	Distribution end signal	DEN
F001#4	Spindle enabling signal	ENB
F001#7	Read end signal	MA
F002#1	Rapid feed signal	RPDO
F002#4	Program restart signal	SRNMV
F002#6	Cutting feed signal	CUT
F002#7	Dry run check signal	MDRN
F003#0	Incremental feed selection signal	MINC
F003#1	MPG feed selection signal	MH
F003#2	Manual continuous feed selection signal	MJ
F003#3	Select manual data input signal	MMDI
F003#4	Select DNC run signal	MRMT
F003#5	Select automatic run signal	MMEM
F003#6	Memory edit selection signal	MEDT
F003#7	Machine zero return detection signal	MZRO
F004#0	Jump optional block detection signal	MBDT
F004#1	All-axes machine lock signal	MMLK
F004#3	Single block signal	MSBK
F004#4	Auxiliary function lock signal	MAFL
F004#5	Manual reference point return signal	MREF
F007#0	Auxiliary function strobe signal	MF
F007#2	Spindle speed strobe signal	SF
F007#3	Tool function strobe signal	TF
F007#5	No.2 M function strobe signal	MF2
F007#6	No. 3M function strobe signal	MF3
F009#4	M decoding signal	DM30
F009#5		DM02
F009#6		DM01
F009#7		DM00

F010~F013	Auxiliary function signal	M00-M31
F014~F017	No. 2M function signal	M100~M131
F018~F021	No. 3M function signal	M200~M231
F022~F025	Spindle function signal	S00~S31
F026~F029	Tool function signal	T00~T31
F034#0~#2	Gear selection signal(output)	GR1O,GR2O,GR3O
F045#0	Spindle alarm signal	SPALM
F045#1	Spindle zero-speed signal	SST
F045#3	Speed arrival signal	SAR
F045#7	Orientation completion signal	ORAR
F054~F057	Output signal used to user macro program	UO000~UO031
F060#0	External data read completion	EREND
F060#1	External data search completion	ERSND
F060#2	External data read cancel	ESCAN
F061#0	B-axis release signal	BUCLP
F061#1	B axis clamp signal	BCLP
F062#7	Signal for reaching the required number of workpiece	PRTSF
F065#0	Spindle rotation direction signal	RGSPP
F70#0~F71#7	Position switch signal	PSW01-PSW16
F076#3	Speed/position switch completion	VPO
F094	Reference point return end signal	ZP1~ZP5
F096	2nd reference point return end signal	ZP21~ZP24
F098	3rd reference point return end signal	ZP31~ZP34
F100	4th 2nd reference point return end signal	ZP41~AP44
F102	Axis moving signal	MV1~MV5
F106	Axis movement direction signal	MVD1~MVD5
F120	Reference point creation signal	ZRF1~ZRF5
F124	Travel limit arrival signal	+OT0~+OT4
F126	Travel limit arrival signal	-OT0~-OT4
F226~F229	Output signal used to macro program	UO100~UO131
F230~F233		UO200~UO231
F234~F237		UO300~UO331
G codes	Address	Signal name
G000~G003	External data input data signal	ED0~ED31
G004#3	Completion signal	FIN
G004#4	No. 2M function end signal	MFIN2
G004#5	No. 3M f unction end signal	MFIN3
G005#0	Auxiliary function end signal	MFIN
G005#6	Auxiliary function lock signal	AFL
G006#0	Program restart signal	SRN
G006#4	Override cancel signal	OVC
G006#6	Skip signal	JUMPP

G007#1	Start lock signal	STLK
G007#2	Automatic run start signal	ST
G008#0	All-axes interlock signal	***IT**
G008#4	Emergency stop signal	***ESP**
G008#5	Feed pause signal	***SP**
G008#2	Optional stop signal(add)	SOP
G008#6	Reset & tap rewinding signal	RRW
G008#7	External reset signal	ERS
G010~G011	Manual feedrate override signal	JV0~JV15
G012	Feedrate override signal	FV0~FV7
G013#0 ~G013#6	External data input address signal	EA0~EA6
G013#7	External data read signal	ESTB
G014#0,#1	Rapid feedrate override signal	ROV1,ROV2
G018#0~#3	MPG feed axis selection signal	HS1A~HS1D
G019#4,#5	MPG feed movement selection signal(incremental feed signal)	MP1,MP2
G019#7	Manual rapid feed selection signal	RT
G028#1-#2	Gear selection signal(input)	GR1,GR2,
G029#4	Spindle speed arrival signal	SAR
G029#5	Spindle orientation signal	SOR
G29#6	Spindle stop signal	***SSTP**
G030	Spindle speed override signal	SOV0~SOV7
G033#5	Spindle motor command polar selection signal	SGN
G033#6	Spindle motor command polar selection signal	SSIN
G033#7	Spindle motor command selection signal	SIND
G043#0~#2	Mode selection signal	MD1,MD2,MD4
G043#4	Step run selection signal	INC
G043#5	DNC run selection signal	DNCI
G043#7	Manual reference point return selection signal	ZRN
G044#0	Jump optional block signal	BDT
G044#1	All-axes machine lock signal	MLK
G046#1	Single block signal	SBK
G046#7	Dry run signal	DRN
G054~G057	Macro call input signal	UI000~UI031
G061#0	Rigid tapping signal	RGTAP
G070#4	Spindle CCW rotation	SRVA
G070#5	Spindle CW rotation	SFRA
G070#6	Spindle orientation output signal	ORCM
G096#0~#6	1% rapid feedrate override signal	HROV0~HROV6
G096#7	1% rapid feedrate override select signal	HROV
G100#0~#4	Feed axis and direction signal	+J1~+J5
G102#0~#4	Feed axis and direction signal	-J1~-J5
G108#0~#4	Each axis machine lock signal	MLK1~MLK5
G114#0~#4	Overtravel signal	*+L1~*+L5

G116#0~#4	Overtravel signal	*-L1~*-L5
G118#0~#4	External deceleration signal	*+ED1~*+ED5
G120#0~#4	External deceleration signal	*-ED1~*-ED5
G132#0~ G132#4	Positive interlock of each axis	+MIT1~+MIT5
G134#0~ G134#4	Negative interlock of each axis	-MIT1~-MIT5
G226~G229	Input signal used to macro program	UI100~UI131
G230~G233		UI200~UI231
G234~G237		UI300~UI331

2 Contour installation dimension drawing

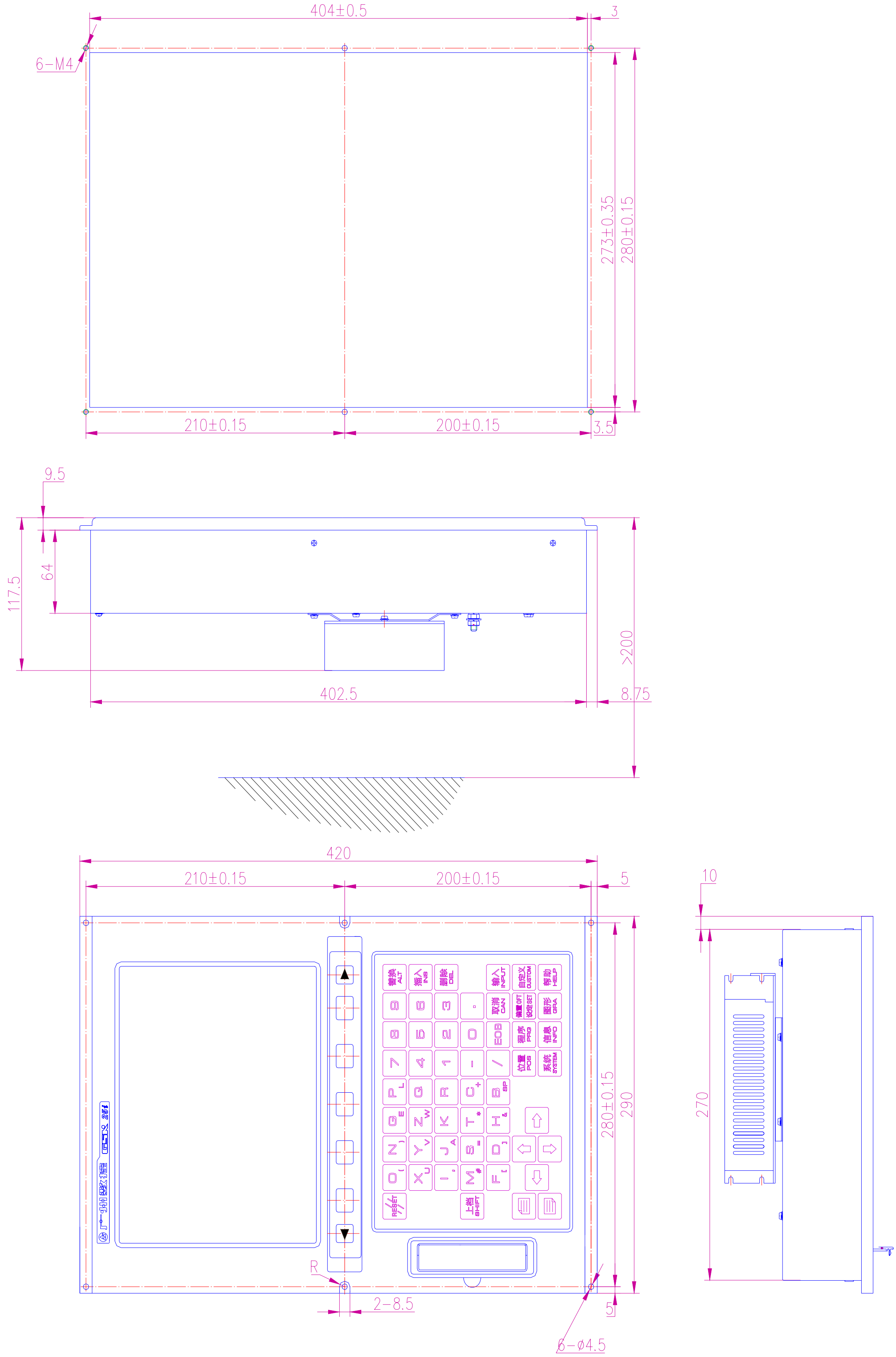


Fig. B-1 **GSK 25i system box installation dimension (unit：mm)**

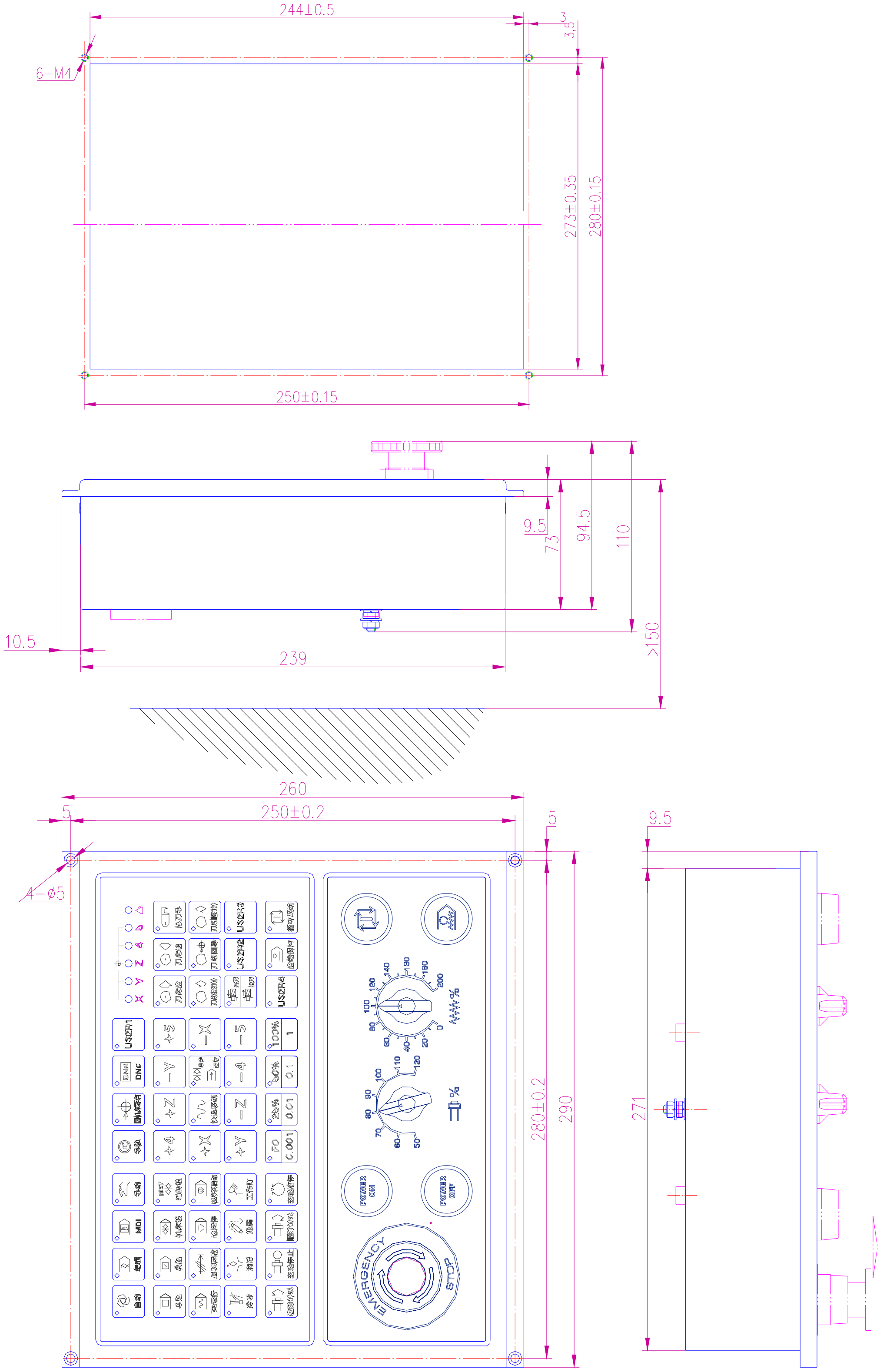


Fig. B-2 **Operation panel installation dimension(unit：mm)**

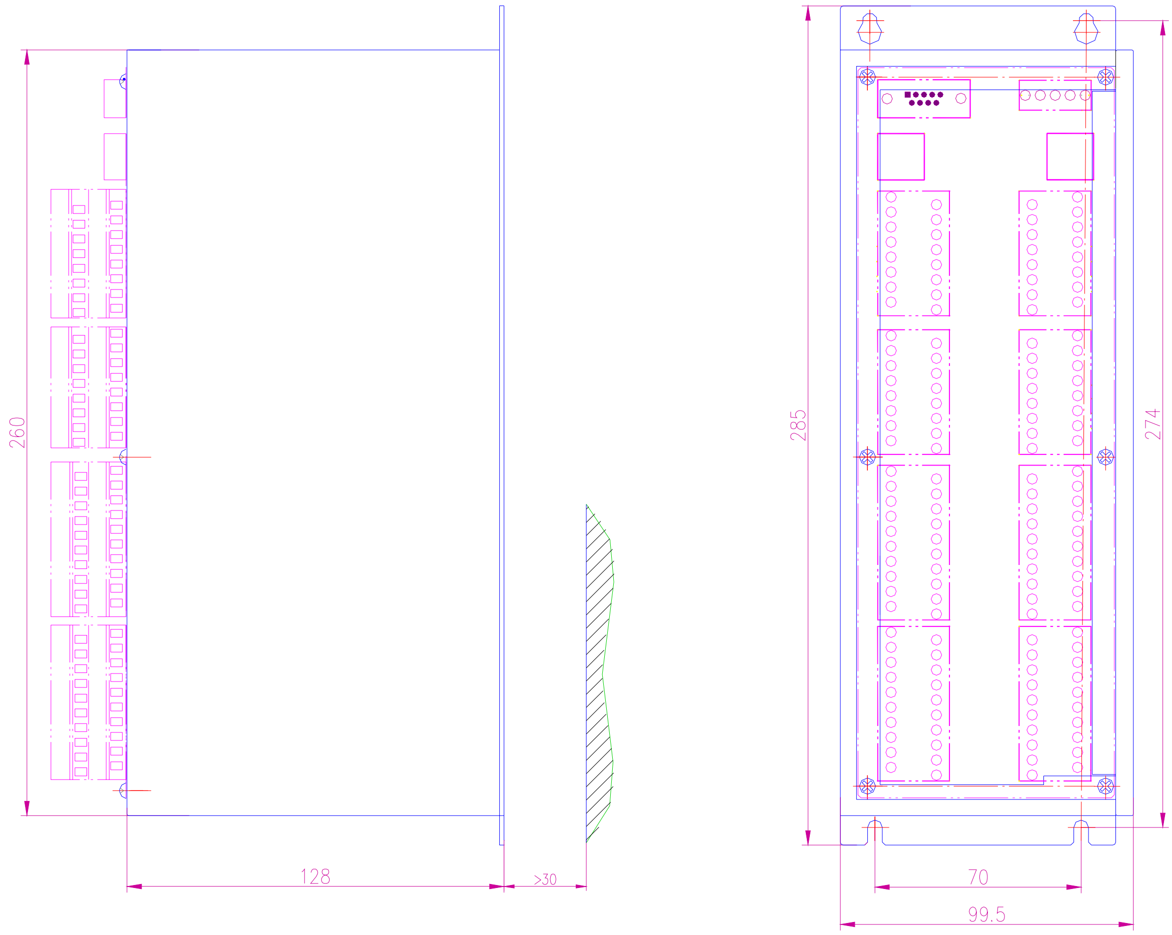


Fig. B-3 **I/O Unit installation dimension(unit：mm)**

Remark: These dimensions are subject to change without further notice. Please refer to the actual product and installation manual for details.

Warning and Precaution

Accident may occur by improper connection and operation！This system can only be operated by authorized and qualified personnel.

Please read this manual carefully before operation！

Please read this manual and a manual from machine tool builder carefully before installation, programming and operation, and strictly observe the requirements.

This manual includes the precautions for protecting user and machine tool. The precautions are classified into Warning and Caution according to their bearing on safety, and supplementary information is described as Note. Read these Warnings, Cautions and Notes carefully before operation.

Warning

User may be injured or equipment be damaged if operation instructions and procedures are not observed.

Caution

Equipment may be damaged if operation instructions or procedures are not observed.

Note

It is used to indicate the supplementary information other than Warning and Caution.

Announcement

- This manual describes various possibilities as much as possible. However, operations allowable or unallowable cannot be explained one by one due to so many possibilities that may involve with, so the contents that are not specially stated in this manual shall be considered as unallowable.

Caution

- Functions, technical indexes (such as precision and speed) described in this user manual are only for this system. Actual function deployment and technical performance of a machine tool with this CNC system are determined by machine tool builder's design, so functions and technical indexes are subject to the user manual from machine tool builder.
- Refer to the user manual from machine tool builder for function and meaning of keys on control panel.

Precautions

■ Delivery and storage

- Packing box over 6 layers in pile is unallowed.
- Never climb the packing box, neither stand on it, nor place heavy objects on it.
- Do not move or drag the products by the cables connected to it.
- Forbid collision or scratch to the panel and display screen.
- Avoid dampness, insolation and drenching.

■ Open-package inspection

- Confirm that the products are the required ones.
- Check that the products are not damaged in delivery.
- Confirm that the parts in packing box are in accordance with the packing list.
- Contact us in time if any inconsistence, shortage or damage is found.

■ Connection

- Only qualified personnel can connect the system or check the connection.
- The system must be earthed, and the earth resistance must be less than 0.1Ω. The earth wire cannot be replaced by zero wire.
- The connection must be correct and firm to avoid any fault or unexpected consequence.
- Connect with surge diode in the specified direction to avoid damage to the system.
- Switch off power supply before plugging out or opening electric cabinet.

■ Troubleshooting

- Only competent personnel are supposed to inspect the system or machine.
- Switch off power supply before troubleshooting or changing components.
- Check for fault when short circuit or overload occurs. Restart can only be done after troubleshooting.
- Frequent switching on/off of the power is forbidden, and the interval time should be at least 1 min.

Safety Responsibility

Manufacturer's Responsibility

——Be responsible for the danger which should be eliminated and/or controlled on design and configuration of the provided CNC systems and accessories.

——Be responsible for the safety of the provided CNC systems and accessories.

——Be responsible for the provided information and advice for the users.

User's Responsibility

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

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

——Be responsible for the failure to observe the provisions for operation, adjustment, maintenance, installation and storage in the manual.

All specifications and designs herein are subject to change without further notice.

Contents

1 Parameter Display

The operations are shown below:

(1) Enter the parameter screen after the function key  on MDI panel is controlled for many times, or press the **[Parameter]** and **[Operation]** soft keys subsequently after pressing the function key  once.

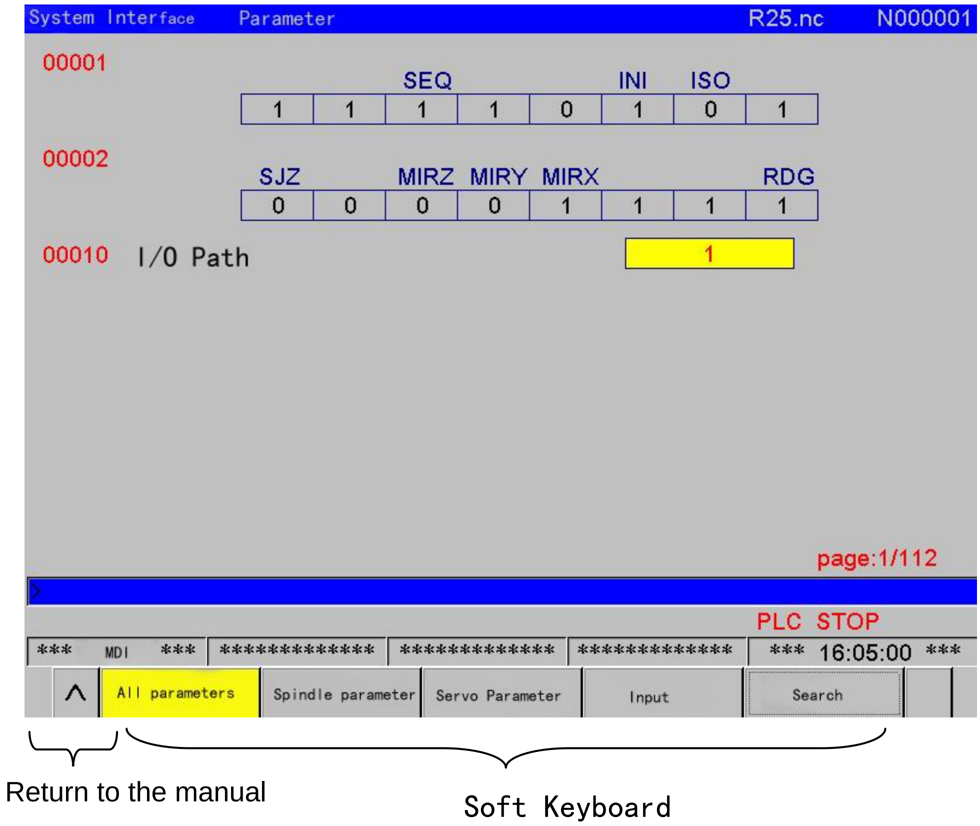


Fig.1-1

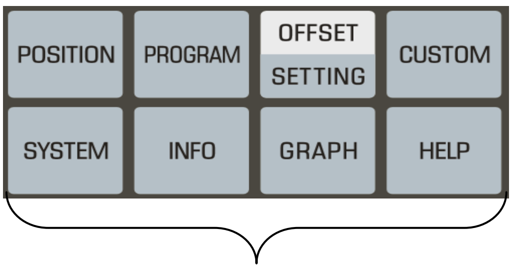


Fig.1-2 Function keys

(2) The parameter screen consists of multiple pages. Use two steps to display the page that contains the parameter you want to display.

(a) The required relative parameters are selected using the soft key, and then the page to be found by the page keys or cursor move keys.

(b) The parameter numbers to be displayed are input from keyboard, and press the **[search]** softkey to search, then the specified parameter page is displayed, and the cursor is positioned to the specified parameter (the data part is turned into the selected color).

2 Parameter Setting in the Mode of MDI

The operation steps of parameters setting are shown below:

(1) Enter the offset setting page by pressing the 

, and firstly to input the correspondence password.

To prevent the machining program and CNC parameters from being maliciously modified, the GSK 25i offers an authority setting function and the password can be divided into 9 levels, from the higher to the lower level, such as the 0 level (the system high level), the 1st level (the system service), the 2nd level (the machine manufacturer), the 3rd level (the installation and debugging), the 4th level (the terminal administration), as well as the 5th level (the operator 1 level), the 6th level (the operator 2 level), the 7th level (the operator 3 level) and the lowest default level (see the figure 2-1). The 0 level is enjoys the highest protection; contrarily, the lowest levels are from 5 to 7, and the highest level can be administrated the lowest levels, which is the low authority function. The parameter password level is 3 except for the special explaination.

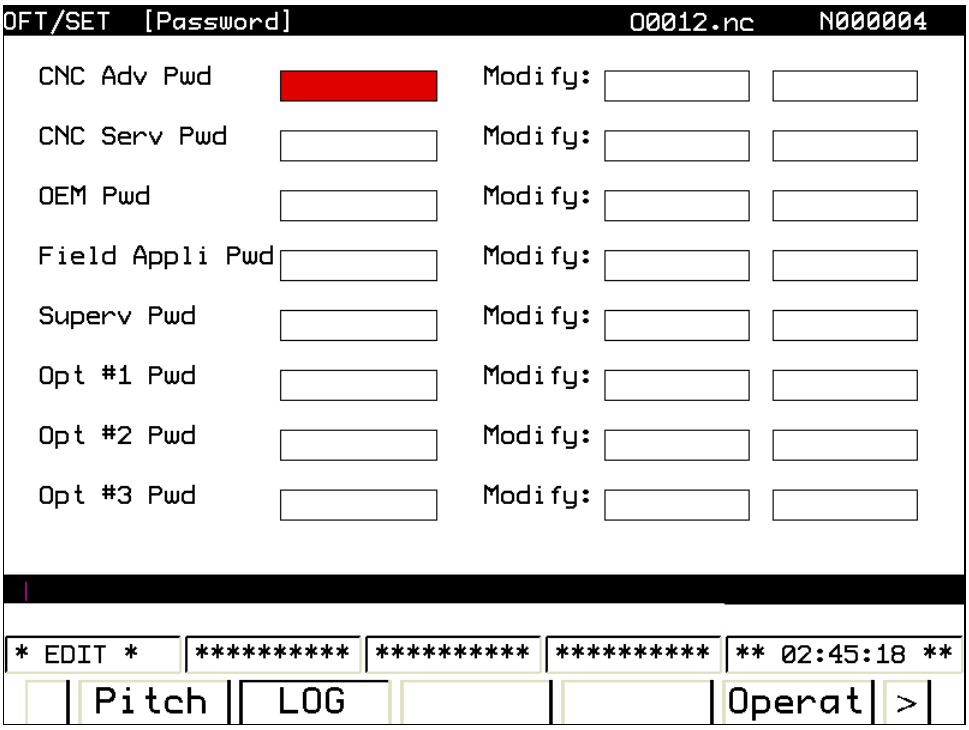


Fig. 2-1

Level 0: the highest authority, reserved by the developer.

Level 1: It is used for the system manufacturer service, which can modified various data.

Level 2: The PLC program, PLC note and the pitch error compensation are modified. The PLC and the pitch error compensation files are input or output. The user customized interface authority is modified/ input or output.

Level 3: The parameter and PLC source data can be modified; the PLC operation is started/stopped; the alarm/operation messages are eliminated; and the files are input or output, and the system, interpolation and positional control maintenance softwares can be upgraded.

Level 4: The program, tool offset, setting, workpiece coordinate system offset and macro program value are modified; these files are input or output and it also has the authority to modify the passward.

The 5^{th}, 6^{th} and 7^{th} levels: it is an operation authorized to corresponding person with bit-parameter by the end user administrator.

The lowest level default by the system: it is an authority operation donated with bit-parameter by end user administrator; no password inputs.

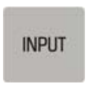
The bit-parameter definitions are authorized by the end user administrator, refer to the following table:

Bit	Significance	Note
0	Modify/input or output the authority of G code program.	Authority
1	Modify the authority of geometrical tool offset/input or output tool offset.	Authority
2	Modify the authority of wear tool offset/input or output tool offset.	Authority
3	Modify the authority of setting	Authority
4	Modify/input or output the authority of a workpiece coordinate system offset.	Authority
5	Modify/input or output the authority of a macro program value	Authority
6	Reserved	
7	Reserved	

(2) In the [MDI/Edit] mode, the MDI mode 

and Edit mode 

can be selected, and the cursor can be moved based on the password authority to the required items.

(3) Press the 

key, the corresponding level password can be input. If the password is correct, a "correct password" may be displayed in the system; otherwise a "wrong password input" may occur.

(4) After the corresponding parameters are modified, the password is cancelled after logging out.

3 Setting or Maintaining the System Parameters by PC Instruction Control Unit Software

3.1 Editing of System Parameters

This software can be edited the system parameter on CNC in the program, and the corresponding backup parameter files can be uploaded and downloaded through the internet. (Refer to the Fig. 3-1, Fig. 3-2 and Fig. 3-3)

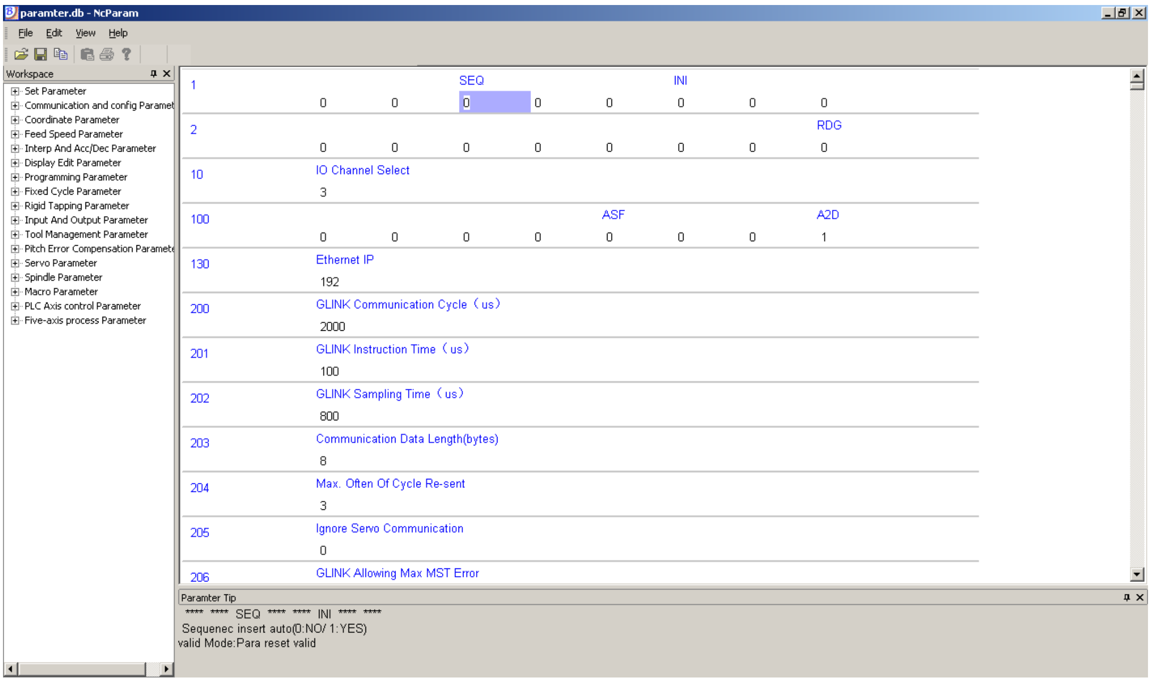


Fig.3-1 Editing the system parameters Ⅰ (Editing of the bit parameters)

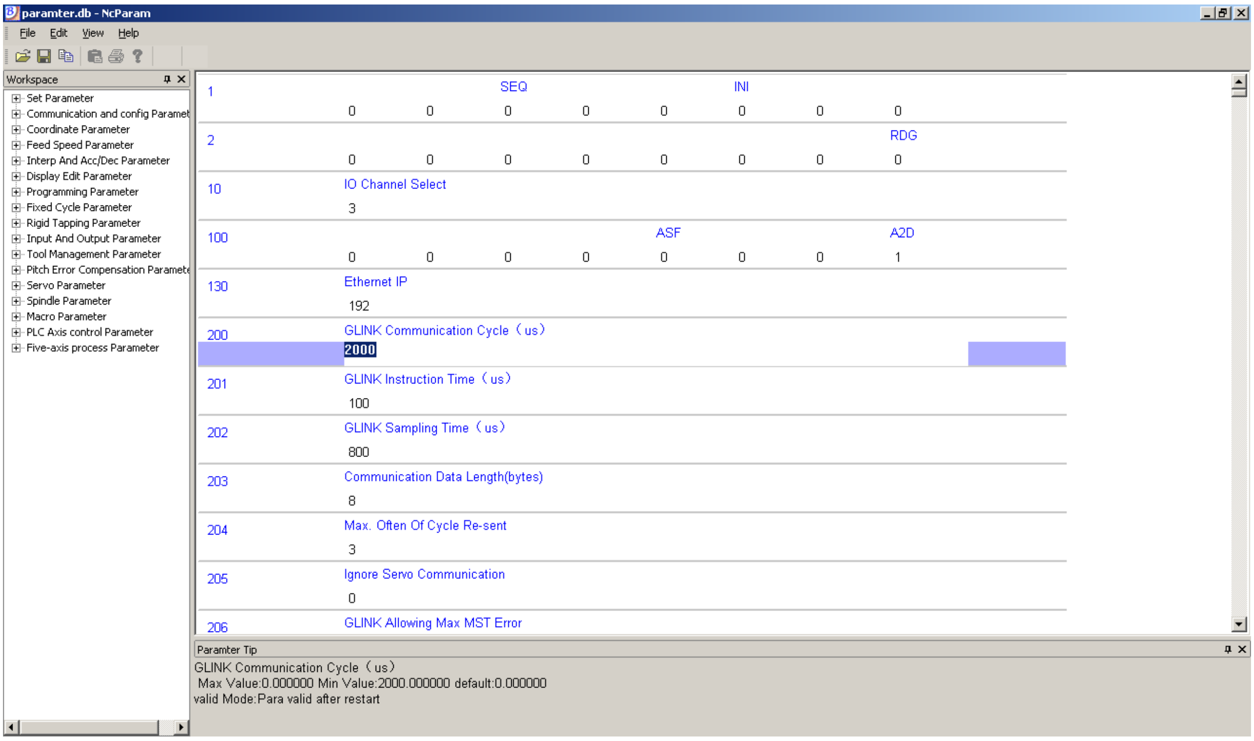


Fig.3-2 Editing the system parameters II (Editing of data parameters)

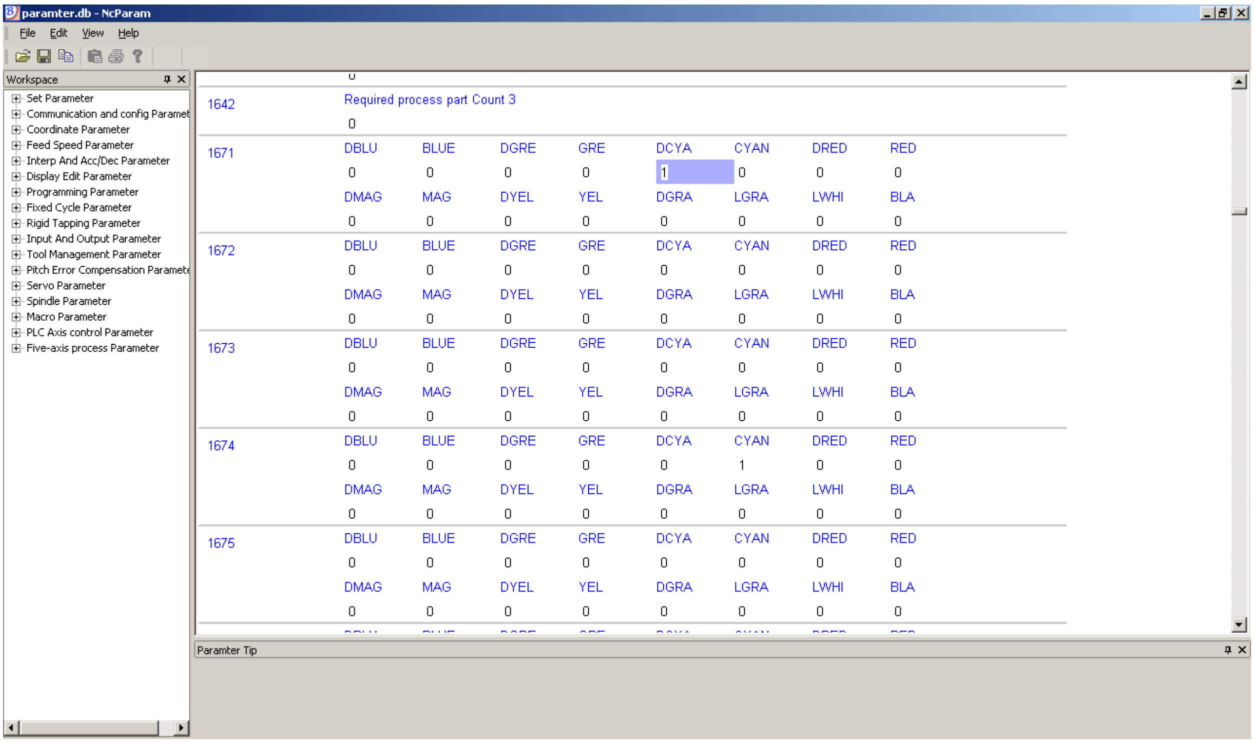
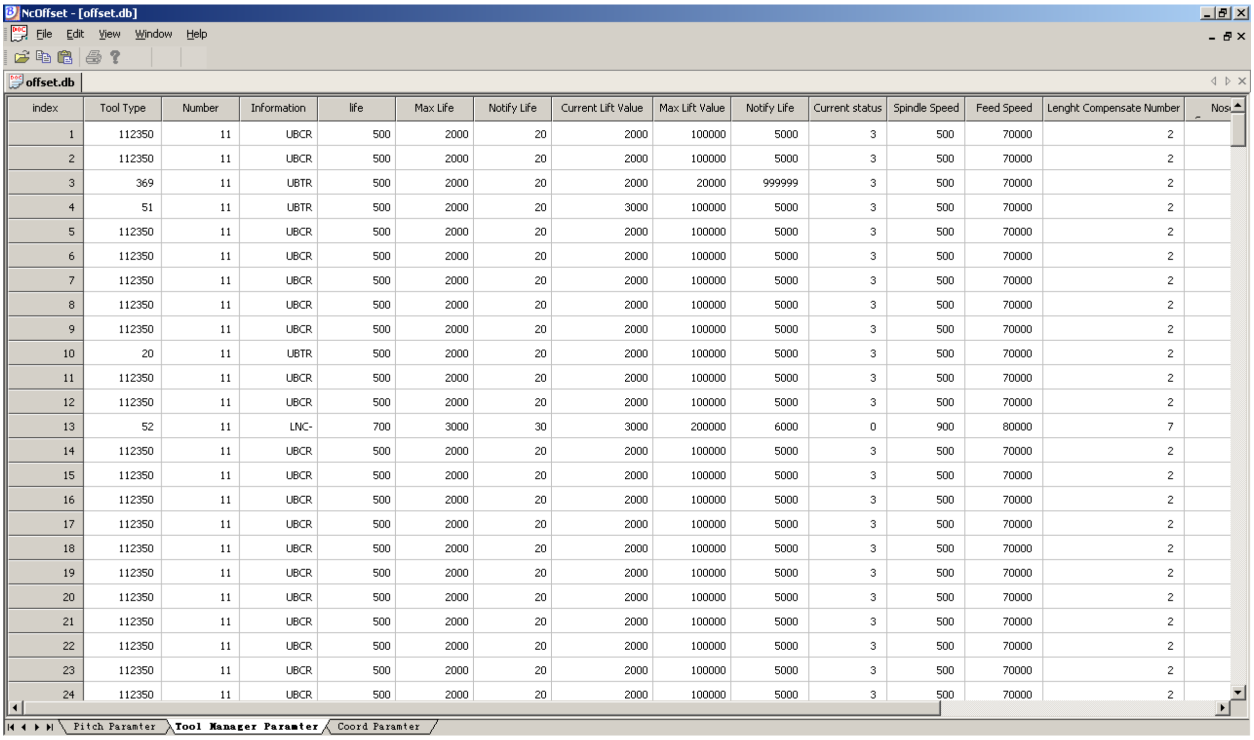


Fig.3-3 Editing the system parameters III (Editing of color parameters)

3.2 Editing of Tool and Offset Parameter

Editing of the tool and offset parameter is as the Fig. 3-4.

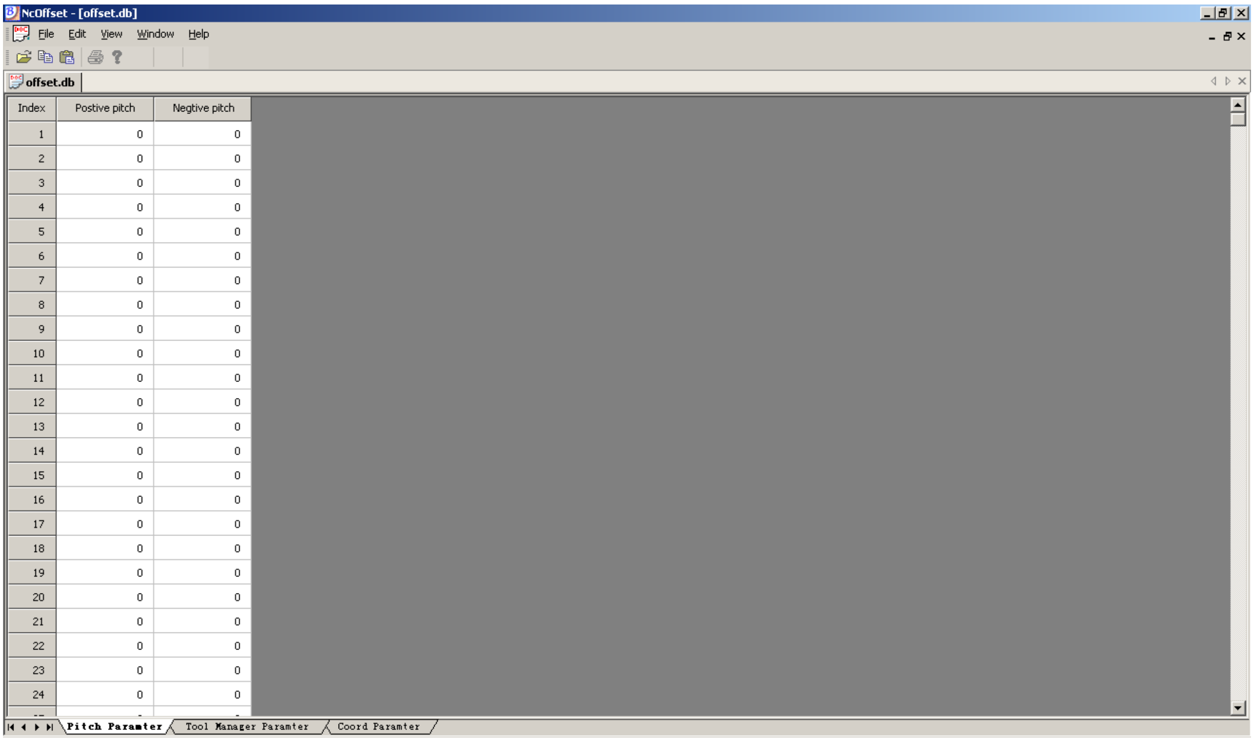


index	Tool Type	Number	Information	life	Max Life	Notify Life	Current Lift Value	Max Lift Value	Notify Life	Current status	Spindle Speed	Feed Speed	Lenght Compensate Number	Nos
1	112350	11	UBCR	500	2000	20	2000	100000	5000	3	500	70000	2	
2	112350	11	UBCR	500	2000	20	2000	100000	5000	3	500	70000	2	
3	369	11	UBTR	500	2000	20	2000	20000	999999	3	500	70000	2	
4	51	11	UBTR	500	2000	20	3000	100000	5000	3	500	70000	2	
5	112350	11	UBCR	500	2000	20	2000	100000	5000	3	500	70000	2	
6	112350	11	UBCR	500	2000	20	2000	100000	5000	3	500	70000	2	
7	112350	11	UBCR	500	2000	20	2000	100000	5000	3	500	70000	2	
8	112350	11	UBCR	500	2000	20	2000	100000	5000	3	500	70000	2	
9	112350	11	UBCR	500	2000	20	2000	100000	5000	3	500	70000	2	
10	20	11	UBTR	500	2000	20	2000	100000	5000	3	500	70000	2	
11	112350	11	UBCR	500	2000	20	2000	100000	5000	3	500	70000	2	
12	112350	11	UBCR	500	2000	20	2000	100000	5000	3	500	70000	2	
13	52	11	LNC-	700	3000	30	3000	200000	6000	0	900	80000	7	
14	112350	11	UBCR	500	2000	20	2000	100000	5000	3	500	70000	2	
15	112350	11	UBCR	500	2000	20	2000	100000	5000	3	500	70000	2	
16	112350	11	UBCR	500	2000	20	2000	100000	5000	3	500	70000	2	
17	112350	11	UBCR	500	2000	20	2000	100000	5000	3	500	70000	2	
18	112350	11	UBCR	500	2000	20	2000	100000	5000	3	500	70000	2	
19	112350	11	UBCR	500	2000	20	2000	100000	5000	3	500	70000	2	
20	112350	11	UBCR	500	2000	20	2000	100000	5000	3	500	70000	2	
21	112350	11	UBCR	500	2000	20	2000	100000	5000	3	500	70000	2	
22	112350	11	UBCR	500	2000	20	2000	100000	5000	3	500	70000	2	
23	112350	11	UBCR	500	2000	20	2000	100000	5000	3	500	70000	2	
24	112350	11	UBCR	500	2000	20	2000	100000	5000	3	500	70000	2	

Fig. 3-4 Editing of tool and offset parameter

3.3 Editing of the Pitch Error Compensation Data

Editing the pitch error compensation data is as Fig. 3-5.



Index	Postive pitch	Negtive pitch
1	0	0
2	0	0
3	0	0
4	0	0
5	0	0
6	0	0
7	0	0
8	0	0
9	0	0
10	0	0
11	0	0
12	0	0
13	0	0
14	0	0
15	0	0
16	0	0
17	0	0
18	0	0
19	0	0
20	0	0
21	0	0
22	0	0
23	0	0
24	0	0

Fig. 3-5 Editing of the pitch error compensation data

3.4 Editing of PLC Parameter

Editing the PLC parameter is as Fig. 3-6.

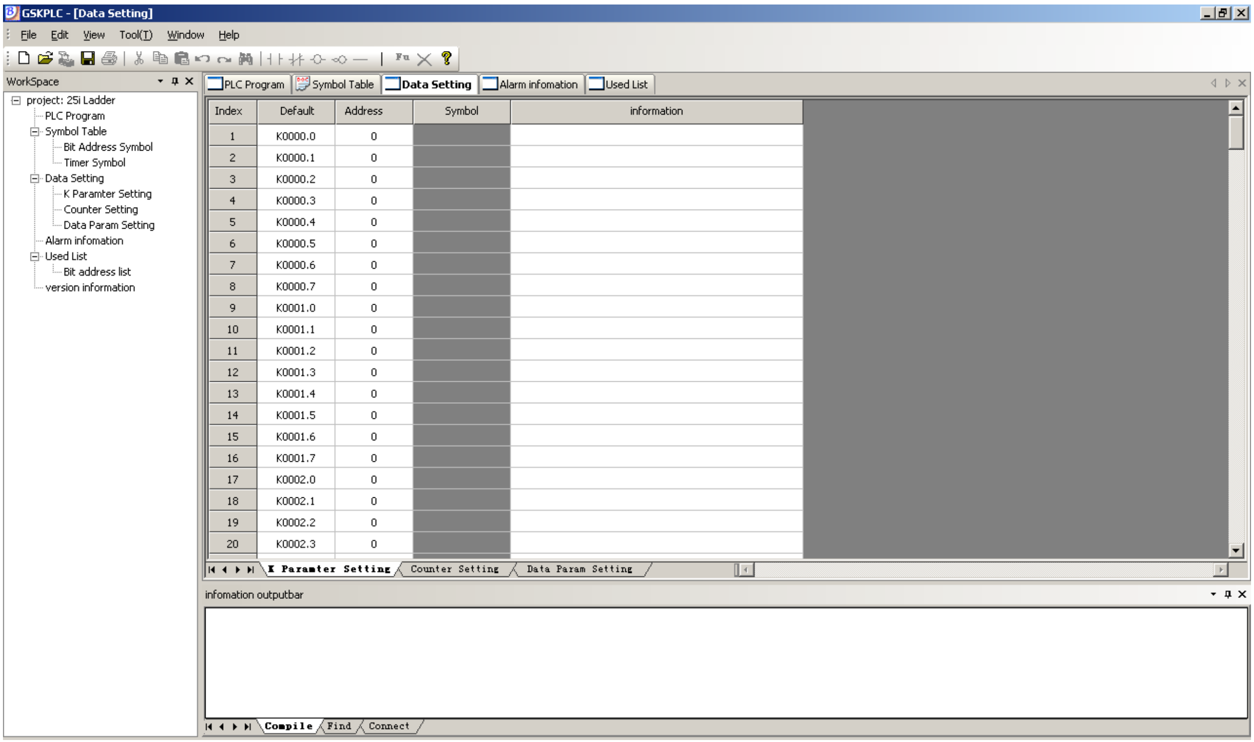


Fig. 3-6 Editing of PLC parameter

4 Parameter Explanation

[Parameter type]

The system parameters can be divided into several types based upon the following parameters (refer to the fig. 4-1).

Table 4-1 data type and data effective range

Data type	Effective data range
Bit	0 or 1
Bit axis	0 or 1
Integrated	—99999999～99999999
Integral axis	—99999999～99999999
Real number	—99999999～99999999
Real number axis	—99999999～99999999

The displayed number of axis type is determined on the total setting axis amount.

[Parameter explanation format]

The system parameter can be defined based on the following format.

Parameter number	Parameter significance explanation

It is important to notice that the cautions may occur in the notice column to remind the user

Note
1. Notice 1
2. Notice 2
3. Notice 3
4,……

4.1 Parameter Setting (1～99)

	7#	6#	5#	4#	3#	2#	1#	0#
0001			SEQ			INI		

[Data type] Bit

[Data range] 0 or 1

INI: Unit of input

0：In mm

1：In inches

SEQ: Automatic insertion of sequence numbers

0：Not performed

1：Performed

Note:
The incremental of sequence number is set in parameter No.1621.

[**Standard setting]** 0 0 0 0 0 0 0 0

	7#	6#	5#	4#	3#	2#	1#	0#
0002								RDG

[Data type] Bit

[Data range] 0 or 1

RDG: Remote diagnosis is

0：Not performed

1：Performed

[**Standard setting]** 0 0 0 0 0 0 0 0

0010	I/O CHANNEL selection	3

[Data type] Integrated type

[Data range] 0-4

[**Standard setting]**

Setting value	Significance
0	RS232C serial port
1	Reserved
2	Reserved
3	USB interface
4	Ethernet interface

4.2 Communication Parameter (100～999)

	7#	6#	5#	4#	3#	2#	1#	0#
0100					ASF			A2D

[Data type] Bit type

[Data range] 0 or 1

A2D: DSP loading method

0：DSP directly start mode

1：Loading DSP using cnc program

ASF：The current file of previous one is whether to save automatically while the file is loaded.

0：Yes

1：No

[**Standard set]** 0 0 0 0 0 0 0 1

0130	Ethernet IP address	192

[Data type] Integrated type
[Data range] 0-255

Note
For example: The value of IP:192.168.2.10 is 10 (192.168.2 is a fixed value)

200	GSK-LINK communication period	200000

[Data type] Integrated type
[Data unit] 10ns
[Data range] 10000～1000000(100us-10ms)

201	GSK-LINK command time	10000

[Data type] Integrated type
[Data unit] 10ns
[Data range] 100～1000000

202	GSK-LINK sampling time	80000

[Data type] Integrated type
[Data unit] 10ns
[Data range] 100～1000000

203	The length of period communication data	8

[Data type] Integrated type
[Data unit] Byte
[Data range] 6～16（Required in multiples of 2）

204	The maximum period repeated times	3

[Data type] Integrated type
[Data unit]
[Data range] 0～16

205	Servo communication ignorance	0

[Data type] Integrated
[Dtat unit]
[Data range] 0～1

> **Note:**
> The system may ignore the servo net communication when it is set to 1, which is mainly used for debugging; when this parameter is set, the power must be turned off before operation is continued.

4.3 Coordinate Parameter (1000～1199)

	7#	6#	5#	4#	3#	2#	1#	0#
1000							ISC	INM

[Data type] Bit

[Data range] 0 or 1

INM: Least command increment on the linear axis

0：In mm (Metric system)

1：In inches (Inch system)

ISC：The least move unit

0：0.001mm, 0.001deg

1：0.0001mm, 0.0001deg

> **Note:**
> When this parameter is set, the power must be turned off before operation is continued.

[**Standard setting]** 0 0 0 0 0 0 1 0

	7#	6#	5#	4#	3#	2#	1#	0#
1001						SFD	DLZ	

[Data type] Bit

[Data range] 0 or 1

DLZ: Function setting the reference position without dog

0：Disabled

1：Enabled

SFD: The function for shifting the reference position is

0：Not used

1：Used

[**Standard setting]** 0 0 0 0 0 0 0 0

	7#	6#	5#	4#	3#	2#	1#	0#
1002			EDN	EDP	HJZ			

[Data type] Bit
[Data range] 0 or 1

HJZ: When a reference position is already set:

0: Manual reference position return is performed with deceleration dogs.

1: Manual reference position return is performed using rapid traverse without deceleration dogs.

EDP: External deceleration signal in the positive direction for each axis

0: Valid only for the rapid traverse

1: Valid for rapid traverse and cutting feed

EDN: External deceleration signal in the negative direction for each axis

0: Valid only for rapid traverse

1: Valid for rapid traverse and cutting feed

[**Standard setting]** 0 0 0 0 1 0 0 0

	7#	6#	5#	4#	3#	2#	1#	0#
1004			ZMIx			RRLn		

[Data type] Bit axis
[Data range] 0 or 1

RRLn: Relative coordinates are

0: Not rounded by the amount of the shift per one rotation

1: Rounded by the amount of the shift per one rotation

ZMIx: Reference position return direction is set for each axis

0: In positive

1: In negative

> **Note:**
> When this parameter is set, the power must be turned off before operation is continued.

[**Standard setting]** 0 0 0 0 0 0 0 0

1020	Program axis name for each axis	

[Data type] Integrated axis
[Data range] 0～127

> **Note**
> The display name is ASCII code, and the allowable input values are X-88, Y-89, Z-90, A-65, B-66 and C-67.

1021	The attribute of each axis in the basic coordinate system	

[Data type] Integrated
[Data range] 0～7

Setting value	Significance
0	Neither the basic three axes nor a parallel axis
1	X axis of the basic three axes
2	Y axis of the basic three axes
3	Z axis of the basic three axes
4	Axis parallel to the X axis
5	Axis parallel to the Y axis
6	Axis parallel to the Z axis

1022	Servo logic address for each axis	

[Data type] Integrated
[Data range] 0～25

Note
The setting of servo logic address is related to the connection of servo network, the 1st slave station connected from the system P1 terminal is 0, according to this, servo logic address is its corresponding set value; Usually, set a same value both the control axis number and the controlled axis number.

	7#	6#	5#	4#	3#	2#	1#	0#
1023							ISRn	AXUn

[Data type] Bit axis
[Data range] 0 or 1

AXUn: Enabling for each axis is
0: Not used
1: Used

ISRn: It is either rotation axis or pallel axis for each axis
0: Pallel axis
1: Rotation axis

[**Standard setting]** 0 0 0 0 0 0 0 1

	7#	6#	5#	4#	3#	2#	1#	0#
1030	ITI	IDX				ABS	REL	RMOD

[Data type] Bit
[Data range] 0 or 1

RMOD: G code rotation command movement method
0: The approximate principle moves to the nearest position

1: Value magnitude moves

REL: Relative coordinate display of rotation axis

0: Out of the 360°

1: Within 360°

ABS: Absolute coordinate display of rotation axis

0: Out of the 360°

1: Within 360°

IDX: Index table indexing sequence.

0: Type A

1: Type B

ITI: The index function of the index table is:

0: Disabled

1: Enabled

[**Standard setting]** 0 0 0 0 0 0 0 0

	7#	6#	5#	4#	3#	2#	1#	0#
1031				G52	RLC			

[Data type] Bit

[Data range] 0 or 1

RLC: Local coordinate system is

0: Not cancelled by reset

1: Cancelled by reset

G52: In local coordinate system setting, a cutter compensation vector is

0: Not considered

1: Considered

[**Standard setting]** 0 0 0 0 1 0 0 0

1040	External workpiece origin offset value	0

[Data type] Real number axis

[Data unit] mm

[Data range] -9999.9999～9999.9999

1041	The origin offset amount of workpiece coordinate system 1(G54)	0

[Data type] Real number axis

[Data unit] mm

[Data range] -9999.9999～9999.9999

1042	The origin offset amount of workpiece coordinate system 2(G55)	0

[Data type] Real number axis
[Data unit] mm
[Data range] -9999.9999～9999.9999

1043	The origin offset amount of workpiece coordinate system 3(G56)	0

[Data type] Real number axis
[Data unit] mm
[Data range] -9999.9999～9999.9999

1044	The origin offset amount of workpiece coordinate system 4(G57)	0

[Data type] Real number axis
[Data unit] mm
[Data range] -9999.9999～9999.9999

1045	The origin offset amount of workpiece coordinate system 5(G58)	0

[Data type] Real number axis
[Data unit] mm
[Data range] -9999.9999～9999.9999

1046	The origin offset amount of workpiece coordinate system 6(G59)	0

[Data type] Real number axis
[Data unit] mm
[Data range] -9999.9999～9999.9999

1050	Coordinate value of the 1st reference position on each axis in the mechanical coordinate system	0

[Data type] Real number axis
[Data unit] mm
[Data range] -9999.9999～9999.9999

> **Note**:
> When this parameter is set, the power must be turned off before operation is continued.

1051	Coordinate value of the 2nd reference position on each axis in the mechanical coordinate system	0

[Data type] Real number axis
[Data unit] mm
[Data range] -9999.9999～9999.9999

> **Note**:
> When this parameter is set, the power must be turned off before operation is continued.

1052	Coordinate value of the 3rd reference position on each axis in the mechanical coordinate system	0

[Data type] Real number axis
[Data unit] mm
[Data range] -9999.9999～9999.9999

> **Note**
> When this parameter is set, the power must be turned off before operation is continued.

1053	Coordinate value of the 4th reference position on each axis in the mechanical coordinate system	0

[Data type] Real number axis
[Data unit] mm
[Data range] -9999.9999～9999.9999

> **Note**
> When this parameter has been set, the power must be turned off before operation is continued.

1060	Amount of a shift per one rotationof a feed axis	0

[Data type] Real number axis
[Data unit] mm or degree
[Data range] 0～999.9999

> **Note**
> When this parameter is set, the power must be turned off before operation is continued.

1068	Amount of rotation angle per one rotation of a revolution axis	360

[Data type] Real number axis
[Data unit] Degree
[Data range] 0～9999.9999

> **Note**
> 1. This parameter is used during cylinderical interpolation.
> 2. When this parameter is set, the power must be turned off before operation is continued.

	7#	6#	5#	4#	3#	2#	1#	0#
1070		LZR	XWG		OT3	OT2		OUT

[Data type] Bit
[Data range] 0 or 1

OUT: The area inside or outside of the stored stroke check 2 is set as an inhibition area.
0: Inside
1: Outside

OT2: Whether stored stroke check 2 is checked for each axis is set.
0: Stored stroke check 2 is not checked.
1: Stored stroke check 2 is checked.

OT3: Whether stored stroke check 3 is checked for each axis is set.
0: Stored stroke check 3 is not checked.
1: Stored stroke check 3 is checked.

XWG: Overtravel alarm switch
0: Alarm ON
1: Alarm OFF

LZR: Checking of stored stroke check 1 during the time from power-on to the manual reference position return.
0: Not checked
1: Checked

[**Standard setting]** 0 0 0 0 0 0 0 0

1080	Coordinate value of stored stroke check 1 in the positive direction on each axis.	999999.9999

[Data type] Real number axis
[Data unit] mm
[Data range] 0～999999.9999

1081	Coordinate value of stroed stroke check 1 in the negative direction on each axis.	-999999.9999

[Data type] Real number axis
[Data unit] mm
[Data range] -999999.9999～0

1082	Coordinate value of stored stroke check 2 in the positive direction on each axis.	999999.9999

[Data type] Real number axis
[Data unit] mm
[Data range] 0～999999.9999

1083	Coordinate value of stored stroke check 2 in the negative direction on each axis.	-999999.9999

[Data type] Real number axis
[Data unit] mm
[Data range] -999999.9999～0

1084	Coordinate value of stored stroke check 3 in the positive direction on each axis.	999999.9999

[Data type] Real number axis
[Data unit] mm
[Data range] 0～999999.9999

1085	Coordinate value of stored stroke check 3 in the negative direction on each axis.	-999999.9999

[Data type] Real number axis
[Data unit] mm
[Data range] -999999.9999～0

1100	Machine struction type	12

[Data type] Integrated
[Data range] 0～21

1101	Controlled axis number of the 1st rotation axis	4

[Data type] Integrated
[Data range] 0～5

1102	The axis direction of the 1st rotation axis	2

[Data type] Integrated
[Data range] 0～5

1103	The rotation direction of the 1st rotation axis	1

[Data type] Integrated
[Data range] 0: negative/ 1: positive

1104	The 1st rotation axis is an inclination angle for angular aixs	0

[Data type] Real number
[Data unit] deg
[Data range] -999999.9999 ~ 999999.9999

1105	The controlled axis number of the 2nd rotation axis	5

[Data type] Integrated
[Data range] 0～5

1106	The axis diretion of the 2nd rotation axis	3

[Data type] Integrated
[Data range] 0～5

1107	The rotation direction of the 2nd rotation axis	1

[Data type] Integrated
[Data range] 0: negative/ 1: positive

1108	The 2nd rotation axis is an inclination angle of the angular axis	0

[Data type] Real number
[Data unit] deg
[Data range] -999999.9999 ~ 999999.9999

1109	The axis direction of tool axis	3

[Data type] Integrated
[Data range] 0~3

1110	The position of index table	0

[Data type] Real number axis
[Data unit] mm
[Data range] -9999.9999～9999.9999

Note
Coordinate parameters of three axes X, Y and Z are included

1111	Offset vector between the 1st and 2nd working table rotation axes	0

[Data type] Real number axis
[Data unit] mm
[Data range] -9999.9999～9999.9999

Note
Coordinate parameters of three axes X, Y and Z are included

1112	Offset vector between the tool axis and tool rotation axis	0

[Data type] Real number axis
[Data unit] mm
[Data range] -9999.9999～9999.9999

Note
Coordinate parameters of three axes X, Y and Z are included

1113	Offset vector between the 2nd and 1st tool rotation axes	0

[Data type] Real number axis
[Data unit] mm
[Data range] -9999.9999～9999.9999

Note
Coordinate parameters of three axes X, Y and Z are included

1114	Tool post offset	0

[Data type] Real number axis
[Data unit] mm
[Data range] -9999.9999～9999.9999

4.4 Feedrate Parameter (1200～1399)

	7#	6#	5#	4#	3#	2#	1#	0#
1200		RDR		RF0				RPD

[Data type] Bit
[Data range] 0 or 1

RPD: Manual rapid traverse during the period from the power-on time to the completion of the reference position return.

0: Disabled

1： Enabled

RF0: When the rapid feedrate override is F0,

0: The machine tool does not stop moving.

1: The machine tool stops moving.

RDR: When the rapid traverse is performed,

0: Dry run is disabled.

1: Dry run is enabled.

[**Standard setting]** 0 0 0 0 0 0 0 0

1210	Dry run speed (common to all axes)	10000

[Data type] Real number
[Data unit] mm/min
[Data range] 0～1000000

> **Note**
> The dry run speed is set when the manual feedrate is set to 100%.

1211	The cutting feedrate occurs by default in the automatic mode	1000

[Data type] Real number
[Data unit] mm/min
[Data range] 0～1000000

> **Note**
> The feedrate is set when the automatic feedrate is set to 100%.

1224	The maximum cutting composite feedrate (common to all axes)	4000

[Data type] Real number
[Data unit] mm/min
[Data range] 0～1000000

1225	Maximum cutting feedrate for each axis in the automation mode	4000

[Data type] Real number axis
[Data unit] mm/min or degree/min
[Data range] 0～1000000

1226	Rapid traverse rate for each axis in the automation mode	10000

[Data type] Real number axis
[Data unit] mm/min or degree/min
[Data range] 0～1000000

Note 1. The rapid traverse rate is set when the rapid traverse rate is set to 100%.

1227	The top allowable speed of move axis is shown when it is started or stopped suddenly during the linkage.	1000

[Data type] Real number axis
[Data unit] mm/min or degree/min
[Data range] 0～1000000

1228	The top allowable speed of move axis is shown when it is performed in negative suddenly during the linkage.	1000

[Data type] Real number axis
[Data unit] mm/min or degree/min
[Data range] 0～1000000

1229	The top allowable acceleration speed of move axis is shown when it is performed in negative suddenly during the linkage.	0.3 [The rotation axis is 75]

[Data type] Real number axis
[Data unit] m/s^2
[Data range] 0～90000000

1231	F0 speed of rapid traverse feedrate override (common to all axes)	100

[Data type] Real number
[Data unit] mm/min or degree/min
[Data range] 0～100000

1232	Feedrate in manual continuous feed (JOG feed) for each axis	1000

[Data type] Real number axis
[Data unit] mm/min or degree/min
[Data range] 0～100000

> **Note**
> JOG feedrate is set when manual feedrate is 100%.

1233	Manual rapid traverse rate for each axis	3000

[Data type] Real number axis
[Data unit] mm/min or degree/min
[Data range] 0～100000

> **Note**
> 1. Rapid traverse rate is set when the rapid traverse rate is 100%, and the value set by No.1226 [the top speed at rapid traverse rate] is employed when this parameter is set to 0.

1234	FL rate of the reference position return for each axis	300

[Data type] Real number axis
[Data unit] mm/min or degree/min
[Data range] 0~15000

1235	Reference position return speed for each axis	4000 [the rotation axis is 2000]

[Data type] Real number axis
[Data unit] mm/min or degree/min
[Data range] 0～100000

1236	The 2^{nd} FL speed of reference position return for each axis	7 [The rotation axis is 2]

[Data type] Real number axis
[Data unit] mm/min or degree/min
[Data range] 0～100000

1239	The top speed of manual feed	5000

[Data type] Real number
[Data unit] mm/min or degree/min
[Data range] 0～100000

1240	The maximum operation speed of single-step	10000

[Data type] Real number
[Data unit] mm/min or degree/min
[Data range] 0～100000

1241	The maximum feed speed of MPG	15000

[Data type] Real number
[Data unit] mm/min or degree/min
[Data range] 0～100000

4.5 Interpolation and Acceleration/Deceleration Control Parameter (1400～1599)

	7#	6#	5#	4#	3#	2#	1#	0#
1400		PACD			PPCK			

[Data type] Bit
[Data range] 0 or 1

PPCK: In-position check
0: Not performed
1: Performed

PACD: The acceleration/deceleration mode before the interpolation
0: Linear type
1: Type S

[**Standard setting]** 0 0 0 0 0 0 0 0

	7#	6#	5#	4#	3#	2#	1#	0#
1401	ALS	WFM		DEF			EFL	

[Data type] Bit

[Data range] 0 or 1

EFL: The flag of the transition of small line segment fold

0: No employed

1: Employed

DEF: The speed is whether to control the speed variable when the controlled axis is stopped suddenly

0: Not considered

1: Considered

WFM: MPG interpolation mode

0: It is treated by the impounding reservoir mode

1: It is treated by the real-time mode

ALS: Automatic corner feed function

0: Invalid

1: Valid

[**Standard setting]** 0 0 0 0 0 0 0 0

	7#	6#	5#	4#	3#	2#	1#	0#
1403			RCOK	RBK			HXS	

[Data type] Bit

[Data range] 0 or 1

HXS: The rotation diretion between MPG and each axis

0: Different

1: Same

RBK: The backlash compensation is performed between the cutting and rapid traverse.

0: Not separately

1: Separately

RCOK: Backlash compensation

0: Not performed

1: Performed

[**Standard setting]** 0 0 0 0 0 0 1 0

1404	Curve frequency of Nurbs interpolation	3

[Data type] Integrated

[Data unit] times

[Data range] 1～4

1405	Standard indensity setting of Nurbs curve interpolation	10

[Data type] Integrated
[Data unit] Point/mm
[Data range] 5～1000

1406	Pre-read sections in its look-ahead treatment	1000

[Data type] Integrated
[Data unit] Section
[Data range] 0～2000

1407	The maximum program sections of Nurbs curve interpolation	200

[Data type] Integrated
[Data unit] Section
[Data range] 10～500

1409	Prospective treatment program section amount	10

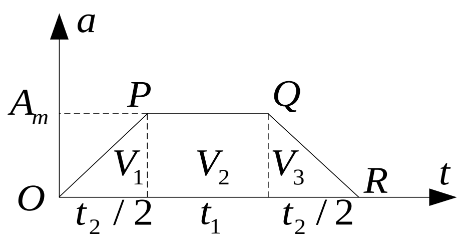
[Data type] Integrated
[Data unit] Section
[Data range] 0～2000

Note
The prospective program section amount is set when adopting the prospect, and the 0 does not indicate prospect.

1410	Acceleration/deceleration type S and time constant T1 are specified before the rapid traverse feed is performed	100

[Data type] Integrated axis
[Data unit] ms
[Data range] 1～4000

The parameter value of its corresponding number is indicated by the P+ parameter number, such as, PA1233 means the No. 1233 parameter. The acceleration/deceleration calculation mode of type S is shown below, where, the t_1 indicates an uniform acceleration time, t_2 means a jerk and decelerating acceleration time, and A_m is the maximum acceleration.



As the above figure mentioned, the ladder area is: $V_m = \frac{(t_1+t_1+t_2)*A_m}{2}$

The maximum acceleration calculation is concluded: $A_m = \frac{V_m}{(2t_1+t_2)}$

And, the calculation of jerk time is: $J_m = \frac{2A_m}{t_2}$

The linear acceleration/deceleration can be regarded as a special example when the type S acceleration/deceleration is on the state of $t_2=0$.

Before the Goo rapid traverse, the maximum acceleration calculation format of acceleration/deceleration type S is:

$A_{m00} = \frac{2\times P_{1226}}{(2\times P_{1410}+P_{1411})}$, And the maximum acceleration calculation format of jerk type S before the Goo rapid traverse is $J_{m00} = \frac{2A_{m00}}{P_{1411}}$.

Note
1. When the acceleration or jerk calculation is used this format during the actual application; it is very necessary to note that the unit conversion must be performed in terms of the unit of parameters.

1411	S-type acceleration/deceleration time constant T_2 at the rapid traverse feed	100

[Data type] Integral axis
[Data unit] ms
[Data range] 0～4000

Note
1. When the acceleration or jerk calculation is used this format during the actual application; it is very necessary to note that the unit conversion must be performed in terms of the unit of parameters.

1440	The maximum acceleration speed	0.4 [The rotation axis is 100]

[Data type] Real number axis
[Data unit] m/ s^2, the rotation axis is: degree/s^2, a general rotation axis value is up to 250 folds related to the parallel axis.
[Data range] 0～25000

Note
It is only valid to the linear acceleration/deceleration control.

1442	The maximum acceleration speed of the circular arc interpolation feed	0.5

[Data type] Real number
[Data unit] m/ s^2
[Data range] 0～25000

1444	Mechanical zero return acceleration speed by default	0.139[The rotation axis is 80]

[Data type] Real number axis
[Data unit] m/s^2, the rotation axis is: degree/s^2, a general rotation axis value is up to 250 folds related to the parallel axis.
[Data range] 0～25000

1445	The acceleration speed is performed during deceleration when dwelling or RESETTING in the process of operation	0.5

[Data type] Real number
[Data unit] m/ s^2
[Data range] 0～25000

1446	MPG acceleration speed	0.5

[Data type] Real number
[Data unit] m/s^2
[Data range] 0～25000

1447	Manual acceleration speed	0.5

[Data type] Real number
[Data unit] m/s^2
[Data range] 0～25000

1471	Cutting feed in-position accuracy	0.001

[Data type] Real number
[Data unit] mm
[Data range] 0～1

1472	Circular arc interpolation control accuracy	0.001

[Data type] Real number
[Data unit] mm
[Data range] 0～1

1473	The maximum contour error of the system	0.001

[Data type] Real number
[Data unit] mm
[Data range] 0～1

1480	The acceleration/deceleration S-type time constant T1 before cutting feed	16

[Data type] Integral axis
[Data unit] ms
[Data range] 0～4000

The maximum acceleration calculation format of acceleration/deceleration type S before the Goo rapid traverse is:

$A_{m01} = \frac{2\times P_{1225}}{(2\times P_{1480} + P_{1481})}$, and the maximum jerk calculation format of acceleration/deceleration type S before the Goo rapid traverse is $J_{m01} = \frac{2A_{m01}}{P_{1481}}$.

Note 1. When the acceleration or jerk calculation is used this format during the actual application; it is very necessary to note that the unit conversion must be performed in terms of the unit of parameters.

1481	Accleration/deceleration S type time constant T2 before cutting feed	16

[Data type] Integral axis
[Data unit] ms
[Data range] 0～4000

Note

1. When the acceleration or jerk calculation is used this format during the actual application; it is very necessary to note that the unit conversion must be performed in terms of the unit of parameters.

1493	Judging the least distance of Nurbs interpolation deceleration point	1．000

[Data type] Real number

[Data unit] mm

[Data rang] 0～50.0000mm

1494	Judge the least corner of Nurbs interpolation deceleration point	10

[Datat type] Real number

[Data unit] deg

[Data range] 0～30

1495	The least conversion corner of fold line transition	150

[Datat type] Real number

[Data unit] deg

[Data range] 120~180

1500	Two blocks' boundary corner of the automatic corner deceleration	0

[Datat type] Real number

[Data unit] deg

[Data range] 0~60

1501	The lowest federate of automatic corner deceleration	120

[Datat type] Real number

[Data unit] mm/min

[Data range] 60～1000

1502	The least circular arc cutting feed deceleration rate inside the automatic corner override	50

[Datat type] Real number

[Data unit] ％

[Data range] 0.00～100.00

1503	The start distance of the internal corner override	1

[Datat type] Real number
[Data unit] 0.1mm
[Data range] 0～4000

1504	The end distance of the internal corner override	1

[Datat type] Real number
[Data unit] 0.1mm
[Data range] 0～3999

1505	The lowest speed of circular interpolation	200

[Datat type] Real number
[Data unit] mm/min
[Data range] 0～9999.9999

4. 6 Editing Parameter Display (1600～1799)

	7#	6#	5#	4#	3#	2#	1#	0#
1601			ENG	CHI				

[Data type] Bit
[Data range] 0 or 1

ENG, CHI: Language selection

ENG	CHI	Language display
0	*	**Simplified Chinese**
1	0	**English**

[**Standard setting]** 0 0 0 1 0 0 0 0

	7#	6#	5#	4#	3#	2#	1#	0#
1603	DAC	DAL	DRC	DRL				

[Data type] Bit
[Data range] 0 or 1

DRL: The relative position display is whether to consider the tool length compensation
0: Considered
1: Not considered

DRC: The relative position display is whether to consider the tool radius compensation
0: Considered
1: Not considered

DAL: The absolute position display is whether to consider the tool length compensation

0: Considered

1: Not considered

DAC: The absolute position display is whether to consider the tool radius compensation

0: Considered

1: Not considered

[**Standard setting]** 0 0 0 0 0 0 0 0

	7#	6#	5#	4#	3#	2#	1#	0#
1605		NPA						PLCD

[Data type] Bit

[Data range] 0 or 1

PLCD: PLC ladder diagram display

0: Not displayed

1: Displayed

NPA: Whether to shift to an alarm screen when the alarm occurs.

0: No

1: Yes

[**Standard setting]** 0 0 0 0 0 0 0 1

	7#	6#	5#	4#	3#	2#	1#	0#
1610				NE9				NE8

[Data type] Bit

[Data range] 0 or 1

NE8: Whether to forbid the subprogram edit of the program numbers from 8000 to 8999

0: Not forbidden

1: Forbidden

NE9: Whether to forbid the subprogram edit of the program numbers from 9000 to 9999

0: Not forbidden

1: Forbidden

[**Standard setting]** 0 0 0 1 0 0 0 1

1621	Automatically insert the incremental value in sequence number	10

[Data type] Integrated

[Data range] 0～9999

1640	The required machining parts are added 1	0

[Data type] Integrated

[Data range] 0～99999999

1641	The required machining parts are added 2	0

[Data type] Integrated
[Data range] 0～99999999

1642	The required machining parts are added 3	0

[Data type] Integrated
[Data range] 0～99999999

1671	Main surface ground colour	

[Data type] Bit

1672	Fixed output color	

[Data type] Bit

1673	Dynamic output color	

[Data type] Bit

1674	Selected basis color	

[Data type] Bit

1675	Selected font color	

[Data type] Bit

1676	The ground color of controllable parts	

[Data type] Bit

1677	The selected color of controllable parts	

[Data type] Bit

1678	The ground color of input column	

[Data type] Bit

1679	The font color of input column	

[Data type] Bit

1680	The ground color of title column	

[Data type] Bit

1681	The font color of title column	

[Data type] Bit

1682	The ground color of state column	

[Data type] Bit

1683	The font color of state column	

[Data type] Bit

1684	Alarm color	

[Data type] Bit

1685	Setting-out color	

[Data type] Bit

1686	Shared color	

[Data type] Bit

	7#	6#	5#	4#	3#	2#	1#	0#
1687						DEF3	DEF2	DEF1

[Data type] Bit
[Data range] 0 or 1

Note
DEF1-DEF3, the default color configuration program, the above-mentioned configuration color can be regarded as valid as long as all default bits are set to 0.

4. 7 Programming Parameter (1800～1999)

	7#	6#	5#	4#	3#	2#	1#	0#
1800								DPI

[Data type] Bit

[Data range] 0 or 1

DPI: The decimal point is ignored when programming

0: It is treated as the least set unit

1: It is regarded as mm, sec

[**Starndard setting]** 0 0 0 0　0 0 0 1

	7#	6#	5#	4#	3#	2#	1#	0#
1801	G23	CLR			G91	G19	G18	G01

[Data type] Bit

[Data range] 0 or 1

G01: The modul issues when the power is turned on or off

0: G00 mode

1：G01 mode

G18, G19: When the power is turned on or off, the panel selection is:

G19	G18	Panel selection
0	0	G17
0	1	G18
1	0	G19

G91: When the power is turned on or off, its set is:

0: G90 mode

1: G91 mode

CLR: When the reset is performed

0: The system is reset

1: The system is eliminated

G23: When the power is turned on

0: G22 mode

1: G23 mode

[**Standard setting]** 1 0 0 0　0 0 0 0

	7#	6#	5#	4#	3#	2#	1#	0#
1802		AD2						

[Data type] Bit

[Data range] 0 or 1

AD2: More than two same addresses are specified in a same command.

0: Command is valid

1: System alarm

[**Standard set]** 0 0 0 0　0 0 0 0

	7#	6#	5#	4#	3#	2#	1#	0#
1803		M3B					POL	

[Data type] Bit
[Data range] 0 or 1

POL: Decimal point command address program
0: Used
1: Not used
M3B: M code number can be specified in program
0: One
1: Up to 3

[**Standard setting]** 0 1 0 0 0 0 0 0

1810	Allowable error of circular arc radius	0.01

[Data type] Real number
[Data unit] mm
[Data range] 0～9999.9999

Note
The circular arc radius error is not to be checked when the set value is 0.

	7#	6#	5#	4#	3#	2#	1#	0#
1850	SCR	XSC			SCL			RIN

[Data type] Bit
[Data range] 0 or 1

RIN: The rotation angle of its coordinate rotation
0: Absolute coordinate command
1: G90/G9 command
SCL: Scaling for each axis
0: Disabled
1：Enabled
XSC: Scaling override for each axis
0: Disabled
1：Enabled
SCR: The scaling override unit
0: 0.001 times
1: 0.0001 times

[**Standard setting]** 0 0 0 0 0 0 0 0

1860	The rotation angle is used when the angle in coordinate rotation does not occur.	0

[Data type] Real number
[Data unit] deg
[Data range] -360.000～ 360.000

1861	The scaling override is used when the scaling command override does not occur.	1

[Data type] Real number
[Data range] 0～ 99.999

1862	Scaling for each axis	1

[Data type] Real number axis
[Data range] 0～ 99.999

	7#	6#	5#	4#	3#	2#	1#	0#
1870							PDI	MDL

[Data type] Bit
[Data range] 0 or 1

MDL: Single direction positioning G code (G60)

0: The modul code does not set

1: Modul code

PDI: Single direction positioning in-position check

0: Not performed

1: Performed

[**Standard set]** 0 0 0 0 0 0 0 0

1880	The direction and overtravel amount of single direction positioning for each axis	0

[Data type] Real number axis
[Data unit] mm
[Data range] -999.9999～ 999.9999

1931	The least angle of index table	0

[Data type] Real number
[Data unit] deg
[Data range] 0～360.000

	7#	6#	5#	4#	3#	2#	1#	0#
1940	SKF							

[Data type] Bit
[Data range] 0 or 1

SKF: Whether the G31 skip command is valid to the dry run, override and authomatic acceleration/deceleration:

0: Valid

1: Invalid

[**Standard set]** 0 0 0 0 0 0 0 0

	7#	6#	5#	4#	3#	2#	1#	0#
1950	MOU							

[Data type] Bit
[Data range] 0 or 1

MOU: Whether to input the M, S, T and B codes when the program is restarted:

0: Not output

1: Output

[**Standard set]** 0 0 0 0 0 0 0 0

1960	The move sequence for each axis moves to the program restart position	

[Data type] Integral axis
[Data range] 0～ Controllable axis number

	7#	6#	5#	4#	3#	2#	1#	0#
1971	ESC	ESR						

[Data type] Bit
[Data range] 0 or 1

ESR: External program number index

0: Ineffective

1: Effective

ESC: The reset is input from ESTB input to index.

0: Index performed

1: Not performed

[**Standard set]** 0 0 0 0 0 0 0 0

4.8 Fixed Cycle Parameter (2000～2099)

	7#	6#	5#	4#	3#	2#	1#	0#
2000		M5B	RD2	RD1				FXY

[Data type] Bit

[Data range] 0 or 1

FXY: The drilling axis in the drilling canned cycle is:

0: Always the Z-axis

1: The axis selected by the program

RD2, RD1 Set the retraction axis direction of G76 or G87

RD2	RD1	G17	G18	G19
0	0	+X	+Z	+Y
0	1	-X	-Z	-Y
1	0	+Y	+X	+Z
1	1	-Y	-X	-Z

M5B: G76 G87 spindle orientation

0: Outputs M05 before an orientated spindle stops

1: Not ouput M05 before an oriented spindle stops

[**Standard setting]** 0 0 0 0 0 0 0 0

2010	Return d of high speed peck drilling G73	0.5

[Data type] Real number

[Data unit] mm

[Data range] 0～99.9999

2011	Clearance d of canned cycle G83	0.5

[Data type] Real number

[Data unit] mm

[Data range] 0～99.9999

2034	Clearance of small diameter peck drilling cycle	0.5

[Data type] Real number

[Data unit] mm

[Data range] 0～99.9999

4.9 Rigid Tapping Parameter (2100～2299)

2112	Return or clearance in peck tapping cycle	0.5

[Data type] Real number
[Data unit] mm
[Data range] 0～99.9999

2140	The maximum speed of spindle (the 1st step gear) in rigid tapping	1000

[Data type] Integrated
[Data unit] r/min
[Data range] 0～9999

2141	The maximum speed of spindle (the 2nd step gear) in rigid tapping	1000

[Data type] Integrated
[Data unit] r/min
[Data range] 0～9999

2142	The maximum speed of spindle (the 3rd step gear) in rigid tapping	1000

[Data type] Integrated
[Data unit] r/min
[Data range] 0～9999

2170	The position control circuit gain is performed (the 1st step gear) between spindle and tapping axis when the rigid tapping is performed.	1000

[Data type] Integrated
[Data unit] 0.01/s
[Data range] 0～9999

> **Note**
> When this parameter is set, the power must be turned off before operation is continued.

2171	The position control circuit gain is performed (the 2nd step gear) between spindle and tapping axis when the rigid tapping is performed.	1000

[Data type] Integrated
[Data unit] 0.01/s

[Data range] 0～9999

> **Note**
> When this parameter is set, the power must be turned off before operation is continued.

2172	The position control circuit gain is performed (the 3rd step gear) between spindle and tapping axis when the rigid tapping is performed.	1000

[Data type] Integrated
[Data unit] 0.01/s
[Data range] 0～9999

> **Note**
> When this parameter is set, the power must be turned off before operation is continued.

2180	Spindle circuit gain coefficient in rigid tapping (the 1st step gear)	1000

[Data type] Integrated
[Data range] 0～32767

2181	Spindle circuit gain coefficient in rigid tapping (the 2nd step gear)	1000

[Data type] Integrated
[Data range] 0～32767

2182	Spindle circuit gain coefficient in rigid tapping (the 3rd step gear)	1000

[Data type] Integrated
[Data range] 0～32767

2210	Spindle backlash compensating value of rigid tapping (the 1st step gear)	10

[Data type] Integrated
[Data unit] Check unit
[Data range] 1～127

2211	Spindle backlash compensating value of rigid tapping (the 2nd step gear)	10

[Data type] Integrated

[Data unit] Check unit
[Data range] 1～127

2212	Spindle backlash compensating value of rigid tapping (the 3rd step gear)	10

[Data type] Integrated
[Data unit] Check unit
[Data range] 1～127

2221	Return value of rigid tapping	0.5

[Data type] Real number
[Data unit] mm
[Data range] 0～9999.9999

4.10 Parameter of Manual, Auto and MPG Operation (2300～2499)

	7#	6#	5#	4#	3#	2#	1#	0#
2300	HNGD		JAG	HPF	HCL	IHD		JHD

[Data type] Bit
[Data range] 0 or 1

JHD: MPG feed in JOG mode or incremental feed in MPG feed mode
0: Invalid
1: Valid

IHD: The travel increment of MPG is:
0: Output unit
1: Input unit

HCL: Whether the softkey is cleared the display of MPG interruption
0: Disabled
1: Enabled

HPF: When a MPG feedrate exceeding the rapit traverse rate is issued, the rate is clamped at the rapid traverse rate and exceeded part of pulse
0: Ignored
1: Not ignored, but stored in the CNC

JAG: Manual absolute switch of system
0: OFF
1: ON

HNGD: Axis movement direction for rotation direction of MPG
0: Same in direction
1: Reverse in direction

[**Standard setting]** 0 0 0 0 0 0 0 0

2310	Least command increment setting for JOG feed	0.001

[Data type] 2-word
[Data unit] mm
[Data range] 0～10.000

2320	Number of MPG	

[Data type] Byte
[Data range] 1～3

2321	MPG feedrate override m	

[Data type] Byte
[Data range] 1～127

2322	MPG feedrate override n	

[Data type] Word
[Data range] 1～999

MP2(G19#5)	MP1(G19#4)	MPG movement
0	0	Least input increment ×1
0	1	Least input increment ×10
1	0	Least input increment ×m
1	1	Least input increment ×n

2323	Allowable number of pulses that can be accumulated during MPG feed	

[Data type] 2-Word
[Data range] 0～99999999

4.11 Parameter (2500～2599) Input/Output

	7#	6#	5#	4#	3#	2#	1#	0#
2501	LTM		DEC		SWI			

[Data type] Bit
[Data range] 0 or 1

SWI: Position switch symbol

0: Invalid

1: Valid

DEC: Deceleration signal for reference position return

0: Deceleration is applied when the signal is 0.

1: Deceleration is applied when the signal is 1.

LTM: Hard limit treatment method

0: Hard limit deceleration treatment

1: Stops immediately

[**Standard setting]** 0 0 0 0 0 0 0 0

2510	The delay time of strobe signal MF, SF, TF and BF	64

[Data type] Integrated

[Data unit] ms

[Data range] 16～32767

2511	The acceptable width of the M, S, T and B completion signal	64

[Data type] Integrated

[Data unit] ms

[Data range] 16～32767

2512	Distributed address to the skip signal	0

[Data type] Integrated

[Data range] 0～127

Note

1. It is invalid when the parameter is less than 10.

2513	Distributed address to the measure arrival signal	0

[Data type] Integrated

[Data range] 0～127

Note

1. It is invalid when the parameter is less than 10.

2518	Output time of reset signal	100

[Data type] Integrated

[Data unit] ms

[Data range] 0～1000

2540-2555	Position switch corresponds servo axis	0

[Data type] Integrated
[Data range] 0～6

Note
1. Position switch function is valid when the bit SWI is set to 1.
2. Position switch function is invalid when the bit SWI is set to 0.

2556-2571	The maximum range of position switch positive	0

[Data type] Integrated
[Data range] 0～99999999

2572-2587	The maximum range of position switch negative	0

[Data type] Integrated
[Data range] -99999999～ 0

4.12 Tool Administration Parameter (2600～2799)

	7#	6#	5#	4#	3#	2#	1#	0#
2600							TLB	

[Data type] Bit
[Data range] 0 or 1

TLB: Tool length compensation selection

0: Tool compensation A (Always Z axis irrespective of plane specification)

1: Tool compensation B (Axis perpendicular to plane specification)

[**Standard setting]** 0 0 0 0 0 0 1 0

	7#	6#	5#	4#	3#	2#	1#	0#
2601	ODI	LVK				CCN		

[Data type] Bit
[Data range] 0 or 1

CCN: G28 command moves to the intermediate point in radius compensation

0: The radius compensation is cancelled in movement to an intermediated position

1: The radius compensation is not cancelled in movement to an intermediate position, but is cancelled in movement to the reference position.

LVK: Tool length offset value

0: Not cleared, but held by reset

1: Cleared by reset

ODI: A cutter compensation amount is set using:

0: A diameter

1: A radius

[**Standard setting]** 0 0 0 0 0 1 0 0

	7#	6#	5#	4#	3#	2#	1#	0#
2602			G39			TPH	CNI	OIM

[Data type] Bit

[Data range] 0 or 1

OIM: When the unit is switched between the inch and metric systems, automatic tool offset value conversion is:

0: Not performed

1: Performed

CNI: Interference check for radius compensation is:

0: Performed

1: Not performed

TPH: Tool format of cutter compensation

0: Type A

1: Type B

G39: The corner rounding function in radius compensation mode is:

0: Disabled

1: Enabled

[**Standard setting]** 0 0 0 0 0 0 0 0

2610	The vector limit value is ignored when cutter compensation moves along with the corner external.	10

[Data type] Real number

[Data unit] mm

[Data range] 0～99.9999

2611	The maximum amount of tool wear compensation value	60

[Data type] Real number

[Data unit] mm

[Data range] 0～99.9999

2651	Automatic tool length compensation measure speed	1000

[Data type] Real number

[Data unit] mm/min

[Data range] 0～15000

2652	The r value of automatic tool length compensation measure	0

[Data type] Real number
[Data range] 0～9999.9999

2653	The e value of automatic tool length compensation	0

[Data type] Real number
[Data range] 0～9999.9999

	7#	6#	5#	4#	3#	2#	1#	0#
2700			EIS			LTM	GS2	GS1

[Data type] Bit
[Data range] 0 or 1

GS2 and GS1 are composed of tool number combination

GS2	GS1	Group	Tool number
0	0	1～16	1～16
0	1	1～32	1～8
1	0	1～64	1～4
1	1	1～128	1～2

LTM: Tool life
0: Specified by the number of times
1: Specified by time

EIS: When the life of a tool is measured in time-based units:
0: The life is counted every four seconds.
1: The life is counted every second.

[**Standard setting]** 0 0 0 0   0 0 0 0

2710	The omissive number of tool life administration	

[Data type] Integrated
[Data range] 0～ 9999

2711	M code for restarting tool life count	

[Data type] Integrated
[Data range] 0～255

2712	The rest of tool life (frequence of use)	

[Data type] Integrated
[Data range] 0～ 9999

2713	The rest of tool life (time of use)	

[Data type] Integrated
[Data unit] min
[Data range] 0～ 9999

4.13 Pitch Compensation Parameter (2800～2999)

	7#	6#	5#	4#	3#	2#	1#	0#
2800							WDIR	SCRW

[Data type] Bit
[Data range] 0 or 1

SCRW: Pitch compensation

0: Not performed

1: Performed

WDIR: Pitch compensation selection

0: Unidirection

1: Bidirection

[**Standard set]** 0 0 0 0 0 0 0 1

> **Note**
> When this parameter is set, the power must be turned off before operation is continued.

2810	Reference position pitch error compensation number for each axis	

[Data type] Integral axis
[Data range] 0～1023

> **Note**
> 1. When this parameter is set, the power must be turned off before operation is continued.

2811	The farthest pitch error compensation point number for each axis in negative direction	

[Data type] Integral axis
[Data range] 0～1023

> **Note**
> 1. When this parameter is set, the power must be turned off before operation is continued.

2812	The farthest pitch error compensation point number for each axis in positive direction	

[Data type] Integral axis
[Data range] 0～1023

> **Note**
> 1. When this parameter is set, the power must be turned off before operation is continued.
> 2. This parameter setting value is more than the No.2810 (reference position pitch error compensation number).

2813	Pitch error compensation override for each axis	

[Data type] Real number axis
[Data unit] %
[Data range] 1～100

> **Note**
> 1. When this parameter is set, the power must be turned off before operation is continued.

2814	The interval of pitch error compensation point for each axis	

[Data type] Real number axis
[Data unit] mm
[Data range] 0～9999.9999

> **Note**
> 1. Pitch error compensation point is distributed in equidistant, the least value of interval = the maximum feedrate/7500 * compensation override
> 2. When this parameter is set, the power must be turned off before operation is continued.

4.14 Servo Parameter (4000～4999)

	7#	6#	5#	4#	3#	2#	1#	0#
4000		ALMG		IGN5	IGN4	IGNZ	IGNY	IGNX

[Data type] Bit
[Data range] 0 or 1

IGNn: The servo axis is:

0: Not ignored

1: Ignored

ALMG: The corresponding switch for system alarms to servo

0: Not alarmed

1: Alarmed

[**Standard setting]** 0 0 0 0 0 0 0 0

	7#	6#	5#	4#	3#	2#	1#	0#
4001	RAST				APZ			

[Data type] Bit axis

[Data range] 0 or 1

APZ: Absolute encoder position and machine position

0: Not corresponding

1: Corresponding

RST: Whether to use the optical grating

0: No

1: Yes

[**Standard setting]** 0 0 0 0 1 0 0 0

4100	Move axis pulse equivalent	

[Data type] Real number axis

[Data unit] Pulse/mm

[Data range]1～99999999

> **Note**
> 1. This parameter is valid when the pulse drive servo is used.

4111	The maximum allowable position offset when each axis is stopped	30

[Data type] Real number axis

[Data unit] um

[Data range] 0～32767

4120	Grid or reference point offset value for each axis	

[Data type] Real number axis

[Data range] 0～99999999

4121	Backlash compensating value for each axis	1

[Data type] Real number axis

[Data unit] mm

[Data range] -9999.9999～9999.9999

4122	Backlash compensating value in rapid traverse rate for each axis	1

[Data type] Real number axis
[Data unit] mm
[Data range] -9999.9999～9999.9999

4123	Step distance of backlash compensation	1

[Data type] Real number axis
[Data unit] mm
[Data range] 0～9999.9999

4200	Password	315

[Data type] Integral number axis
[Data range] 0～9999

4201	Motor type code	65

[Data type] Integral number axis
[Data range] 0～100

4202	Version number	

[Data type] Integral number axis

4203	Initial display state	0

[Data type] Integral number axis

4204	Control mode selection	0

[Data type] Integral number axis
[Data range] 0～7

4205	Speed proportional gain	135

[Data type] Integral number axis
[Data range] 5～1280

4206	Speed integration time constant (ms)	80

[Data type] Integral number axis
[Data unit] ms
[Data range] 0～32767

4207	Torque command filter (%)	50

[Data type] Integral number axis
[Data range] 1~2000

4208	Speed check low-pass filter (%)	110

[Data type] Integral number axis
[Data range] 40~2000

4209	Position proportional gain	245

[Data type] Integral number axis
[Data range] 0~2000

4210	Position feedforward gain (%)	0

[Data type] Integral number axis
[Data range] 0~1280

4211	Position feedforward low-pass cut-off frequency (Hz)	300

[Data type] Integral number axis
[Data unit] Hz
[Data range] 2000

4212	Electron gear rate numerator	8192

[Data type] Integral number axis
[Data range] 0~32767

4213	Electron gear rate denominator	5000

[Data type] Integral number axis
[Data range] 0~32767

4214	Position pulse input mode	0

[Data type] Integral number axis
[Data range] 0~2

4215	Position pulse direction	0

[Data type] Integral number axis
[Data range] 0~1

4216	The completion range of positioning	20

[Data type] Integral number axis
[Data range] 0~32767

4217	Position excess-error check range	30000

[Data type] Integral number axis
[Data range] 0~10000000

4218	Position excess-error check is enabled	0

[Data type] Integral number axis
[Data range] 0~1

4219	Differential proportional coefficient	0

[Data type] Integral number axis
[Data range] 0~127

4220	Invalid drive forbiddance	0

[Data type] Integral number axis
[Data range] 0~1

4221	JOG speed	100

[Data type] Integral number axis
[Data range] -6000~6000

4223	The maximum speed limit	2500

[Data type] Integral number axis
[Data range] 0~6000

4224	Internal speed 1	0

[Data type] Integral number axis
[Data range] -6000~6000

4225	Internal speed 2	100

[Data type] Integral number axis
[Data range] -6000~6000

4226	Internal speed 3	300

[Data type] Integral number axis
[Data range] -6000~6000

4227	Internal speed 4	-100

[Data type] Integral number axis
[Data range] -6000~6000

4228	Arrvial speed	500

[Data type] Integral number axis
[Data range] -6000~6000

4229	The 2nd integration time constant of speed	220

[Data type] Integral number axis
[Data range] 0~32767

4230	Linear speed conversion numerator	10

[Data type] Integral number axis
[Data range] 0~32767

4231	Linear speed conversion denominator	1

[Data type] Integral number axis
[Data range] 0~32767

4232	Linear speed decimal point position	3

[Data type] Integral number axis
[Data range] 0~4

4233	The speed in the mode of motor check	100

[Data type] Integral number axis
[Data range] 0~6000

4239	Acceleration time ms	0

[Data type] Integral number axis
[Data unit] ms
[Data range] 0~10000

4241	Servo output pulse number	0

[Data type] Integral number axis
[Data unit]
[Data range] 0~32767

4246	Analog command and pulse output are reverse	0

[Data type] Integral number axis
[Data range] 0~3

4256	The output time is performed in advance when the feedback pulse is more than 10000	20

[Data type] Integral number axis
[Data unit] ms
[Data range] 0~32767

4257	Speed command feedforward gain	200

[Data type] Integral number axis
[Data range] 0~32767

4258	Acceleration command feedforward gain	200

[Data type] Integral number axis
[Data range] 0~32767

4259	Inertia stop decay coefficient	2

[Data type] Integral number axis
[Data range] 1~4

4260	Current proportional gain	1050

[Data type] Integral number axis
[Data range] 0~12800

4261	Current integration time constant	130

[Data type] Integral number axis
[Data unit] ms
[Data range] 0~32767

4262	Current integration separation point	20

[Data type] Integral number axis
[Data range] 0~32767

4264	Current low-pass cut-off frequency	280

[Data type] Integral number axis
[Data unit] Hz
[Data range] 0~32767

4265	Speed integration separation point	200

[Data type] Integral number axis
[Data range] 0~32767

4267	Position excess-error corner 0	20

[Data type] Integral number axis
[Data range] 0~32767

4268	Position excess-error corner 1	350

[Data type] Integral number axis
[Data range] 0~32767

4269	The position proportional gain change rate of position excess-error corner 0	100

[Data type] Integral number axis
[Data range] 0~32767

4270	The position proportional gain change rate of position excess-error corner 1	100

[Data type] Integral number axis
[Data range] 0~32767

4271	Motor rotor inertia	133

[Data type] Integral number axis
[Data range] 0~32767

4272	Motor rated torque	60

[Data type] Integral number axis
[Data range] 0~32767

4273	Motor rated speed	2500

[Data type] Integral number axis
[Data range] 0~32767

4274	The maximum allowable current of current sampling circuit	2500

[Data type] Integral number axis
[Data range] 0~32767

4275	Motor rated current	60

[Data type] Integral number axis
[Data range] 0~32767

4276	The maximum overload capability	200

[Data type] Integral number axis
[Data range] 0~32767

4279	Differential regulation time	200

[Data type] Integral number axis
[Data range] 0~32767

4280	The 2nd integration time constant of current	500

[Data type] Integral number axis
[Data range] 0~32767

4288	The communication error counter of absolute encoder	2

[Data type] Integral number axis
[Data range] 0~32767

4289	The longest time executes when the encoder is set to 0.	3000

[Data type] Integral number axis
[Data range] 0~32767

4290	Zero point offset of encoder	0

[Data type] Integral number axis
[Data range] 0~32767

4294	Wheel display time in alarm	15000

[Data type] Integral number axis
[Data range] 0~32767

4295	Magnetic polar number	4

[Data type] Integral number axis
[Data range] 0~32767

4296	Opened-loop operation voltage	1395

[Data type] Integral number axis
[Data range] 0~32767

4297	Opened-loop operation speed	16

[Data type] Integral number axis
[Data range] 0~32767

4298	SON enforcement is valid	0

[Data type] Integral number axis
[Data range] 0~1

4299	The voltage value when the encoder is set to 0	4000

[Data type] Integral number axis
[Data range] 0~32767

4300	Feedback speed gain	1000

[Data type] Integral number axis
[Data range] 0~32767

4302	The adjustable pulse width of pulse Z output	30

[Data type] Integral number axis
[Data range] 0~32767

4304	Speed filter cut-off frequency 0	200

[Data type] Integral number axis
[Data range] 0~32767

4305	Speed filter cut-off frequency 1	230

[Data type] Integral number axis
[Data range] 0~32767

4306	Speed filter cut-off frequency 2	250

[Data type] Integral number axis
[Data range] 0~32767

4307	Speed filter cut-off frequency 3	280

[Data type] Integral number axis
[Data range] 0~32767

4309	Speed filter wave separation point 0	3

[Data type] Integral number axis
[Data range] 0~32767

4310	Speed filter wave separation point 1	10

[Data type] Integral number axis
[Data range] 0~32767

4311	Speed filter wave separation point 2	35

[Data type] Integral number axis
[Data range] 0~32767

4312	The check is permited when the encoder command return is incorrect	0

[Data type] Integral number axis
[Data range] 0~1

4313	Inertial stop selection	1

[Data type] Integral number axis
[Data range] 0~1

4314	Over-current alarm delay time	1

[Data type] Integral number axis
[Data range] 0~20

4316	3-second key time	800

[Data type] Integral number axis
[Data range] 400~32767

4317	Open strobe delay time	1500

[Data type] Integral number axis
[Data range] 0~32767

4318	Brake delay time	10000

[Data type] Integral number axis
[Data range] 0~32767

4320	Speed display error compensation	4

[Data type] Integral number axis
[Data range] 0~32767

4.15 Spindle Control Parameter (5000～5999)

	7#	6#	5#	4#	3#	2#	1#	0#
5000	LOOPS				SVAL	ALMS	SWG	SAR

[Data type] Bit
[Data range] 0 or 1

SAR: The spindle speed arrvial signal is:
0: Not checked
1: Checked

SWG: Spindle alarm switch
0: Ignored
1: Accepted and treated

ALMS: Spindle alarm LEVEL availability
0: Low LEVEL
1: High LEVEL

SVAL: Spindle speed display selection
0: Command speed
1: Actual speed

LOOPS: Spindle position control mode selection
0: Opened-loop
1: Closed-loop

5100	The gain regulation data of spindle speed analog output	

[Data type] Integrated
[Data range] 0～1250

5101	The compensation value of spindle speed analog output offset voltage	0

[Data type] Integrated
[Data range] -1024 ～ 1024

5102	Spindle acceleration	2222

[Data type] Real number
[Data unit] rev / s^2
[Data range] 0～99999

5103	Spindle analog output direction	0

[Data type] Integrated
[Data range] 0～1(0: Positive, 1: Negative)

5105	The spindle maximum acceleration in rigid tapping	139

[Data type] Real number
[Data unit] rev / s^2
[Data range] 0～99999

5106	The direction control of closed-loop spindle	0

[Data type] Integrated
[Data range] 0～3

Setting value	Command direction	Feedback direction
0	1	1
1	-1	-1
2	1	-1
3	-1	1

5110	The motor's speed when the spindle orientation or spindle gear shifting are performed	100

[Data type] Integrated
[Data unit] r/min
[Data range] 0～100000

5111	The lowest speed constant of spindle motor	0

[Data type] Integral word
[Data unit] r/min
[Data range] 0～100000

5112	The highest speed constant of spindle motor	6000

[Data type] Integrated
[Data unit] r/min
[Data range] 0～100000

5113	Time check of spindle speed arrival signal	

[Data type] Integrated
[Data unit] ms
[Data range] 0～255

5120	The spindle maximum speed of gear 1	1500

[Data type] Real number
[Data unit] r/min
[Data range] 0～100000

5121	The spindle maximum speed of gear 2	3000

[Data type] Real number
[Data unit] r/min
[Data range] 0～100000

5122	The spindle maximum speed of gear 3	4500

[Data type] Real number
[Data unit] r/min
[Data range] 0～100000

5130	Spindle speed shift point between gear 1- and gear 2	2000

[Data type] Real number
[Data unit] r/min
[Data range] 0～4095

5131	Spindle speed shift point between gear 2- and gear 3	3500

[Data type] Real number
[Data unit] r/min
[Data range] 0～4095

5135	Spindle speed shift point between 1- and gear 2 when the tapping cycle is performed.	1500

[Data type] Real number
[Data unit] r/min
[Data range] 0～100000

5136	Spindle speed shift point between 2- and gear 3 when the tapping cycle is performed.	2000

[Data type] Real number
[Data unit] r/min
[Data range] 0～100000

5139	Spindle default speed	500

[Data type] Integrated
[Data unit] r/min
[Data range] 0～100000

5142	The maximum spindle speed	10000

[Data type] Integrated
[Data unit] r/min
[Data range] 0～100000

5143	Spindle servo loop circuit proportional gain	0.5

[Data type] Real number
[Data range] 0～9999

5144	Spindle servo loop circuit integration gain	0.0005

[Data type] Real number
[Data range] 0～9999

5160	Spindle low gear rate numerator	1

[Data type] Integral number
[Data range] 1～999999

5161	Spindle low gear rate denominator	1

[Data type] Integral number
[Data range] 1～999999

5162	Spindle middle gear rate numerator	1

[Data type] Integral number
[Data range] 1～999999

5163	Spindle middle gear rate denominator	1

[Data type] Integral number
[Data range] 1～999999

5164	Spindle high gear rate numerator	1

[Data type] Integral number
[Data range] 1～999999

5165	Spindle high gear rate denominator	1

[Data type] Integral number
[Data range] 1～999999

5170	The maximum spindle excess-error range	100000

[Data type] Integrated
[Data range] 1～1000000

5171	Spindle pulse equivalence	4096

[Data type] Real number
[Data unit] Pulse number
[Data range] 1～32767

> **Note**
> It is valid when the servo spindle is valid.

4.16 Custom Macro Program Parameter (6000～6999)

	7#	6#	5#	4#	3#	2#	1#	0#
6001						CCV	CLV	TCS

[Data type] Bit
[Data format]

TCS Subprogram
0: Not called using a T code
1: Called using a T code

CLV Custom macro’s local variable Nos. 1 to 33
0: Cleared to “vacant” by reset
1: Not cleared by reset

CCV Custom macro’s common variables No.s 100 to 199
0: Cleared to “vacant” by reset
1: Not cleared by reset

6050	G code that calls the subprogram number O9010	

6051	G code that calls the subprogram number O9011	

6052	G code that calls the subprogram number O9012	

6053	G code that calls the subprogram number O9013	

6054	G code that calls the subprogram number O9014	

6055	G code that calls the subprogram number O9015	

6056	G code that calls the subprogram number O9016	

6057	G code that calls the subprogram number O9017	

6058	G code that calls the subprogram number O9018	

6059	G code that calls the subprogram number O9019	

[Data type] Integerated
[Data range] 1~9999

These parameters set the G codes that call the custom macros of program numbers 9010 through 9019.

Note
These parameters set the G codes that call the custom macros of program numbers 9010 through 9019.

6071	M code that calls the subprogram number O9001	

6072	M code that calls the subprogram number O9002	

6073	M code that calls the subprogram number O9003	

6074	M code that calls the subprogram number O9004	

6075	M code that calls the subprogram number O9005	

6076	M code that calls the subprogram number O9006	

6077	M code that calls the subprogram number O9007	

6078	M code that calls the subprogram number O9008	

6079	M code that calls the subprogram number O9009	

[Data type] Integerated
[Data range] 1~9999

These parameters set the M codes that call the custom macros of program numbers 9001 through 9009.

Note
Setting value 0 is invalid. No subprogram can be called by M00.

6080	M code that calls the subprogram number O9020	

6081	M code that calls the subprogram number O9021	

6082	M code that calls the subprogram number O9022	

6083	M code that calls the subprogram number O9023	

6084	M code that calls the subprogram number O9024	

6085	M code that calls the subprogram number O9025	

6086	M code thatcalls the subprogram number O9026	

6087	M code that calls the subprogram number O9027	

6088	M code that calls the subprogram number O9028	

6089	M code that calls the subprogram number O9029	

[Data type] Integerated
[Data unit] 1~9999

Custom macro program M code is called the program numbers from 9020 to 9029.

> **Note**
> Setting value 0 is invalid. No custom macro program can be called by M00.

4.17 System Diagnosis Configuration Parameter (9000～9999)

9101	System control axis number selection	3

[Data type] Integerated
[Data range] 2～5

9120	Screen-protection time waiting	50

[Data type] Integerated
[Data unit] min.
[Data range] 0～9999

9121	System interpolation period time ms	2

[Data type] Real number
[Data unit] ms
[Data range] 0.01～9999.9999

> **Note**
> When this parameter is set, the power must be turned off before operation is continued.