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

This User Manual is suitable for the GSK CNC EQUIPMENT CO., LTD.

The following CNC System are manufactured by GSK

Series	Product Type	Structure	LCD Dimension	Remark
GSK218MC	GSK218MC	Integral	10.4	LCD dimension is default as 10.4 inch
	GSK218MC-U1	Integral	8.4	LCD dimension is default as 8.4 inch
	GSK218MC-H	Horizontal	8.4	LCD dimension is default as 8.4 inch
	GSK218MC-H2	Horizontal	10.4	LCD dimension is default as 10.4 inch
	GSK218MC-V	Horizontal	10.4	LCD dimension is default as 10.4 inch

Wherein, GSK218MC, GSK218MC-H and GSK218MC-V are provided with three kinds of communication interfaces, such as the RS232, USB and network, which are set at the front face of the host.

GSK218MC-U1 owns only two communication interfaces, such as the RS232 and USB, USB interface sets on the front face of the host, and RS232 sets at the back of the host.

PREFACE

Dear Users,

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

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

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

SECURITY WARNING

⚠ Improper operations may cause unexpected accidents. Only the person who owns the qualified professional can operate this system.

Special prompt: The power supply fixed on/in the cabinet is exclusively used for the CNC system made by GSK.

Never attempt to use this power as other purposes. Otherwise, the enormous hazard may occur!

SAFETY PRECAUTIONS

■ Transportation and storage

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

■ Open-package inspection

- Confirm the product is the one you purchased after opening the package.
- Check whether the product is damaged during transportation.
- Confirm all the elements are complete without damage by referring to the list.
- **It is necessary to contact our company immediately if the product type is inconsistent with the packing list, lack of accessories or damage in transportation.**

■ Wiring

- Only the person who executes the wiring and inspection should have the corresponding professional capacity.
- The product should be reliably grounded, and its resistance should be less than 0.1Ω and can not be used the neutral conductor (zero cable) to replace the ground wire.
- The connection must be correct and secured. Otherwise, the product may be damaged or unexpected results may occur.
- The surge absorb diode connected with the product should be linked based upon the described direction, otherwise, it may damage the product.
- Turn off the power before inserting or unplugging a plug, or opening the electric cabinet.

■ Overhaul

- Turn off the power supply before troubleshooting or replacing components.

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

DECLARTION

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

NOTICE

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

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

BOOK Ⅰ PROGRAMMING

This part introduces the technolog specification, production types, command code and program format of the GSK218MC.

BOOK Ⅱ OPERATION

This part introduces to the operation of the machining center CNC system of GSK 218MC series.

APPENDIX

This part introduces the factory standard parameter and alarm list, etc. of the machining center CNC system of the GSK218MC series

SECURITY RESPONSIBILITY

Security responsibility of the manufacturer

—— Manufacturer should take responsibility for the design and structure danger of the CNC system and the accessories which have been eliminated and/or controlled.

—— Manufacturer should take responsibility for the security of the CNC system and accessories

—— Manufacturer should be take responsibility for the offered information and suggestions for the user.

Security responsibility of the user

—— User should know and understand about the contents of security operation by learning and training the CNC system safety operations.

—— User should take responsibility for the security and danger because of increasing, changing or modifying the original CNC system and accessories by themselves.

—— User should take responsibility for the danger without following the operations, maintenance, installations and storages described in the manual.

This manual is reserved by final user.

Sincere thanks for your support of GSK's product!

Chinese version of all technical documents in Chinese and English languages is regarded as final.

LIST

VOLUME I PROGRAMMING

VOLUME I
PROGRAMMING

CHAPTER ONE SUMMARY

1.1 Product Brief

GSK218MC Series Machining Center CNC System is a kind of upgraded product of GSK218M, which uses the high-velocity spline interpolation calculation; the machining velocity, accuracy and the fineness of the surface are greatly enhanced accordingly; the newly designed man-machine interface is more beatiful, friendly and useful, which supports the GSK-LINK Ethernet bus function, and the connection is more convenient; as well, it supplies the statement macro program (Macro B), so that the programming is briefer. It can be adapted with the Milling Machine Center, High velocity CNC engraving and milling machine, grinding machine and hobbing machine, etc.

- High velocity & high accuracy, complicated curve surface machining efficient velocity 8m/min, optimal machining velocity 4m/min.
- Top position velocity 60m/min, the Max. feedrate150m/min.
- Up to 1000 for the pretreatment sections, it owns the prospect function, high velocity, high accuracy and good smoothness.
- The installation structure divides into integral, horizontal and vertical, which is

separately used the 8.4/10.4 inch high resolution color LCD.

- the newly designed man-machine interface is more beatiful, friendly and useful.
- It supports the Chinese, English, Russian, Spanish and Turkey, etc.
- It supports the functions, such as the PLC on-line monitoring, editing, compiling and signal trace.
- It supports many kinds of tool magazines, for example, the turntable, disk and servo one.
- It supports the statement macro program (Macro B)
- Abundant help, prompt information; and easy to learn, use and debugging.
- It spports the RS232, USB and network 3-kind communication interfaces, which carries out the file transmission, DNC machining, USB on-line machining.
- It supports the GSK-LINK Ethernet bus function for convenient connection and strong extension, and supports the 17-bit absolute encoder with high accuracy, against to zero return, which can be carried out the full-closed-loop control (Optional).

1.2 Technology Specification

Movement control	Controllable axis and linkage axis: the standard configuration is 4-axis 3-linkage; the 4-axis 4-linkage can be matched. Each axis can be regarded as linear axis or rotation one.
	Interpolation method: Positioning (G00), straight line (G01), circular arc (G02, G03) and helix interpolation.
	Position command range: Metric: -99999.999mm～99999.999mm, the least command unit: 0.001mm; Inch: -9999.9999inch～9999.9999inch, the least command unit 0.0001inch;
	Electric gear: Command multiplication coefficient 1～65536, command frequency-devision coefficient 1～65536.
	Rapid traverse rate: Up to 60m/min, rapid override: F0, 25％, 50％ and 100％ 4-level real-time adjustment.
	Cutting feedrate: Up to 15m/min (G94) or 500.00mm/r (G95) Feedrate: 0～200％ 21-level real-time adjustment, wave-section control available;

	MPG feed: 3-gear (0.001, 0.01 and 0.1mm); single feed: 4-gear (0.001, 0.01, 0.1 and 1mm)
Acceleration/ deceleration	Forward acceleration: The linear acceleration/deceleration or S-type one can be selected; the acceleration/deceleration time constant can be set.
	Backward deceleration: The linear acceleration/deceleration or index one can be alternative; the acceleration/deceleration time constant can be set.
	Wherein, Manual, MPG and single step are used the backward deceleration. The rapid positioning, cutting feed can be selected the forward/backward acceleration or deceleration.
Miscellaneuos function Tool function	It is specified by M and 2-digit; M function can be self-defined.
	System internal M command (do not repeatedly define): End-of-program M02, M30; Program stop M00; Optional stop M01, Tool-magazine call M06, Subprogram call M98 and Subprogram end M99;
	The defined M commands by PLC: M03, M04, M05, M08, M09, M10, M11, M16, M17、M18, M19, M20, M21, M22, M23, M2, M26, M27, M28, M29, M35, M36, M44, M45, M50 and M51
Tool function	● T and 4-digit tool selection ●256 groups tool offset value ● length compensation ● worn compensation ●C-type Cutter comensation
Spindle function	● S2 digit (I/O gear control) /S5 digit (Analog output) ● Top spindle velocity limit ● Constant line velocity function;
	Spindle encoder: The encoder linear number can be set to (100～5000p/r); the driving ratio between encoder and spindle: (1～255) : (1～255);
	Spindle override: 50%~120% 8-level real-time adjustment in total, which can be performed the wave-band control.
	Tapping cycle, flexible tapping and rigid tapping
Automatic compensation	● Pitch error compensation: Compensation interval, compensation origin can be set, compensation value range: -999 ~ +999 pulse equivalent.
	● Reverse interval compensation: The reverse interval value can be compensated by setting the fixed frequency or lifting method.
	● Cutter length compensation: A or B type length compensation function can be selected by parameter.
	● Cutter raidus compensation: C-cutter compensation function, the Max. compensation value is ±999.999mm or ±99.9999inch
Reliability and safety	State signal: ●ESP ●Overtravel ●Stored stroke limit ●NC ready signal ●Servo ready signal ●MST function completion signal ●Auto operation start indicator signal ●Auto operation signal during motion ●Feed hold indicator signal
	Self-diagnosis function: ●Signal ●System ●Position control ●Servo ●Communication ●Spindle

	NC alarm: ●Program ●Operation ●Overtravel ●Servo ●Connection ●PLC ●Memory (ROM and RAM)
Operation function	●Edit ●Auto ●MDI ●Zero return ●Manual ●Single step ●DNC ●Single block ●Skip ●Dry run ●M.S.T lock ●Program restart ●MPG interrupted ●Single step interrupted ●Manual interference ●Machine lock ●Interlocking ●Feed hold ●Cycle start ●ESP ●External reset signal ●External power ON/OFF
Display	●GSK 218MC, GSK 218MC-V system uses the color 10.4 inch LCD with the resolution 800×600. ●GSK 218MC-H, GSK 218MC-U1 system uses the color 8.4 inch LCD with the resolution 800×600. ●It provides 5 kinds of interfaces, such as the Chinese, English, Russian, Spanish and Turkey, which can be selected by parameter.
	●Position information ●User program ●System setting ●PLC ●Diagnosis parameter ●Figure ●Alarm information ●Help
	●Actual feedrate, spindle speed ●Real-time wave diagnosis ●System operation time (NC command and state information)
Program edit	Program capacity: 57M, up to 400 programs can be stored.
	●Program preview ●Program edit ●Backstage edit;
PLC Function	PLC treatment velocity: 3us/step; up to 4700 steps; basis command 10 pieces, 35 function commands; ladder diagram on-line edit;
	IO unit input/output: 48/48, extendible;
Communication function	Support the RS-232 series port, USB, network communication interface, file transportation available, DNC machining function (series port, network) and USB on-line machining function
Adapted drive	DA98 Series, GS Series, GE Series Digit AC Servo, etc.

1.3 Product Type Definition

GSK218	M	(C)	(a)	—	(H)	(1)	(1)
1	2	3	4		5	6	

<table>
<tr><th>Series No.</th><th colspan="2">Code Explanation</th></tr>
<tr><td>①</td><td colspan="2">Product type main character part: GSK218 series</td></tr>
<tr><td>②</td><td colspan="2">Function (Machining object) configuration: It expresses by Capital English Letters.
M - Milling machine</td></tr>
<tr><td>③</td><td colspan="2">Series continuation: It is indicated by Capital English Letters. Without: Initial version</td></tr>
<tr><td>④</td><td colspan="2">Sub-series continuation (or improvemen number): It expresses by Lowercase English Letters, such as a, b, c.… Without: Initial version</td></tr>
<tr><td>⑤</td><td>Structure type or dedicated type</td><td>Structure type: It separately expresses by Capital Letters U, H, V and B.
U-Integral, H-Horizontal, V-Vertical, B-Cabinet.
Dedicated type: It expresses by Capital Letter P.</td></tr>
<tr><td>⑥</td><td>LCD dimension (structure) or dedicated code</td><td>LEC dimension: It expresses by one number (1～9).
1: 8.4 inch, 2: 10.4 inch, 3～9: …,
Dedicated type: It expresses by two numbers (01～99).</td></tr>
</table>

Example

- **GSK218MC-H:** 218MC Series, Horizontal structure, 8.4 inch LCD (Defualt dimension)
- **GSK218MC-H2:** 218MC Series, Horizontal structure, 10.4 inch LCD
- **GSK218MC-U1:** 218MC Series, Integral structure, 8.4 inch LCD
- **GSK218MC-P01:** 218MC Series, No.01 dedicated machine
- **GSK218MCa-P25:** 218MCa Series, No.25 dedicated machine

1.4 Bus Function Explanation

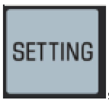
This system adds Ethernet Bus Communication method from the beginning of the system software version V1.4.

The functions described in this manual are suitable for the Bus and Pulse transmission methods. For the former, the new addition function for this system will particularly explain.

Refer to the following description when selecting the Ethernet Bus Communication Method or Pulse Communication Method.

Method one:

1. Enter the <MDI> operation method;

2. Enter the <SET> page by , then enter the password page by [PASSWORD], input the corresponding level password; refer to the Section 3.4.5 Setting and Modification of Password Authorization *Book II OPERATION* in this manual;

3. Set the parameter switch to “1” in the <SETTING> interface;

4. Enter into to the page to perform the seting together by the , then the [⊞ Bus Configuration] (Refer to the Fig. 1-4-1):

1) Move the cursor to the item of the “Bus or not”;

2) Input the “1”, select the Bus; or input the “01”, select the pulse;

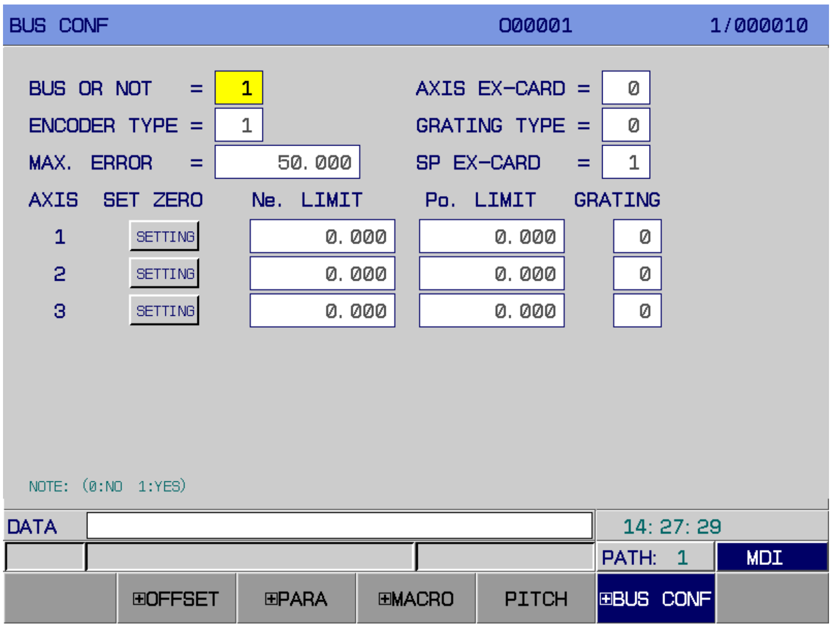


Fig. 1-4-1

Mehtod 2:

The drive transmission method is Bus by directly setting the bit parameter **No: 0#0=1**; it is the pulse by setting the **No: 0#0=0**

Note: It is necessary to cut off the overall powers before modifying this parameter; turn on the power again after performing it.

CHAPTER TWO PROGRAMMING BASIS

2.1 Controllable axis

Table 2-1-1

Item	GSK218MC
Basis controllable axis number	3 axes (X, Y and Z)
Extension controllable axis number (total)	Up to 5 axes

Occasionally, there is not alternative other than to use a additional axis because the structure design requirement for some machine tools, such as the revolving worktable, rotation worktable, etc. This axis can be set as both linear axis and rotation axis. GSK218MC can be set each axis as linear oen or rotation one.

2.2 Axis Name

The names of the 3 basic axes are X, Y, Z and A by default.

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

Note: If the inputted axis name is repeated, the system may automatically initialize as X, Y, Z, A and B.

2.3 Axis Display

When the additional axis is set as rotation one, and the unit of the rotation axis displays as deg; if it is set as linear one, the display is identical with the basis 3 axes (X, Y and Z), and its unit is mm. The following figure shows that the 4th axis is regarded as the linear one, and the 5th is treated as the rotation one.

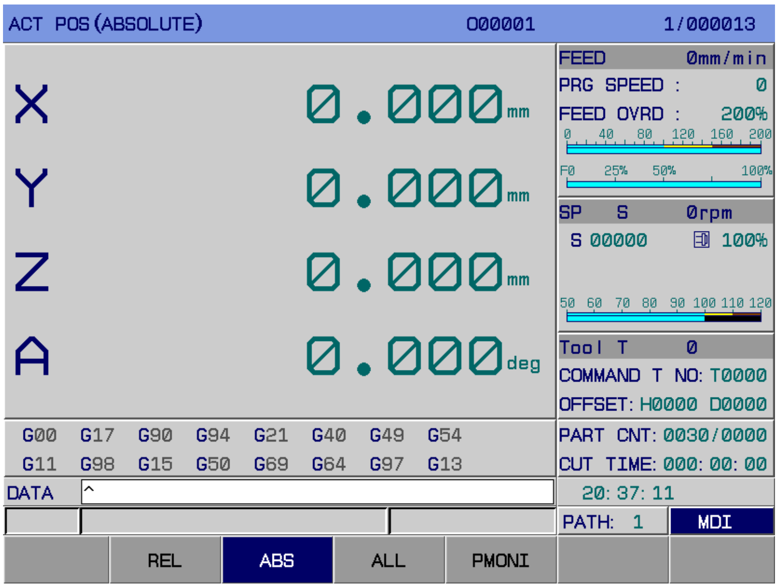


Fig. 2-3-1

2.4 Coordinate System

2.4.1 Machine Tool Coordinate System

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

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

2.4.2 Reference Position

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

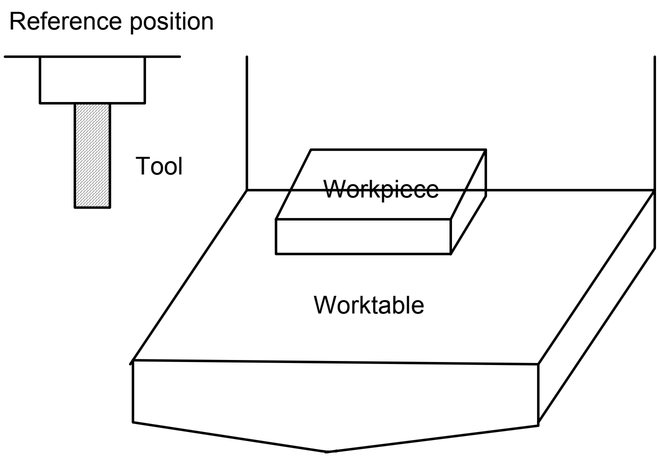


Fig. 2-4-2-1

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

1. Manual reference position return (Refer to the Chapter Nine Zero Operation)

2. Auto reference point return

2.4.3 Workpiece Coordinate System

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

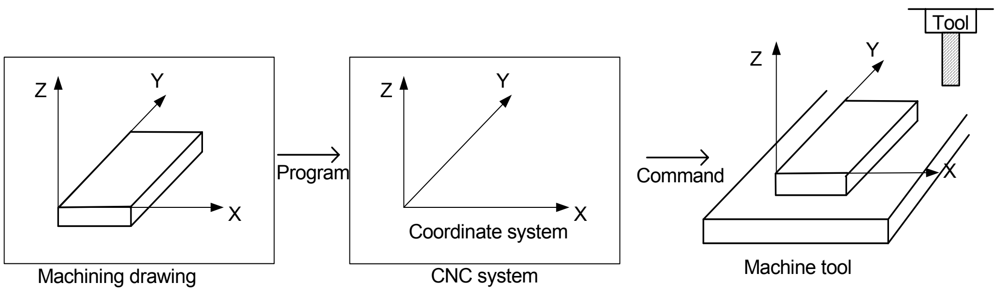


Fig. 2-4-3-1

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

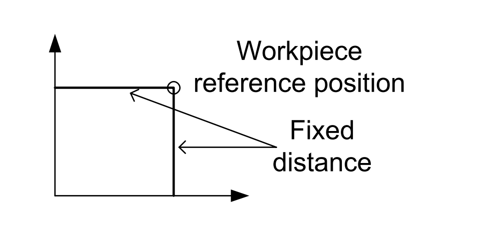
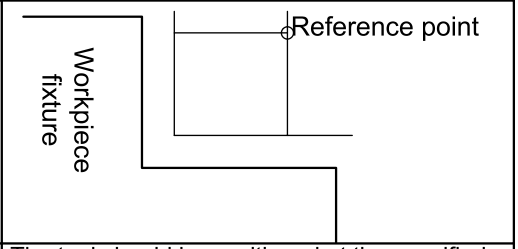
I) The reference point with component	II) Install the component on the fixture directly
	
Tool center movement aims at the component reference position, set the workpiece coordinate system by CNC command at this place; in this case, the workpiece coordinate system is overlapped with the one of the programming.	The tool should be positioned at the specified distance (it can be a reference point) from the reference position, because the tool center can not directly located at the workpiece point. Set the workpiece coordinate system specified by CNC using the specified distance. (For example: G92)

Fig. 2-4-3-2

A machining program sets a workpiece coordinate system (selecting a workpiece coordinate system). The workpiece coordinate system set can be changed by moving its origin.

There are two methods to set the workpiece coordinate system:

1. Using G92, see 4.2.11 for details.
2. Using G54-G59, see 4.2.8 for details.

2.4.4 Absolute Coordinate Programming and Relative Coordinate Programming

There are absolute and relative definitions to define the axis moving amount. The absolute definition is a method to program by the coordinate of the end point of the axis movement, which is called absolute programming. Relative definition is method to program directly by the axis moving amount, which is call relative programming (also called incremental programming).

1) Absolute coordinate value

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

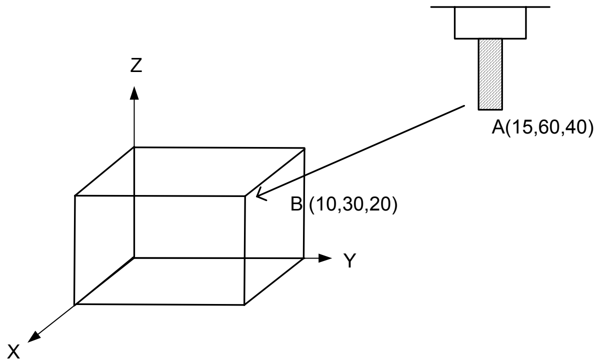


Fig. 2-4-4-1

G90 G54X10 Y30 Z20;

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

2) Incremental coordinate value

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

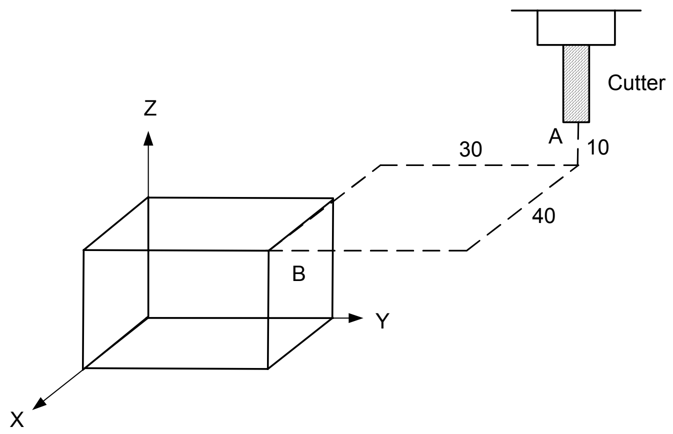


Fig. 2-4-4-2

Tool rapidly moves to the point B from A, its code as follows:

G0 G91 X40 Y-30 Z-10；

2.5 Modal and One-shot

Modal, that of the value of one address, will keep enabling as soon as setting, until this address is reset again. The another meaning of the modal is that this field may not input if the same function is used in the following block after one function word is set.

- For example:

G0 X100 Y100; (Rapid positioning to X100 Y100)

X20 Y30； (Rapid positioning to X20 Y30，G0 is modal, which can be ignored.)

G1 X50 Y50 F300; (Linear interpolation to X50 Y50，feedrate 300mm/min G0→G1)

X100； (Linear interpolation to X100 Y50，feedrate 300mm/min，G1, Y50 and F300 are the modal, which can be ignored.)

G0 X0 Y0； (Rapid positioning to X0 Y0)

The initialized state is the default one after the system is turned on. Refer to the table 4-1-2.

- For example:

O00001

X100 Y100；（Rapd positioning to X100 Y100，G0 is the system iniliatization）

G1 X0 Y0 F100；（Linear interpolation to X0 Y0, feed/min., feedrate is 100mm/min）

One-shot, the corresponding address value, is only enabled in the block written this code; if this address value is used again in the next block which should be specified it again. Refer to the G function code in the group 00 in table 4-1-2.

The modal and one-shot descriptions of the function words are refer to the table 2-5-1

Table 2-5-1 Modal and one-shot of function code

Modal	Modal G function	A group of G functions can be cancelled each other. These functions will always enable once they are performed until cancelled by a same group G functions.
	Modal M function	A group of M functions can be cancelled each other, which keeps enabling before canceling by another function in the same group.
One-shot	One-shot G function	It is only enabled in the specified block, which is cancelled when the block ends.
	One-shot M function	It is only enabled in the block of writing this code.

CHAPTER THREE　COMPONENT PROGRAM CONFIGURATION

3.1　Program Composition

A program composes of doubles of blocks; a block consists of words. Each block is divided into by EOB code (ISO is LF; EIA is CR). The character “;” described in this manaual means the code of EOB.

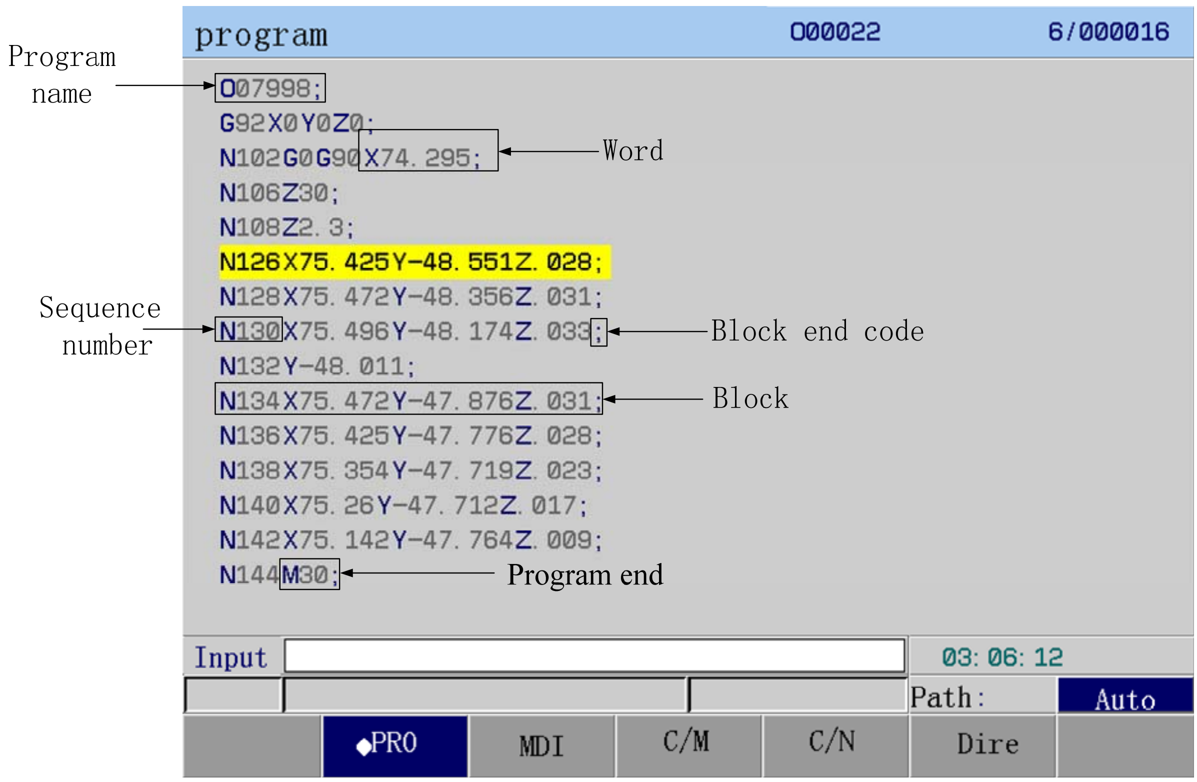


Fig. 3-1-1　Program structure

The code series collection for controlling CNC machine tool to perform the component machining is regarded as program. After the written program is input to the CNC system, the system can be moved the tool along with the straight line, arc or performed the rotation and stop to the spindle based upon the codes. In program, these codes will be compiled according to the actual movement sequence of the machine tool. The program structure is as Fig. 3-1-1.

3.1.1 Program Name

The system memory can be registered several programs. At the beginning of the program, in order to distinguish from these programs, the address O and its following 5-digit is treated as program name, refer to the Fig. 3-1-1-1.

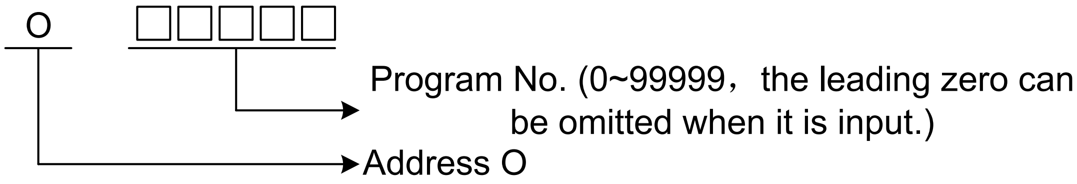


Fig. 3-1-1-1 The composition of program name

3.1.2 Sequency Number and Block

Program consists of multiple codes, one of the code unit is regarded as the block (Refer to the **Fig. 3-1-1**). Program end codes are divided into the blocks (Rerfer to the **Fig. 3-1-1**). The EOB code will show by the character ";" in this manual.

The sequence number (Refer to the Fig. 3-1-1) constituted with the address N and its following 4 numbers can be used at the beginning of the block, the front code leading-zero can be omitted. The order of the sequence number is arbitary (Whether to insert the sequence number is set by bit parameter NO:0 # 5, or directly set in the setting interface, refer to the Section 3.4.1 in the OPERATION); its intervals can be set as in variety (the interval size is determined by parameter P210). Also, the sequence number can be specified for either the overall blocks or the important blocks. Generally, the machining sequence is increasing gradually. It is for the convient that the sequence numbers are specified at the important position (For example, tool-change or the worktable index moves to the new machining interface, etc.)

Note: N code is not regarded as the line number whe it is shared a same block with the G10.

3.1.3 Code Word

Code word (Fig. **3-1-3-1**) is the essential factor for composing the block, which is cosists of the address and its following numbers (sometimes, the +/- symbol may specify in front of the number).

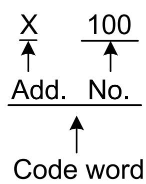


Fig. 3-1-3-1 the composition of the code word

The address is one of the English letters (A~Z), which specifies the meaning of its following numerical value. In this system, the usable address and its meaning, as well the resolution range are as the Fig. **3-1-3-1.**

Occasionally, one address owns different meanings based upon the variable preparation functions.

If two or more same addresses are displayed at a same code, which is determined by bit parameter **No: 32#6** to alarm or not.

Table 3-1-3-1

Add.	Resolution range	Function meaning
A, B, C	It is set by data parameters P175~179	Axle name address
D	0～255	Radius offset number, D0 is regarded as 0 by default; user can neither set nor modify it.
E		Not use yet
F	0.001～99999.999（mm/min）	Feedrate/min.
	0.001～500(mm/r)	Feedrate/rev.
G	00～99	Preparation function
H	01～99	Calculation symbol in G65
	0～255	Length offset number, H0 is treated as 0 by default; User can neither set nor modify it.
I	-99999999～99999999（mm）	The relative start of circular center is at the X axis vector (Circular arc/helix interpolation, zoom in/out)
	I shouls be more than the current tool radius	G22/G23 groove radius inside circle
	(Cutter radius + J) <I≤99999.999mm, it is the absolute value when in the negative.	G24/G25, G26/G32 finish-milling radius of the circle
	I > {(Data parameter P269 setting value * cutter radius) + cutter radius}*2. Cutter radius under the helix < {(I/2) – cutter radius}	Width of G33/G34 rectangle groove along with the X axis direction

Add.	Resolution range	Function meaning
	0<I≤99999.999mm, it is the absolute value when in the negative.	Width of G35/G36, G37/G38 rectangle groove along with the X axis direction
J	-99999999～99999999（mm）	The vectior of the circular arc center relative start along with the Y axis (Arc/helix interpolation and scaling)
	0≤J≤99999.999mm, it is the absolute value when in the negative.	The circle center distance between G24/G25, G26/G32 finish-milling start and the finish-milling circle
	J > {(Data parameter P269 setting value * cutter radius) + cutter radius}*2. Cutter radius under the helix < {(J/2) – cutter radius}	Width of G33/G34 rectangle groove along with the Y axis direction
	0<J≤99999.999mm, it is the absolute value when in the negative	Width of G35/G36, G37/G38 rectangle groove along with the Y axis direction
K	-99999999～99999999（mm）	The vectior of the circular arc center relative start along with the Z axis (Arc/helix interpolation and scaling)
	1～99999	Fixed cycle repeated times
L	1～99999	Repeated calling times of subprogram
	Cutter diameter>L>0	The cutting width increment of the G22/G23 inner circle groove cycle inside the XY plane
	Cutter diameter>L>0, it is the absolute value when in the negative.	The cutting width increment of G33/G34 in the specified plane
	Cutter radius ≤ L ≤99999999mm, it is the absolute value when in the negative.	The distance between the G37/G38 finish-milling start and rectangle edge X axis direction.
M	The setting of data parameter P204	Miscellanous function output, program execution schedule, subprogram call
N	0～99999	Sequence number
	0～999	Parameter sequency number (G10 on-line modification)
O	0～99999	Program name

Add.	Resolution range	Function meaning
P	0～99999.9999（ms）	Dwell time
	1～99999	Subprogram number call
	-9999.9999～9999.9999	Scaling
	Data parameters P281~282	The dwell time at the bottom of the hole in the canned cycle or the time at the point R when retracting.
Q	-99999.999～99999.999（mm）	The cutting detpth or the offset value at the bottom of the hole in canned cycle
R	-99999999～99999999（mm）	Circular arc radius/angle offset value/corner value
	-99999.999～99999.999（mm）	The R plane in the canned cycle
S	The setting of data parameter P205	Spindle rotation specification
	00～04	Multi-gear spindle output
T	The setting of data parameter P206	Tool function
U	The setting of data parameter P175~179	Axle name address
	The resolution range of the U is I/2≥|U|≥D/2, the smaller value in J/2	Corner arc radius in canned cycle
V	The setting of data parameter P175~179	Axle name address
	V>0	The distance from the unprocess plane when cutting at the rapid traverse rate
W	The setting of data parameter P175~179	Axle name address
	W>0 (It should be exceeded the slot-bottom position at the first, and then directly machining based upon its bottom	The first cutting distance of Z axis direction from R plane in the canned cycle
X	The setting of data parameter P175~179	Axle name address
	-99999.999～99999.999（mm）	Coordinate address along with X direction
	0～9999.999（S）	Dwell time specification

Add.	Resolution range	Function meaning
Y	The setting of data parameter P175~179	Axle name address
	-99999.999～99999.999（mm）	Coordinate address along with Y direction
Z	The setting of data parameter P175~179	Axle name address
	-99999.999～99999.999（mm）	Coordinate address along with Z direction

It is the overall restriction values of **CNC** equipment in table **3-1-3-1,** regardless of the one of the machine tool. And therefore, it is important to program based upon the comprehension of the programming limit for referring this manual and the one of the machine tool manufacturer.

Note: The length of each code word should be less than 79 characters.

3.2 General Structure of Program

Program divides into Main program and subprogram. Usually, the CNC moves based upon the indication of the main program; if the code for calling the subprogram on the main program, the CNC is then operated according to subprogram; when meeting the main program code return on subprogram, CNC is then returned to the next program for that the main program calls the subprogram block to be consecutively performed. The program motion sequence is as Fig. 3-21.

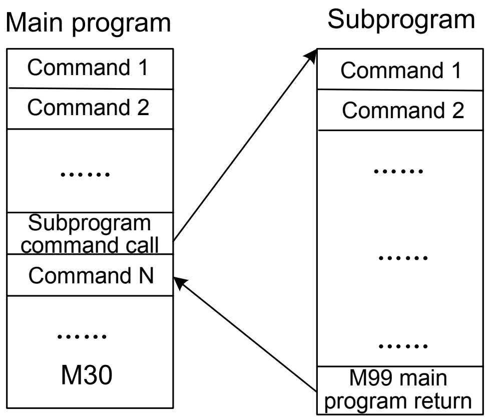


Fig. 3-2-1

The composition structure of main program and subprogram are consistent.

When some one fixed sequence exists and occurs repeatedly in the program, which can be regarded as subprogram, and then save to the register in advance instead of writing again to simplify the program. Subprogram can be called in the Auto mode, generally, the calling can be performed by

M98 among main programs, and also, the called subprogram can be called other subprograms. The called subprogram from the main program is called as the First Subprogram, and there are four subprograms can be called (Refero to the Fig. **3-2-2)**. The last block of the subprogram can be returned to the main program by M99 code, call the next block of the subprogram to be consecutively performed. (If the last block of the subprogram is ended using the M02 or M03 code; the function is same as the M99, return to the main program, and then call the next block of the subprogram to be consecutively performed.)

When the end of the main program is M99, the program is repeatedly performed.

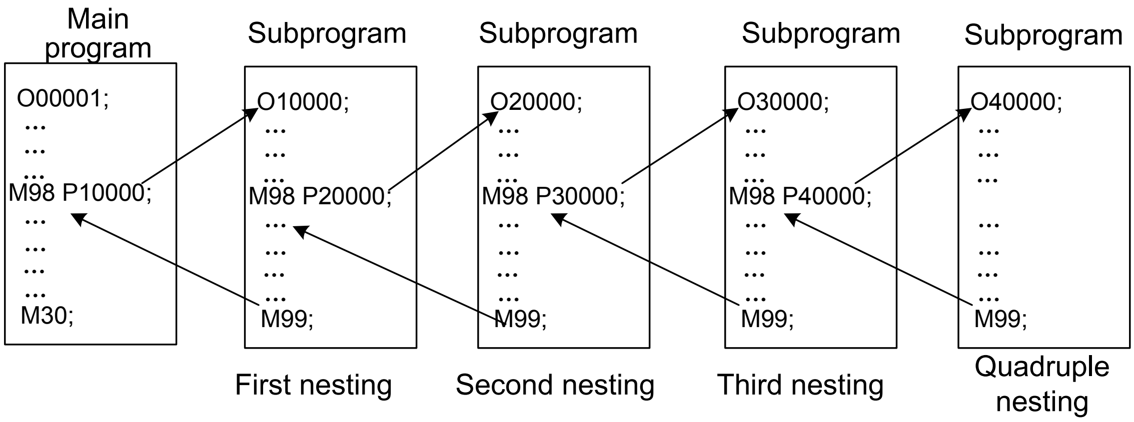


Fig. 3-2-2 Quadruple subprogram nesting

One subprogram calling code can be consecutively and repeatedly called the same subprogram, and up to 9999 times can be repeatedly called.

3.2.1 Subprogram Compiling

Compile a subprogram based upon the following format:

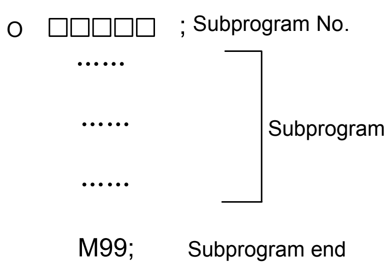


Fig. 3-2-1-1

Write the subprogram No. following the address O at the beginning of the subprogram, the subprogram, at last, is M99 code (M99 compiling formate is as the above-mentioned figure).

VOLUME I PROGRAMMING

3.2.2 Subprogram Calling

The subprogram is performed by main program or calling the code. The subprogram calling code formate is as follows:

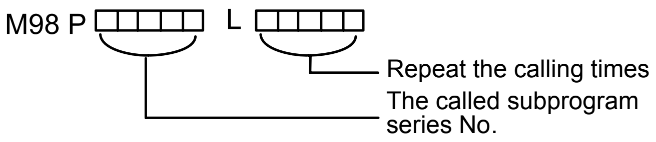


Fig. 3-2-2-1

- If the repeated times are omitted, the repeated times will be regarded as 1.

  (Example) M98 P1002L5 ; (The subprogram No.1002 is consecutively called for 5 times)
- The performance sequence calling from main program to subprogram.

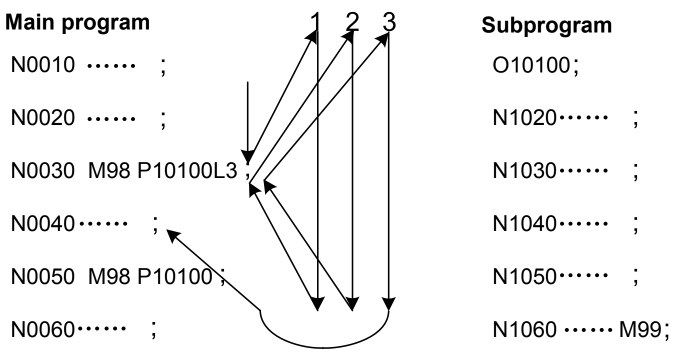


Fig. 3-2-2-2

Calling the subprogram in the subprogram is same as the calling in the main program.

Note 1: When the specified subprogram number by address P can not be indexed; the alarm occurs.

Note 2: The subprogram from No. 90000 to 99999 are the system reserved. The system can be performed the content of the subprogram instead of display the content when user calls these subprograms.

Note 3: Up to 4 layers can be nested for the subprogram calling.

3.2.3 Program End

Program begins from its name which is ended by M02, M30 or M99 (Refer to the **Fig. 3-2-2-2**). If the program end code is detected in the execution of the program: M02, M30 or M99; The program is performed and then becomes to restting state when M02, M30 code ends; Whether the bit 4 of parameter **No.33** for controlling the M30 returns to the beginning of the program; Whether the bit 2 of parameter **No.33** for controlling the M02 returns to the beginning of the program. If M99 code ends, return to the beginning of the program, and then the program is circularly performed; If the M99, M02 and M30 are ended in the subprogram, return to the program of the subprogram calling, and then perform the following blocks continuously.

CHAPTER FOUR PREPARATORY FUNCTION G CODE

4.1 Type of Preparatory Function G Code

The preparatory function is expressed by G code and its following numbers, which specifies the meaning of its block. G codes are divided into the following two types.

Table 4-1-1

Type	Meaning
One-shot code	The G code is enabled only in the block in which it is specified
Modal G code	It is always effective before the other G codes in the same group.

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

G01 X __ ;

Z ______ ; G01 enabled

X ______ ; G01 enabled

G00 Z____ ; G00 enabled

The system bit 7 of parameter No. 0 is common machining mode when it is set to 0. It is the high-velocity high-accuracy machining mode when the bit 7 of No. 0 sets to 1.

Note 1: F: It is indentical with the common machining mode T: high-velocity high-accuracy machining mode

Note 2: Refer to the system parameter table for details.

Table 4-1-2 G code and its function

G code	Group	Code form	High-velocity/ high-accuracy mode enabled or not	Function
*G00	01	G00 X_Y_Z_	T	Positioning (Rapid traverse)
G01		G01 X_Y_Z_F_	T	Linear interpolation (Cutting feed)
G02		G02 X_Y_ R_ F_;	T	Circular arc interpolation CW (Clockwise)

G code	Group	Code form		High-velocity/ high-accuracy mode enabled or not	Function
G03		G03 I_J_		T	Circular arc interpolation CCW (Counter Clockwise)
G04	00	G04 P_ or G04 X_		F	Dwell, Exact stop
G10		G10 L_N_P_R_		F	Programmable data input
*G11		G11		F	Programmable data input mode cancel
*G12	16	G12 X_Y_Z_ I_J_K_		F	Stored stroke check function ON
G13		G13		F	Stored stroke check function OFF
*G15	11	G15		F	Polar coordinate code cancel
G16		G16		F	Polar coordinate code
*G17 G18 G19	02	Write in the block, it is used in the both arc interpolation and cutter compensation		F	XY plane selection ZX plane selection YZ plane selection
G20	06	Specify this command in a single block		F	Data input in inch
G21					Data input in metric
G22	09	G22 X_Y_Z_R_I_L_W_Q_V_D_F_ K		F	Groove rough mill inner circle CCW
G23		G23 X_Y_Z_R_I_L_W_Q_V_D_F_K		F	Groove rough mill inner circle CW
G24		G24 X_Y_Z_R_I_J_D_F_K_		F	Finish mill circular inner the whole-circle CCW
G25		G25 X_Y_Z_R_I_J_D_F_K_		F	Finish mill circular inner the whole-circle CW
G26		G26 X_Y_Z_R_I_J_D_F_K_		F	Finish mill circular of excircle CCW
G27	00	G27	X_Y_Z_	T	Reference position return check
G28		G28		T	Reference position return

G code	Group	Code form	High-velocity/ high-accuracy mode enabled or not	Function
G29		G29	T	Return from the reference position
G30		G30Pn	T	2nd, 3rd and the 4th reference position return
G31		G31	F	Skip function
G32	09	G32 X_Y_Z_R_I_J_D_F_K_	F	Finish mill circular of excircle CW
G33		G33X_Y_Z_R_I_J_L_W_Q_V_U_D_F_K	F	Rectangle groove rough mill CCW
G34		G34X_Y_Z_R_I_J_L_W_Q_V_U_D_F_K	F	Rectangle groove rough mill CW
G35		G35 X_Y_Z_R_I_J_L_ U_D_F_K_	F	Finish mill circular inner the rectangular groove CCW
G36		G36 X_Y_Z_R_I_J_L_ U_D_F_K_	F	Finish mill circular inner the rectangular groove CW
G37		G37 X_Y_Z_R_I_J_L_ U_D_F_K_	F	Finish mill circular out of the rectangle CCW
G38		G38 X_Y_Z_R_I_J_L_ U_D_F_K_	F	Finish mill circular out of the rectangle CW
G39	00	G39	F	Corner offset circular arc interpolation
*G40	07	G17 G40 G41 G42 D_X_Y_	T	Cutter compensation cancel
G41		G18 G40 G41 G42 D_X_Z_	T	Left cutter compensation
G42		G19 G40 G41 G42 D_Y_Z_	T	Right cutter compensation
G43	08	G43 H_Z_	T	Tool length compensation + direction
G44		G44 H_Z_	T	Tool length compensation - direction
*G49		G49 H_Z_	T	Tool length compensation cancel

<table>
<tr><th>G code</th><th>Group</th><th>Code form</th><th>High-velocity/ high-accuracy mode enabled or not</th><th>Function</th></tr>
<tr><td>*G50</td><td rowspan="2">12</td><td>G50</td><td>T</td><td>Scaling cancel</td></tr>
<tr><td>G51</td><td>G51 X_ Y_ Z_ P_</td><td>T</td><td>Scaling</td></tr>
<tr><td>G53</td><td>00</td><td>Write in the program</td><td>T</td><td>Machine tool coordinate system selection</td></tr>
<tr><td>G54</td><td rowspan="6">05</td><td rowspan="6">Write in the block, it generally places at the beginning of the program.</td><td rowspan="6">T</td><td>Workpiece coordinate system 1</td></tr>
<tr><td>G55</td><td>Workpiece coordinate system 2</td></tr>
<tr><td>G56</td><td>Workpiece coordinate system 3</td></tr>
<tr><td>G57</td><td>Workpiece coordinate system 4</td></tr>
<tr><td>G58</td><td>Workpiece coordinate system 5</td></tr>
<tr><td>G59</td><td>Workpiece coordinate system 6</td></tr>
<tr><td>G60</td><td>00/01</td><td>G60 X_ Y_ Z_</td><td>T</td><td>Unidirectional positioning</td></tr>
<tr><td>G61</td><td rowspan="4">14</td><td>G61</td><td>T</td><td>Exact stop mode</td></tr>
<tr><td>G62</td><td>G62</td><td>T</td><td>Automatic corner override</td></tr>
<tr><td>G63</td><td>G63</td><td>T</td><td>Tapping mode</td></tr>
<tr><td>*G64</td><td>G64</td><td>T</td><td>Cutting mode</td></tr>
<tr><td>G65</td><td>00</td><td>G65 H_P# i Q# j R# k</td><td>T</td><td>Macro program code</td></tr>
<tr><td>G68</td><td rowspan="2">13</td><td>G68 X_ Y_ R_</td><td>T</td><td>Coordinate rotation</td></tr>
<tr><td>*G69</td><td>G69</td><td>T</td><td>Coordinate rotation cancel</td></tr>
<tr><td>G73</td><td rowspan="4">09</td><td>G73 X_Y_Z_R_Q_F_;</td><td>F</td><td>High-speed peck machining cycle</td></tr>
<tr><td>G74</td><td>G74 X_Y_Z_R_P_F_;</td><td>F</td><td>Counter tapping cycle</td></tr>
<tr><td>G76</td><td>G76 X_Y_Z_Q_R_P_F_K_;</td><td>F</td><td>Fine boring cycle</td></tr>
<tr><td>*G80</td><td>Write along with other programs in the block</td><td>F</td><td>Canned cycle cancel</td></tr>
</table>

G code	Group	Code form	High-velocity/ high-accuracy mode enabled or not	Function
G81		G81 X_Y_Z_R_F_;	F	Drilling cycle (spot drilling cycle)
G82		G82 X_Y_Z_R_P_F_;	F	Drilling cycle, counter boring cycle
G83		G83 X_Y_Z_R_Q_F_;	F	Peck drilling cycle
G84		G84 X_Y_Z_R_P_F_;	F	Right tapping cycle
G85		G85 X_Y_Z_R_F_;	F	Boring cycle
G86		G86 X_Y_Z_R_F_;	F	Boring cycle
G87		G87 X_Y_Z_R_Q_P_F_;	F	Back boring cycle
G88		G88 X_Y_Z_R_P_F_;	F	Boring cycle
G89		G89 X_Y_Z_R_P_F_;	F	Boring cycle
*G90	03	Write in the block	T	Absolute command
G91				Increment command
G92	00	G92 X_Y_Z_	T	Floating coordinate system setting
*G94	04	G94	T	Feed per minute
G95		G95	T	Feed per rotation
G96	15	G96S_	T	Constant cycle-speed control (Cutting speed)
*G97		G97S_	T	Constant cycle-speed control cancel (Cutting speed)
*G98	10	Write in the block	T	Return to the initial plane in canned cycle
G99				Return to the R point plane in canned cycle

Note 1: If the modal code is shared with a same block with the one-shot modal, and the one-shot modal shoud be considered in priority; simultaneously, change the corresponding mode based upon the other modal codes in a same block, instead of executing them.

Note 2: The system is on G code state of which this G code is with * when the power is turned on. (Partial G codes are determined by bits 0~7 of parameter No.:31)

Note 3: The G codes of the 00 goup are the non-modal G codes other than the G10, G12 and G92.

Note 4: If th G codes without showing in the G code table are used, the alarm then occurs; perhaps, the alarm will also be generated if the G code without the optional function is specified.

Note 5: Several different G codes can be specified in a same block. In principle, two or more G codes in a same group can not be specified in a same block. If a same code that does not alarm in a same block is set, it is subject to the following G codes.

Note 6: The G codes of 01 and 09 groups are shared with a same block. In the canned cycle modal, if the G code in group 01 is specified, the canned cycle will be automatically cancelled and then turn into G80 state.

Note 7: G codes are separately expressed by each group based upon the different types. Whether clear the G codes of each group when resetting or in the ESP, which are set by both the bits 0~7 of parameter No.35 and bits 0~7 of parameter No.36.

Note 8: When the rotation scaling codes and the codes in the 01 or 09 group are shared with a same block, it is better to subject to the rotation scaling code; simultaneously, change the modal of 01 or 09 group. The system will alarm when the rotation scaling code and the code in 00 group are shared with a same block.

4.2 Simple G Code

4.2.1 Rapid Positioning G00

Format: G00 X_Y_Z_

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

Either of the following tool paths can be selected according to bit **1** of parameter **No.12. (**Refer to the **Fig. 4-2-1-1)**

1. **Linear interpolation positioning:** The tool path is identical with 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. **Nonlinear interpolation positioning:** The tool is separately positioned at the rapid traverse rate for each axis. The tool path, generally, is not a straight line.

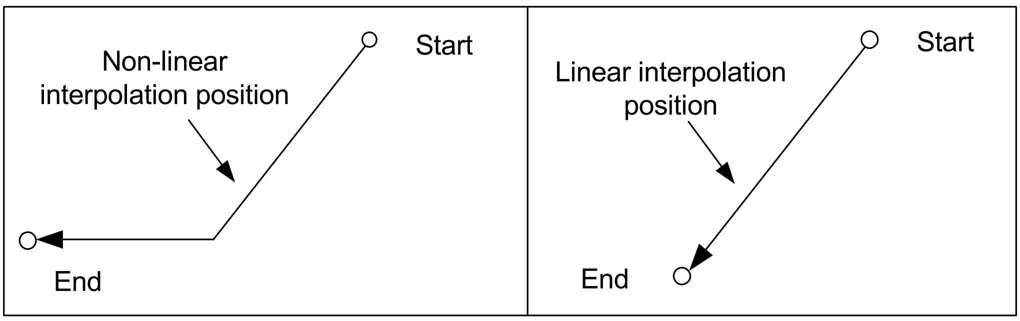


Fig. 4-2-1-1

Explanations:

1. After executing the G00, the modal of current tool movement method turns into G00 by system. Whether the system default mode when the power is turned on is G00 (The parameter value is 0) or G01 (The parameter value is 1) by changing the value of **bit 0 of parameter No.31**.
2. Fail to move the positioning parameter tool without specification. The system is only changed the modal (G00) of the current tool movement mode
3. Both G00 and G0 are the equivalent format.
4. The X, Y, Z and the 4^{th} axis and G0 speed are determined by parameter **P88~P91.**

Restriction:

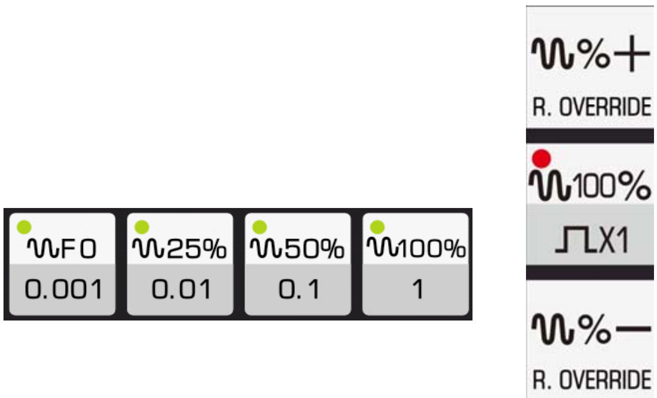
Rapid traverse rate is determined by parameter, for example, set F speed in G0 code, it is the cutting feedrate of the following machining section.

For example:

G0 X0 Y10 F800; Rapid feed using the speed set by system parameter

G1 X20 Y50; Use the feedrate of F800

Rapid positioning speed is adjusted by buttons on the operation panel (Refer to the Fig. **4-2-1-2**), F0, 25%, 50% and 100%; F0 speed is set by data parameter **P93**; general-purpose for each axis.



218MC, 218MC-H, 218MC-V buttons 218MC-U1 button

Fig. 4-2-1-2 The button of rapid feedrate

Note: It is important to note the worktable and workpiece positions when programming to prevent the tool from impacting.

4.2.2 Linear Interpolation G01

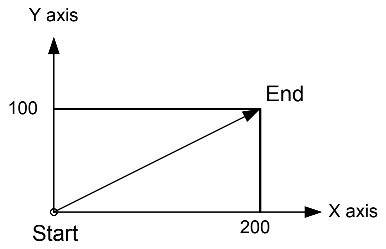
Format: G01 X_ Y_ Z_ F_

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

Explanations:

1. X_ Y_ Z_ is the end coordinate value. It is concerned to the concept of coordinate system, refer to the Sections 2.4.1～2.4.4.
2. The feedrate specified in F is always enabled unitl a new value is specified. The feedrate specified with F code is calculated along with the linear path interpolation. If the F code does not specify in program, the feedrate uses the system ON default F value. (Refer to the data parameter **P87** for the setting).

Program example (Refer to the Fig. **4-2-2-1**)



G01 X200 Y100 F200 ;

Note: The speed of each axis direction:

G01 Xα Yβ ZγFf ;

In this block

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

Feedrate of X axis direction: $F_Y = \frac{\beta}{L} \times f$

Feedrate of X axis direction: $F_Z = \frac{\gamma}{L} \times f$

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

Fig. 4-2-2-1

Notice:

1. The code parameters other than the F are the positioning parameter. Set the upper-limit value of the feedrate F by the data parameter P96. If the actual cutting feed (The feedrate after using the override) exceeds the upper-limit, which is then restricted at the upper-limit. Its unit is mm/min. If the actual cutting feedrate (The feedrate after using the override) is lower than the lower-limit, which is then retricted on the lower-limit, and its unit is mm/min.

2. The tool does not move when the G01 is not specified the positioning parameter. The system is only changed the modal of current tool movement which is the G01. Wheter the system default mode is G00 (When the parameter value is 0) or G01 (When the parameter value is 1) can be set when the power is turned on by changing the value of the bit 0 of the parameter No.:31

4.2.3 Circular Arc (Helical) interpolation G02/G03

A. Circular arc interpolation G02/G03

Both G02 and G03 are specified as follows:

The circular arc interpolation inside the plane is that its path is performed along with the specified rotation and radius (or center) in the specified plane. Thanks to the recognized start and end can not fully affirmed the arc path, and therefore, it is necessary to offer:

- Arc roataion direction (G02, G03)
- The plane of arc interpolation (G17, G18, G19)
- Center coordinate or radius is led out two code formats, center coordinate I, J and K or radius R programming.

The interpolation calculation can be performed in the coordinate system only when the above-mentioned 3 cases are fully affirmed.

The arc interpolation can be performed by following code; and the tool can be operated along with the arc, refer to the following figure:

Arc in the XY plane

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

Arc in the ZX plane

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

Arc in the YZ plane

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

Table 4-2-3-1

Item	Specified content		Command	Description
1	Plane specification		G17	Specification of arc on XY plane
			G18	Specification of arc on ZX plane
			G19	Specification of arc on YZ plane
2	Revolving direction		G02	CW
			G03	CCW
3	End position	G90 method	Two axes of X, Y or Z	End position coordinate in workpiece coordinate system
		G91 method	Two axes of X, Y or Z	The coordinate for the end relative to the start

4	The vector from start to center	Two axes of I, J or K	The position coordinate for the center relative to the start
	Arc radius	R	Arc radius
5	Feedrate	F	Tangent velocity of arc

It is so called the CW and CCW, which is viewed from + to - direction of Z axis (Y axis, X axis) for the XY plane (ZX plane, YZ plane) in the right-hand rectangular coordinate system; refer to the Fig. **4-2-3-1**.

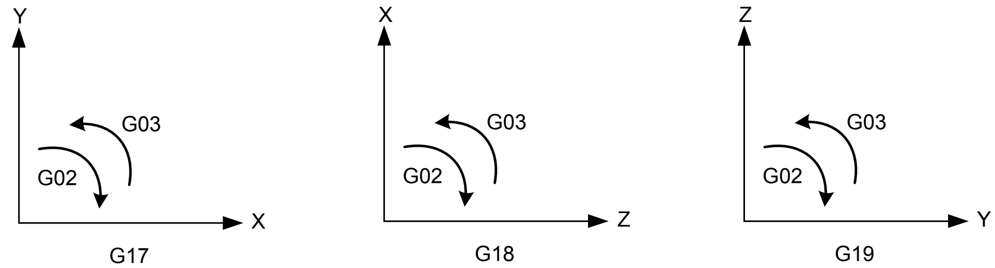


Fig. 4-2-3-1

Setting the bit 1 and 2 of parameter No. 31 can be specified the defaulted plane modal information when starting up.

Specify the arc end by parameter word X, Y or Z. The corresponding G90 command is indicated by absolute value, the G91 by increment value, and the increment value is the coordinate for the end relative to the start. The arc center is specified by parameter word I, J or K, which are separately corresponding to the X, Y or Z. I, J or K parameter value, are the coordinate (Simply, it is temporary to regard the start as the coordinate origin, the coordinate of the center located) for the center relative to the arc start, whenever in the absolute mode G90 or in the relative mode G91, which are the increment value included with the symbol. Refer to the Fig. **4-2-3-2.**

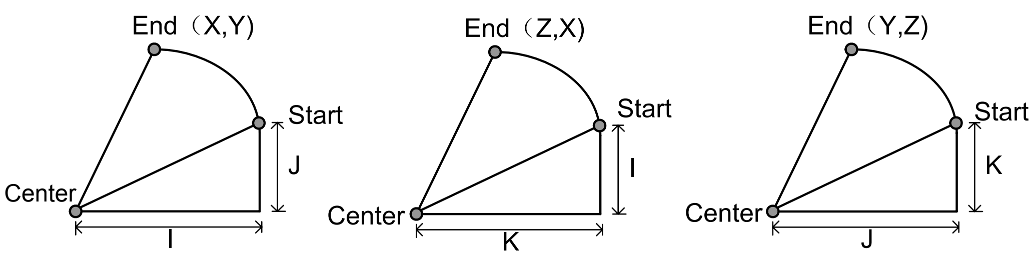


Fig. 4-2-3-2

I, J or K is with the symbol based upon the center relative to the start direction. Arc center also can be specified by radius R other than the I, J or K.

G02 X_ Y_ R_ ；

G03 X_ Y_ R_ ；

1. In this case, the following two arcs can can be drawn, one is the circle more than 180°, the other is the one less than 180°. The arc radius more than 180° is specified by negative value.

(Example: Fig. **4-2-3-3**) When the arc of ① is less than 180°.

G91 G02 X60 Y20 R50 F300 ；

When the arc of ② is more than 180°.

G91 G02 X60 Y20 R-50 F300 ；

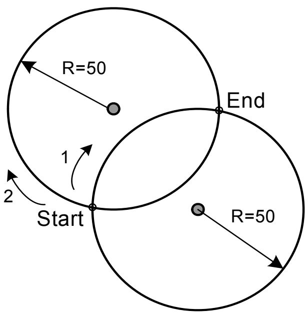


Fig. 4-2-3-3

2. If the arc that equals to 180° can be programmed by both the I, J or K and the R:

Example: G90 G0 X0 Y0；G2 X20 <u>I10</u> F100；

Equal to G90 G0 X0 Y0；G2 X20 <u>R10</u> F100

or G90 G0 X0 Y0；G2 X20 R-10 F100

Notice: The + or – value of the180° arc R is regardless of the motion path of the arc.

3. If the arc that equals to the 360° is only can be programmed by I, J or K.

(Program example):

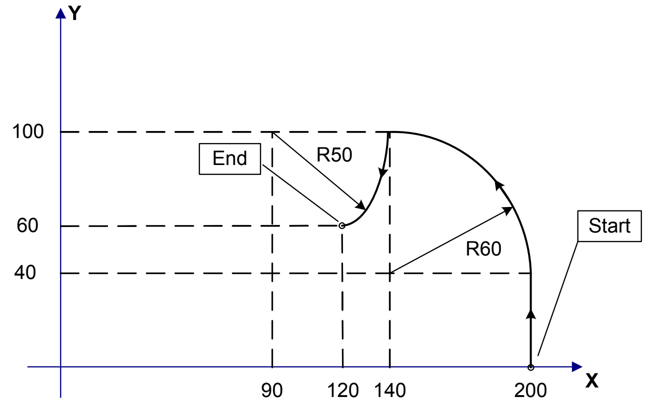


Fig. 4-2-3-4

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

1. In absolute programming

```
G90 G0 X200 Y40 Z0;
G3 X140 Y100 R60 F300;
G2 X120 Y60 R50;
```

Or

```
G0 X200 Y40 Z0;
G90 G3 X140 Y100 I-60 F300;
G2 X120 Y60 I-50;
```

2. In incremental programming

```
G0 G90 X200 Y40 Z0;
G91 G3 X-60 Y60 R60 F3000;
G2 X-20 Y-40 R50;
```

Or

```
G0 G90 X200 Y40 Z0;
G91 G3 X-60 Y60 I-60 F300;
G2 X-20 Y-40 I-50;
```

Restrictions:

1. When the address I, J or K and R are simultaneously specified by program, the arc specified by R is prior, and the others are omitted.

2. If the arc radius parameter does not specify from the start to the arc center, the system will then alarm.

3. If you want to interpolate a whoel circle, only the parameter I, J or K can be specified from the start to arc center, instead of specifying the R.

4. It is note the setting of the coordinate plane when the arc interpolation is performed.

5. Tool is invariables if X, Y and Z are omitted, that is the start and end are shared with a same position, and the R is specified (For example: G02R50;).

B. Helical interpolation

Code format: G02/G03

Circular arc on XY plane

$$G17\begin{Bmatrix}G02\\G03\end{Bmatrix}\quad X_p\_Y_p\_Z_p\_\quad\begin{Bmatrix}I\_J\_\\R\_\end{Bmatrix}\quad F\_$$

Circular arc on ZX plane

$$G18\begin{Bmatrix}G02\\G03\end{Bmatrix}\quad X_p\_Y_p\_Z_p\_\quad\begin{Bmatrix}I\_K\_\\R\_\end{Bmatrix}\quad F\_$$

Circular arc on YZ plane

$$G19\begin{Bmatrix}G02\\G03\end{Bmatrix}\quad X_p\_Y_p\_Z_p\_\quad\begin{Bmatrix}J\_K\_\\R\_\end{Bmatrix}\quad F\_$$

Fig. 4-2-3-5

Function: The tool from the current point moves to the specified position with the helical path based upon the feedrate appointed by parameter F.

Explanation:

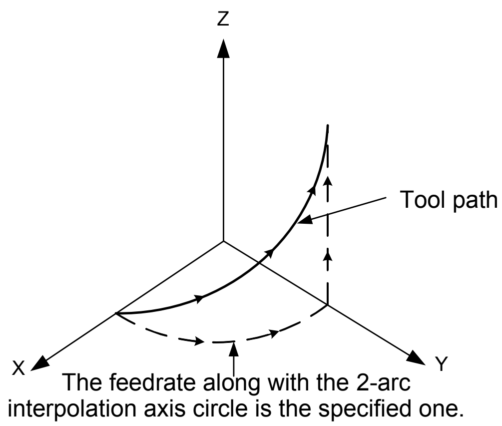


Fig. 4-2-3-6

The previous two-digit of code parameters are positioning parameter. The parameter words are the 2 axes numbers (X, Y or Z) inside the current plane. These two positioning parameter are specified a tool that should be moved to the position inside the current plane. The parameter words of the 3rd code parameter is the linear axis other than the arc interpolation axis. Its parameter value is the helical height. The meanings and restrictions of other code parameters are identical with the circular arc interpolation.

If the system can not machined a circle based upon the offered code parameter, the system is then returned the incorrect information. The modal of the current tool movement method changes to G02/G03 by system after it executes.

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

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

Affirm that the feedrate of linear axis is less than any of the restriction values.

Restriction:

It is note the coordinate plane selection when the helical interpolation is performed.

4.2.4 Absolute/Incremental Programming G90/G91

Format: G90/G91

Function: The code axis movement value method are included the absolute value code and the incremental value code.

The absolute value code is programmed with the coordinate value by the end of axis movement, and the end position is concerned with the concept of the coordinate system; refer to the Sections 2.4.1～2.4.4.

The incremental value code is directly programmed by the relative movement value of the axis. The incremental value is regardless of its coordinate system, it only needs to be offered the movement direction and distance that of the end position relative to the start one.

The absolute value code and the increment one are separately used by G90 and G91.

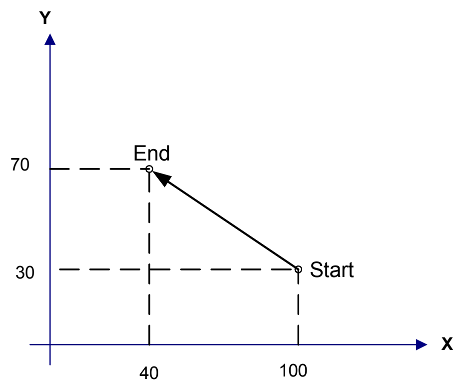


Fig. 4-2-4-1

The movement from start to end in the Fig. **4-2-4-1** is programmed by absolute G90 and increment G91 are shown below:

G90 G0 X40 Y70；

or G91 G0 X－60 Y40 ；

The above-mentioned methods can be performed the same operation. User can use it flexibly based upon the requirements.

Explanations:

- Parameter without code can be written to the block along with other codes.
- G90 and G91 that are the modal values are shared with a same group, that is, the G90 mode is the default before the G91 does not specify; and the G91 is enabled unitl the G90 mode is specified.

System parameter:

Setting the bit 4 of parameter No.31 can be specified whether the default positioning parameter is G90 (When the parameter is 0) or G91 (When the parameter is 1) mode when starting up.

4.2.5 Dwell (G04)

Format: G04 X_ or P_

X, P: Specify a time

Function: G04 executes a dwell, delay performs based upon the specified time and then execute the next block. In addition, in the cutting method, the exact stop detection is performed in the G64, which can be specified the dwell. The feed method G95 per rotation specifies the dwell by the bit 0 of parameter No. 34.

Table 4-2-5-1 Command value range of dwell time (Command by X)

The least movement unit	Command value range	Dwell time unit
No.5#1=0	0.001～9999.999	s or rev
No.5#1=1	0.0001~9999.999	

Table 4-2-5-2 Command value range of dwell time (Command by P)

The least movement unit	Command value range	Dwell time unit
No.5#1=0	1~99999.999	0.001s or rev
No.5#1=1	1~99999.999	0.0001s or rev

Explanations:

1. G04 is the one-short modal code, which is only enabled at the current line.
2. X value is enabled when the X and P parameters are displayed together.
3. When the X and P values are set as negative, the alarm may occur.
4. The system does not perform the dwell when X and P are not specified.
5. The alarm may be generated if the axes commands (Y, Z, U, V, W, A, B or C) other the X are specified after the G04.

4.2.6 Single Direction Positioning (G60)

Format: G60 X_ Y_ Z_

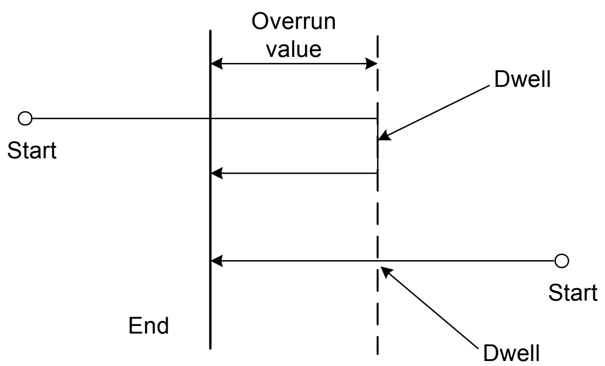


Fig. 4-2-6-1

Function: When the accuracy positioning should be performed for eliminating the machine backlash, which can be performed the accuracy positioning from one direction by using G60.

Explanations:

G60, which is a one-shot G code (Set whether it is modal value by the **bit 0 of parameter No. 48**), can be enabled only in the specified block.

Parameter X, Y and Z are indicated as the end coordinate values in the absolute programming and the tool movement distance in the increment programming. In the tool offset, the path of unidirectional positioning is the one after the tool compensation when the unidirectional positioning is performed.

The marked overrun in the above-mentioned figure can be set by the system parameters **P335, P336, P337 and P338,** and the dwell time can be set by the **P334**, as well the positioning direction can be determined by setting the +/- of the overrun. Refer to the system parameter for details.

For example:

G90 G00 X-10 Y10；

G60 X20 Y25； (1)

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

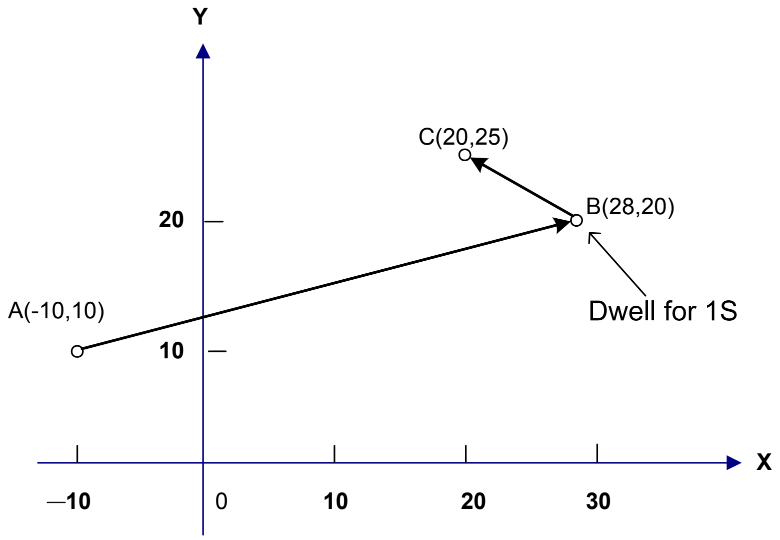


Fig. 4-2-6-2

System parameter:

Table 4-2-6-1

P334	Dwell time for the single direction positioning (Unit: s)
P335	Single direction positioning and overshoot along with the X axis (Unit: mm)
P336	Single direction positioning and overshoot along with the Y axis (Unit: mm)
P337	Single direction positioning and overshoot along with the Z axis (Unit: mm)
P338	Single direction positioning and overshoot along with the 4th axis (Unit: mm)

Note 1: The symbols of the data parameter P335～P338 are expressed the unidirectional positioning, and the value of the parameters are indicated as the overrun

Note 2: overrun value >0, the positioning direction is +.

Note 3: overrun value <0, the positioning direction is -.

Note 4: overrun value =0, do not perform the unidirectional positioning.

4.2.7 System Parameter On-Line Update

Function: This function is used for setting or modifying the values of the cutter radius, length offset, external zero offset value, workpiece zero offset value, additional workpiece zero offset, data parameter and bit parameter in the program.

Format:

G10 L50 N_P _R_； Set or modify the bit parameter

G10 L51 N_ R_； Set or modify the bit parameter

G11； Cancel the parameter input method

Parameter definition:

N: Parameter Number. The parameter series No. to be modified.

P: Parameter bit number. The parameter bit No.

R: Modification value. It is used for specifying the value after the parameter is modified.

The modification specification value can be performed based upon the following codes; refer to the relative Chapters for details:

G10 L2 P_X_Y_Z_A_B_;	Set and alter the external zero offset value or the workpiece zero offset value
G10 L10 P_R_;	Set or alter the length offset value
G10 L11 P_R_；	Set or alter the length wore value
G10 L12 P_R_；	Set or alter the radius offset value
G10 L13 P_R_；	Set or alter the radius wore value
G10 L20 P_ X_Y_Z_A_B_；	Set or alter the additional workpiece zero offset value

Note 1: In the parameter input method, do not specify other NC statements other than the note one.

Note 2: The G10 should be separately specified in block; otherwise, the alarm may occur; It is necessary to cancel the parameter input method by G11 after using the G10 so as not to affect the normal use for the program.

Note 3: The parameter value altered by G10 should be met the range of the system parameter, if does not, the alarm will issue.

Note 4: It is essential to cancel the modal code of the canned cycle before operating the G10; otherwise the system may alarm.

Note 5: The parameter should be restarted after the power is turned off that can not be modified by G10.

Note 6: G20, G21 can not be used the on-line alteration by G10.

Note 7: G10 on-line alters the external zero offset value, workpiece zero offset value, additional workpiece zero offset value and tool offset value; when it is modified in the G91 modal, the commanded offset and the current one will be overlapped by system; in the G90 modal, it is modified based upon the specified offset value.

Note 8: Cancle G10 modal when performing the M00, M01, M02, M30, M99, M98 and 06**.**

Note 9: The bit 7 of parameter No. 0 (Selection mode 0: Common mode, 1: High-velocity and high-accuracy mode) does not support the G10 on-line modification.

4.2.8 Workpiece Coordinate System G54～G59

Function: Specify the current workpiece coordinate system. Select a workpiece coordinate system by specifying a worpiece coordinate system G code in a program.

Format: G54～G59

Explanations:

1. Without code parameters

2. The system, itself, can be set 6 workpiece coordinate systems, which any of the coordinate can be selected by codes 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. The system displays the performed workpiece coordinate system G54～G59, G92 or the additional workpiece coordinate system before the power OFF.

4. When different workpiece coordinate systems are called in a block, specify a movement axis, and then position to the coordinate point under a new workpiece coordinate system; the coordinate will turn into its corresponding coordinate value under a new workpiece coordinate system without specifying a movement axis, however the actual machine's position is invariable.

Example: The coordinate system origin of G54 corresponding with its machine coordinate is (10, 10, 10)

The coordinate system origin of G55 corresponding with its machine coordinate is (30, 30, 30)

The absolute coordinate of the end and the machine coordinate are shown below when performing the program based upon its sequence:

Table 4-2-8-1

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

5. G10 is available to change an external workpiece zero offset value or workpiece zero offset value. Refer to the following methods:

With the code G10 L2 Pp X_Y_Z_

P=0: External workpiece zero offset value (Basis offset value).

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

X_Y_Z_: For an absolute value code (G90), workpiece zero offset value of each axis. For an incremental value code (G91), value to be added (The result of addition becomes the new workpiece zero offset) to the set workpiece zero offset for each axis.

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

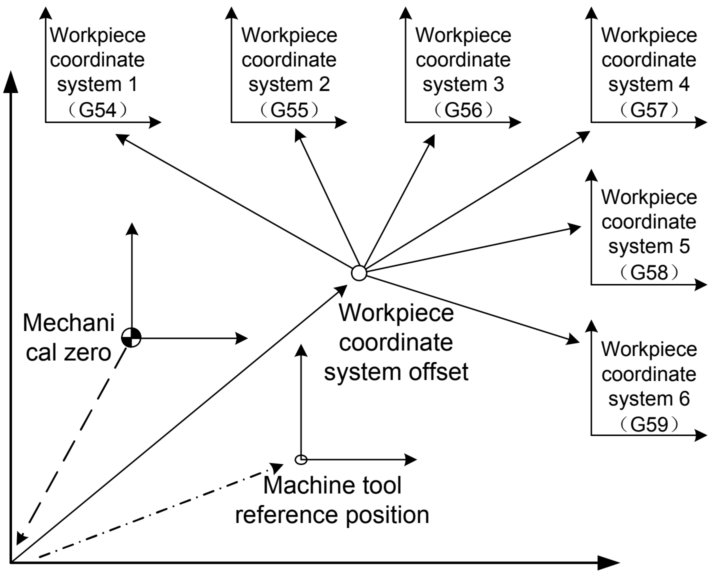


Fig. 4-2-8-1

The above-mentioned Fig. **4-2-8-1** shows that the machine tool returns to the mechanical zero manually after it is starting-up, which is established a machine tool coordinate system by mechanical zero, and therefore, the machine tool reference position occrs and the workpiece coordinate system confirms. The corresponding value of the workpiece coordinate system offset data parameter **P10**～**P13** is the integrated offset value for the 6 workpiece coordinate systems. And the 6 coordinate systems are determined by the distance from the mechanical zero to the each coordinate system zer.

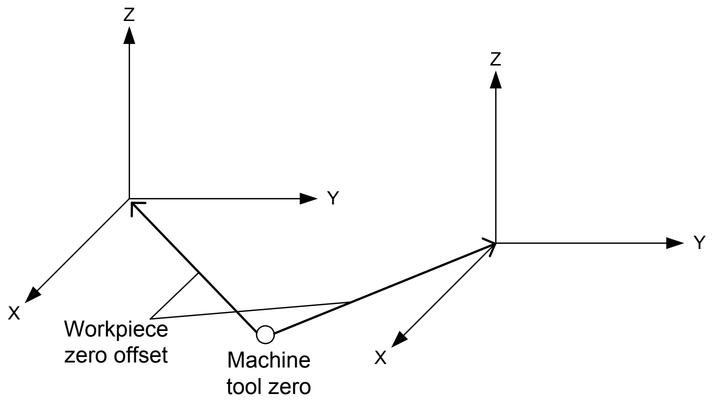


Fig. 4-2-8-2

Example: N10 G55 G90 G00 X100 Y20；

N20 G56 X80.5 Z25.5；

In the above-mentioned example, when the N10 block is executed, rapidly position to the workpiece coordinate system G55 (X=100, Y=20). When the N20 block is executed, rapidly position to the workpiece system G56, and the absolute value is automatically changed into coordinate value (X=80.5, Z=25.5) under the G56.

VOLUME I PROGRAMMING

4.2.9 Additional Workpiece Coordinate System

Also, there are 50 additional workpiece coordinate systems can be used other than the 6 workpiece coordinate systems (from G54 to G59).

Format: G54 Pn

Pn: Codes specifying the additional workpiece coordinate systems. The range of Pn: 1~50.

The setting and restriction of the additional workpiece coordinate system is consistent with the workpiece coordinate system G54~G59.

G10 is available to set a workpiece zero offset value in an additional coordinate system. The methods are shown below:

Code: G10 L20 Pn X_Y_Z_；

n=1 to 50 : Additional workpiece coordinate code.

X_Y_Z_ : Set the axis address and offset value offset by workpiece zero, the specified value is a new offset one.

For an incremental value code (G91), specified value addes the current offset value and gains a new offset value.

G10 command is available for separately changing each workpiece coordinate system.

4.2.10 Machine Coordinate System Selection G53

Format: G53 X_ Y_ Z_

Function: The tool is positioned to the cooresponding coordinate below its coordinte system at a rapid traverse rate.

Explanations:

1. When the G53 is used within the program, its following code coordinate should be the coordinate value below the machine tool coordinate system, and then the machine tool will position to a specified place at a rapid traverse.
2. G53, is one-shot modal code, is only enabled at its current block regardless of the previous defined coordinate system.

Restriction:

Select a machine coordinate system G53

When a command is specified the position 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 enabled 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-specified position such as a tool change position, program the movement in a machine coordinate system based on G53.

Note: When G53 is specified, temperoarily cancel the current comensation and tool length offset, and then recover it at the next compensation axis block to be buffered.

4.2.11 Floating Coordinate System G92

Format: G92 X_ Y_ Z_

Function: Set a floating workpiece coordinate system. Three code parameters are specified an absolute coordinate value of the current tool in a new floating workpiece coordinate system.

Explanations:

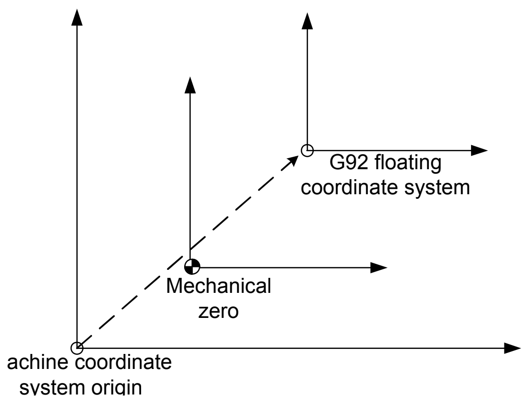


Fig. 4-2-11-1

1. The above-mentioned Fig. 4-2-11-1 shows that the cooresponding origin of the G92 floating coordinate system is the value below the machine coordinate system, which is nothing to do with the workpiece coordinate system.

1) Before calling a workpiece coordinate system

2) Before the machine tool zero return

Generally, the G92 floating coordinate system, is used for correcting the machining of the temporary workpiece, is operated at the beginning of the program, or specify the G92 in MDI mode before automatically operating the program.

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

1) Coordinate system confirmation based upon a tool nose

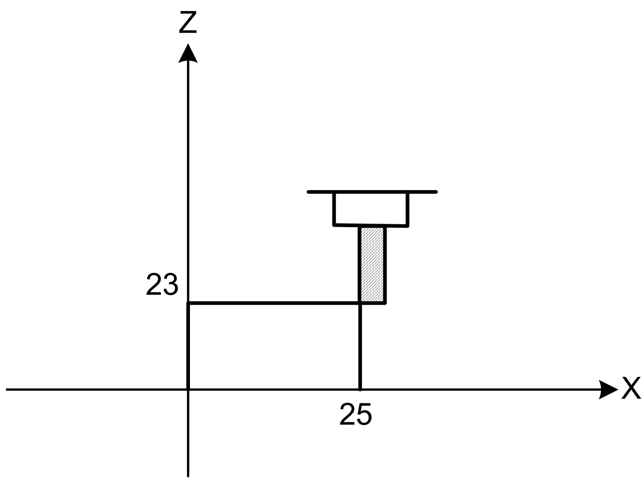


Fig. 4-2-11-2

The Fig. **4-2-11-2** shows that, G92 X25 Z23, the tool nose position is treated as (X25, Z23) point in the floating coordinate.

2) One fixed point at the tool handle is regarded as the reference point coordinate system:

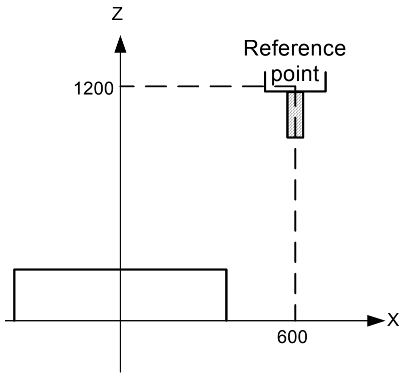


Fig. 4-2-11-3

The Fig. 4-2-11-3 shows that the G92 X600 Z1200 is used; specify the coordinate system setting (When one reference point on the tool handle is regarded as the start). And therefore, if it is movement according to the absolute code in the program, and then the reference position moves to the specified position; the tool length compensation should be added; its value is the difference from the reference point to the tool nose.

Note 1: If the G92 setting coordinate system is used in the tool offset. The tool offset does not add for the tool length compensation, and set the coordinate system by G92

Note 2: When, however, there is no alternative other than to cancel the cutter compensation before using the G92 code.

4.2.12 Plane Selection G17/G18/G19

Format: G17/G18/G19

Function: Select the planes for circular arc interpolation, cutter compensation or drilling, and boring. In this case, the plane selection can be performed by G17/G18/G19.

Explanation: Without command parameter, the system is regarded as G17 by default when starting up. Also, set the bit 1, 2 and 3 of the parameter No.: 31 to determine the default plane after starting up. The corresponding relationships between codes and planes are as follows:

G17-------------XY Plane

G18-------------ZX Plane

G19-------------YZ Plane

The plane is invariable when the G17, G18 and G19 are not belonged to the specified block.

Example: G18 X_ Z_; ZX plane

G0 X_ Y_; Plane invariable (ZX plane)

Additional, the movement code is regardless of the plane selection. For example, the following code shows that the Y axis is out of the ZX plane, and the Y axis movement is regardless of the ZX plane.

G18Y_;

Prompt: At present, only the canned cycle in the G17 plane is supported. For the specification's sake, it is better to define the specified plane in the corresponding block when the programming is performed, specially when several persons are shared with a same system. So that, the accident or abnormality may avoid due to the incorrect programming.

4.2.13 Polar Coordinate Start/Cancel G16/G15

Format: G16/G15

Function:

G16 specifys the polar coordinate of positioning parameter, which means the beginning of the method.

G15 specifys the polar coordinate of positioning parameter, which means the cancellation of the method.

Explanation:

Without command parameter

Set the G16, the coordinate value can be inputted by polar coordinate radius and angle. The plus direction of the angle is CCW of the selected plane first axis + direction, and the minus directionis CW. Both the radius and angle can be specified in either absolute or incremental command (G90, G91).

After the G16 occurs, the 1^{st} axis of the positioning parameter of the tool movement command expresses the polar-radius below the polar coordinate system; the 2nd axis means the polar-angle below the polar coordinate system.

Set the G15 to cancel the polar coordinate method, so that the coordinate value returns to use the rectangle coordinate input.

The specification of the polar coordinate origin:

1. In the G90 absolute method, the workpiece coordinate system zero is regarded as the polar coordinate origin when specifying by G16.

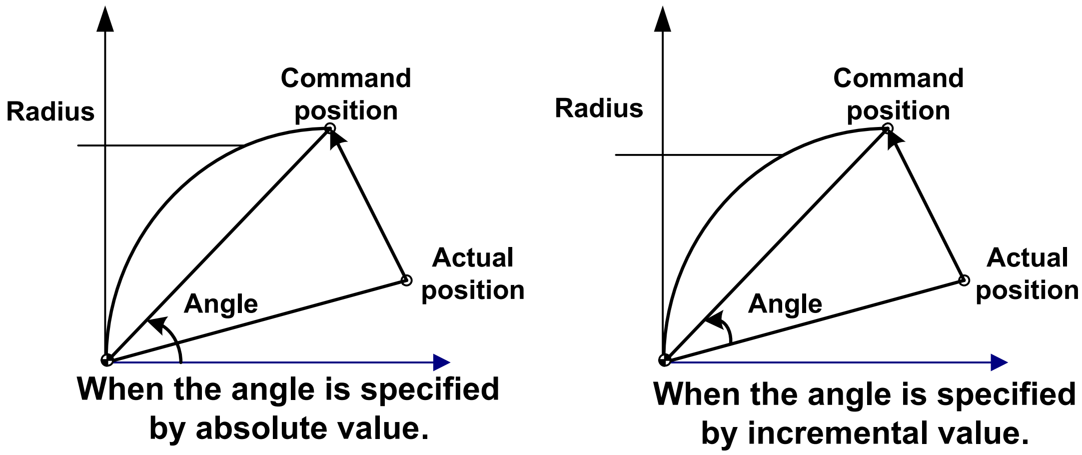


Fig. 4-2-13-1

2. In the G91 increment method, the current point is treated as the polar coordinate origin when specifying by G16.

For example: Bolt hole circle (The zero of the workpiece coordinate system is set as the origin of polar coordinate, select the X—Y plane).

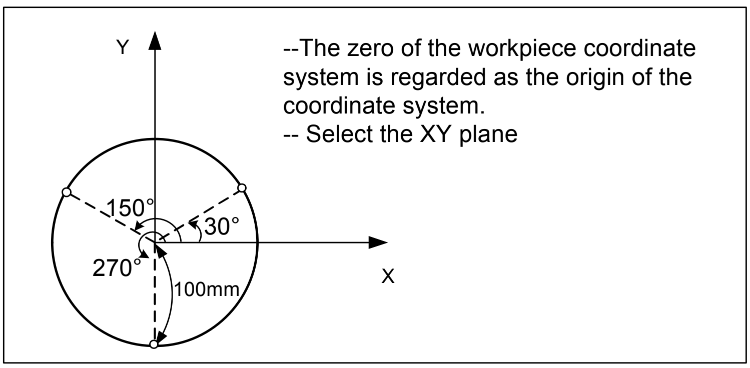


Fig. 4-2-13-2

- Specify an angle and a radius with absolute value

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

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

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

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

G15 G80; Canceling the polar coordinate code

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

G17 G90 G16; Specifying the polar coordinate code and selecting XY plane; setting the zero of the workpiece coordinate system as the origin of the polar coordinate system

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

G91 Y120; Specifying a distance of 100mm and an incremental angle of +120°

Y120; Specifying a distance of 100mm and an incremental angle of +120°

G15 G80; Canceling the polar coordinate code

In addition, when the polar coordinate is programmed, it is notice to set the current coordinate plane. The polar coordinate plane is related with the current one; for example: G91, if the current coordinate plane is G17, the X and Y axes components at the current tool position is the origin. If the current coordinate plane is G18, and then the Z and X axes components at the current tool position is the origin.

If the positioning parameter of the 1st hole circle command followed with G16 does not specify, the current tool position of the system is regarded as the default position parameter of the hole circle. At present, the 1st canned cycle code followed with the polar coordinate should be completed; otherwise, the tool path will incorrect.

The parameter word of the tool movement command positioning parameter is related with the concrete plane selection modal other than the hole circle followed with the G16. The default current tool position, followed with the movement code, after using the G15 to cancel the coordinate, is the start of the movement code.

4.2.14 Scaling Within Plane G51/G50

Format:

G51 X_ Y_ Z_ P_ (X.Y.Z: The absolute value code of scaling center coordinate value, P: Scling each axis with same proportion)

… The machining block of scaling

G50 Canceling the scaling

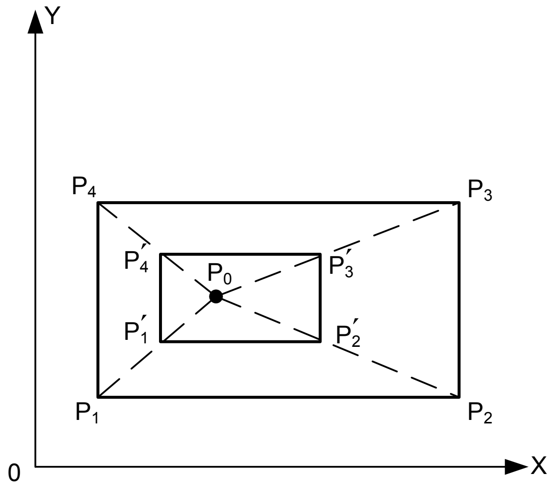
Or, G51 X_ Y_Z_ I_ J_ K_ (Each axis is separately performed acoording to different scaling (I, J and K)

… Machining block of scaling

G50 Canceling the scaling

Function:

The programmed shape is regarded as a center with the specified position by G51, scaling the same or different proportion. It is necessary to point out that the G51 is specified (otherwise, the unexpected situation may occur) with the single block and then cancel it by G50.



P_0: Scaling center

Fig. 4-2-14-1 scaling (P1P2P3P4→ P1'P2'P3'P4')

Expanations:

1. Scaling center: 3 positioning parameters X_Y_Z_ can be performed with G51, which are the optional parameters. The positioning parameter is used to specify the scaling center of the G51. If the positioning parameter does not specify, the system is set the current position of the tool as the scaling center. The scaling center is always specified with the absolute positioning mothod regardless of the current positioning method is absolute or increment. Moreover, the parameter in G51 is also indicated with the rectangular coordinate system in G51 code based upon the polar coordinate G16 method.

Example: G17 G91 G54 G0 X10 Y10;

G51 X40 Y40 P2； Incremental method, scaling center is still the absolute coordinate (40, 40) in the G54 coordinate system.

G1 Y90； Parameter Y is still use the incremental method.

2. Scaling: The scaling is always expressed with the absolute method regardless of the current method is G90 or G91.

The scaling can be set in parameter other than specifying in program; data parameter P330 sets the scaling multiplication of each axis; data parameter **P331~P333** are separately corresponding to scaling of the 1st, 2nd and 3rd axis; if the scaling codes are not performed here, and when the bit 6 of parameter No.:47 is set to 0, the scaling can be performed according to the setting value from the data parameter P330; when the bit 6 of No.:47 is set to 1, the scaling is then performed based upon the setting value of data parameter P331～P333.

If the parameter value of the parameters P or I, J and K are negative, the corresponding axes are performed the image.

3. Scaling setting: Whether the scaling function is used by the **bit 5 of parameter No.: 60**; Whether the **bit 3 of parameter No.: 47** sets the 1st axis is enabled; Whether the **bit 4 of parameter No.: 47** sets the 2nd axis is enabled; Whether the **bit 5 of parameter No.: 47** sets the 3rd axis is enabled; **bit 6 of parameter No.: 47** set the specification method of scaling override along with each axis (0: Using P commands each axis; 1: Using I, J and K .
4. Cancel the scaling: After the scaling is cancelled by using the G50 code, and when the movement code is followed, the coordinate scaling is cancelled; the tool position is the start point of the movement code.
5. Scaling state can not specify the G codes (such as G27～G30) of reference position return and the one (G53～G59, G54P1~G54P50, G92) of the coordinate system. If these G codes must be specified, it should be specified after cancelling the scaling function; otherwise, the system alarm occurs.
6. Tool does not draw out the ellipse path even if the arc interpolation and each axis are specified the different scaling.

When the scaling along with each axis is different, and the arc interpolation is programmed by radius R, its interpolation figure is as the Fig. **4-2-14-2** (The proportion of X axis is 2, Y is 1).

G90 G0 X0 Y100;
G51 X0 Y0 Z0 I2 J1;
G02 X100 Y0 R100 F500;
The above-mentioned commands are equal to the following one:
G90 G0 X0 Y100;
G02 X200 Y0 R200 F500;
The proportion of radius R is scaled based upon the bigger one of the I or J.

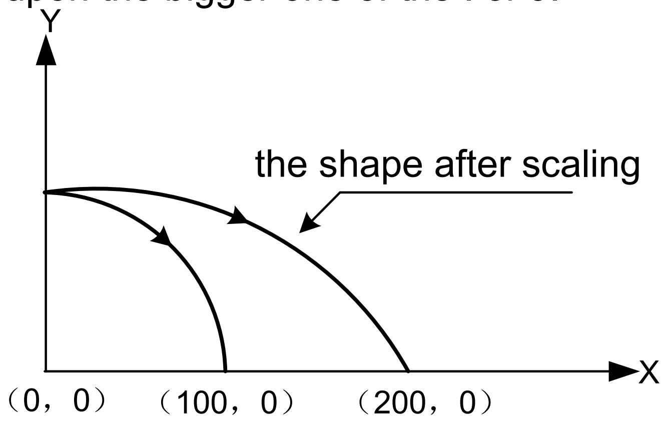


Fig. 4-2-14-2 The scaling of circular arc interpolation

When the scaling along with each axis is different, and when the arc interpolation is programmed by I, J and K, the arc is not performed and then the system alarms.

7. The scaling is disabled to the tool offset value, refer to the Fig. **4-2-14-3.**

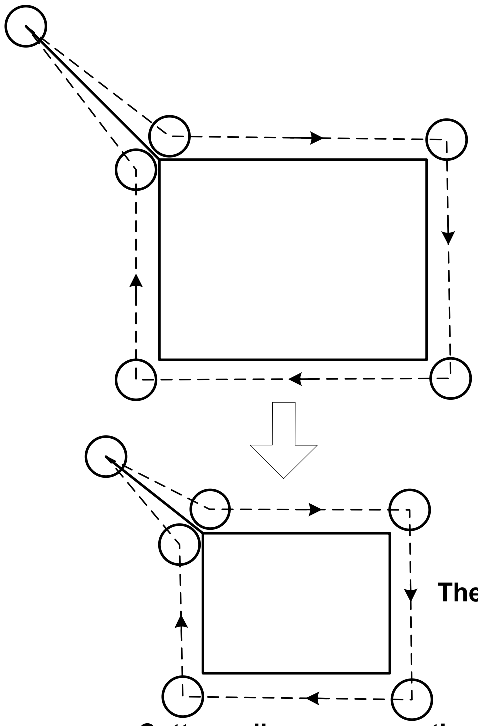


Fig. 4-2-14-3 The scaling of current compensation

Image program example:

Main program:

G00 G90;

M98 P9000;

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

M98 P9000;

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

M98 P9000;

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

M98 P9000;

G50;

M30;

Subprogram:

O9000;

G00 G90 X60.0 Y60.0;

G01 X100.0 F100;

G01 Y100;

G01 X60.0 Y60.0；

M99；

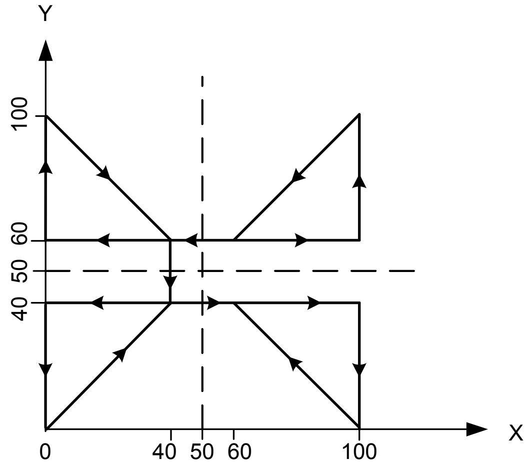


Fig. 4-2-14-4

Restrictions:

1. In the canned cycle, the depth of Z axis cutting value, the 1st cutting depth distance W of the return value d, Z and the movement scaling of the rapid cutting distance V, are all disabled.

2. In the manual operation, the movement distance can not be increased or decreased by scaling function.

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

Note 2: There is an axis of the specification plane is performed the image, and its result is shown below:

1) Circular code……………Negative rotation direction

2) Cutter compensation C…………..Negative offset direction

3) Coordinate system rotation……………Negative rotation angle

4) Change the direction of the cutting feed

4.2.15 Coordinate System Rotation G68/G69

The machining workpiece consists of the figures by many same shapes, which can be programmed by coordinate rotation function; only the subprogram should be programmed for the figure unit, and then call the subprogram by the rotation function.

Code format: G17 G68 X_ Y_ R_；

Or G18 G68 X_ Z_ R_；

Or G19 G68 Y_ Z_R_ ；

G69；

G17~G19： Plane selection.

X_，Y_，Z_: Absolute command for two of the X_, Y_ and Z_ axes that correspond to the current plane selected by a command (G17,

G18 and G19). The command specifies the coordinates of the center of rotation for the values.

Specifying the axis commands (Y, Z, U, V, W, A, B and C) other than the X after the G04 command, the system may alarm, and it is necessary to add the **absolute command** explanation in the User Manual; **specify the rotation center followed with the G68**.

R_: **Angle shifting, positive value means CCW.**

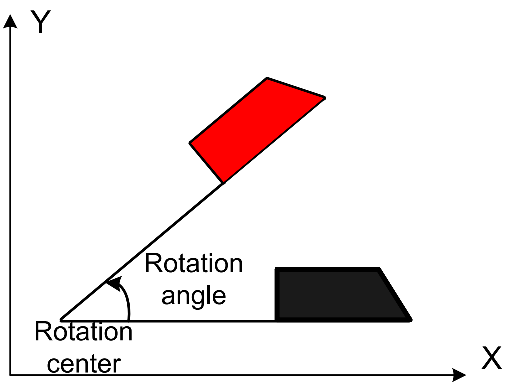


Fig. 4-2-15-1

Function: The programmed shape inside the plane is rotated based upon that the specified center is regarded as origin by the G68. G69 is used for cancelling the coordinate system rotation.

Explanations:

1. G68 can be held 2 positioning parameters, which are regarded as the optional parameters. The positioning parameters are specified the center of the rotation operation. If the rotation center does not specify, the system is regarded the current tool position as the rotation center. The positioning parameters are related with the current coordinate plane, and select the X, Y in the G17; Z,X in G18; Y, Z in G19.
2. The system is treated the specified point as the rotation center if the current positioning is absolute method. The system is regarded the current point as the rotation center if the positioning is relative method. G68, also, can be held a command parameter R, its parameter value is the angle for the rotation, the positive value is CCW, and its rotation angle is degree. The used rotation angle is determined by data parameter P329 when there is no rotation angle in the coordinate rotation.
3. The system is regarded the current tool position as the rotation center in the G91 mode; whether the rotation angle is performed the increment is set by the bit 0 of parameter No.: 47 (The rotation angle of coordinate, 0: Absolute code, 1: G90/G91 code).
4. The plane operation selection can not be performed when the system is on the rotation mode; otherwise, the alarm occurs. It is necessary to note it when programming.
5. The G codes (G27～G30, etc.) of the reference point return and the G codes (G53～G59, G54P1~G54P50, G9, etc.2) of command coordinate system can not be specified in the

coordinate system rotation method. When, however, there is not alternative other than to specify these G codes, it is better to specify it after the rotation function is cancelled; otherwise, the system alarm may issue.

6. Perform the cutter compensation, tool length compensation, tool offset and other operations after the coordinate system rotates.
7. Perform the coordinate rotation code in the scaling method (G51), the coordinate value of the rotation center is also scaled instead of scaling the corner; when the movement code issues, firstly perform the scaling and then rotate the coordinate.

Example 1: Rotation

G92 X-50 Y-50 G69 G17；

G68 X-50Y-50 R60；

G90 G01 X0 Y0 F200；

G91 X100；

G02 Y100 R100；

G3 X-100 I-50 J-50；

G01 Y-100；

G69；

M30；

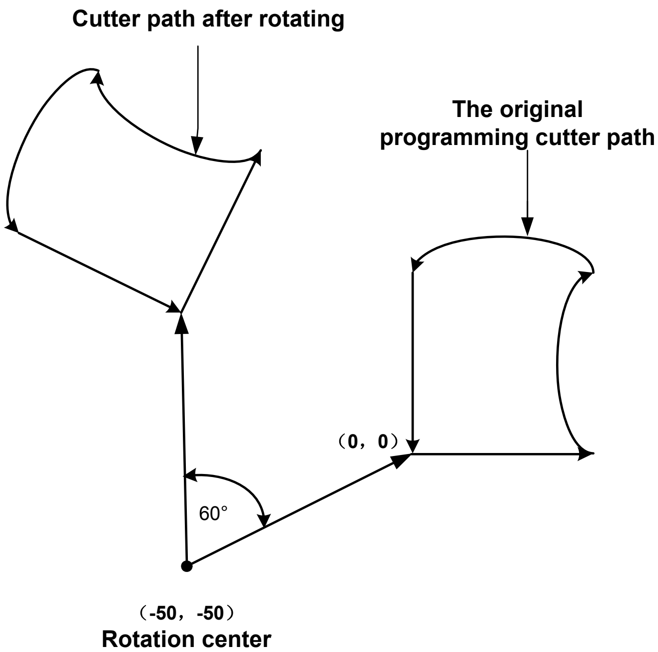


Fig. 4-2-15-2

Example 2: Scaling with rotation:

G51 X300 Y150 P0.5；

G68 X200 Y100 R45；

G01 G90 X400 Y100；

G91 Y100；

X-200；

Y-100；

X200；

G69 G50；

M30；

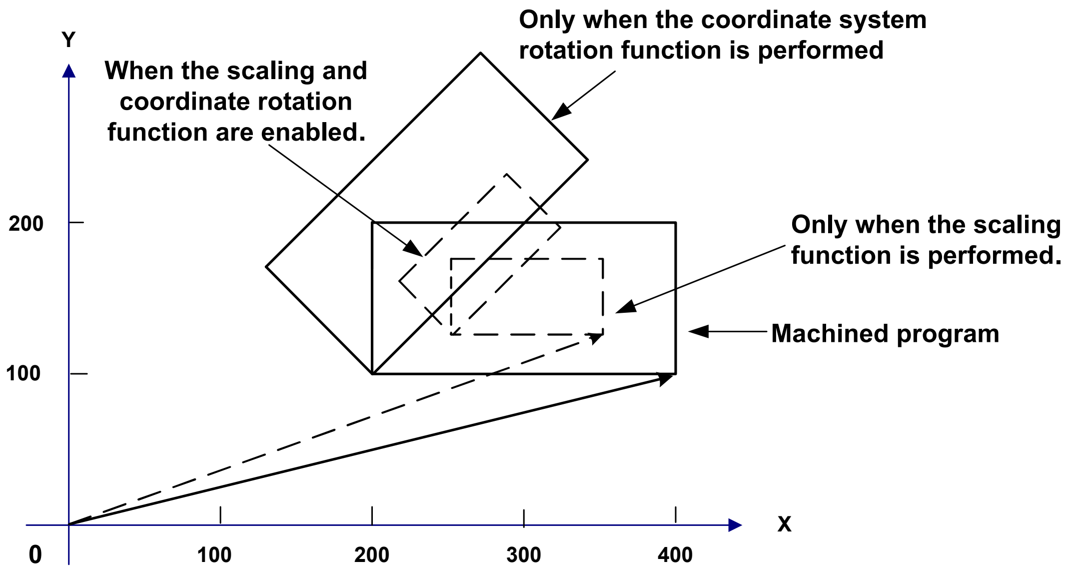


Fig. 4-2-15-3

Example 3： Repeatedly use the G68

According to the program (Main program)

G92 X0 Y0 Z20 G69 G17；

M3 S1000；

G0 Z2；

G42 D01； (Tool offset setting)

M98 P2100(P02100)； (Subprogram calling)

M98 P2200L7； （Call for 7 times）

G40；

G0 G90 Z20；

X0Y0；

M30；

Subprogram 2200

O2200

G91

G68 X0 Y0 R45.0; (Relative rotation angle)

G90;

M98 P2100 ;

(Subprogram O2200 calls the subprogram O2100)

M99;

Subprogram 2100

O2100 G90 G0 X0 Y-20;

(Establish a right tool compensation method)

G01Z－2 F200;

X8.284;

X14.142 Y-14.142;

M99;

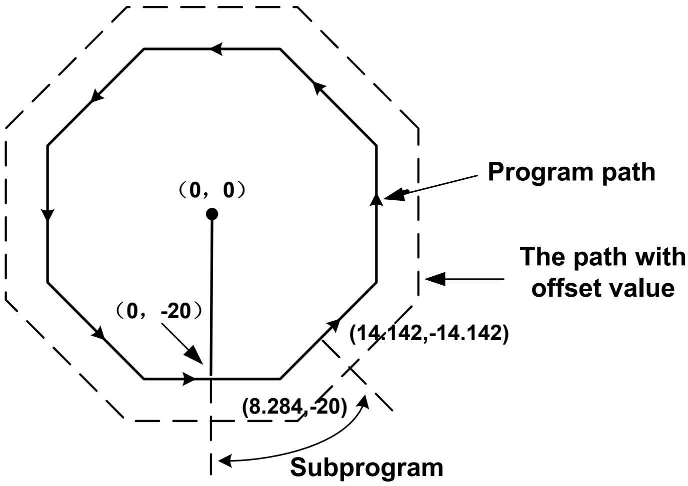


Fig. 4-2-15-4

4.2.16 Skip Function G31

Code format: G31 X_Y_Z_

Function: Linear interpolation can be commanded following the G31 code, like G01. If an external skip signal is input during the execution of this code, and this execution of the code is then interrupted and the next block is performed. The skip function is used when the end of the machining is not programmed but specified with a signal from the machine, for example, in grinding. It is used also for measuring the dimension of a workpiece.

Explanations:

1. G31 is one-shot mode G code, which is only enabled in the specified block.
2. The alarm occurs if the G31 code issues when the cutter compensation is applied; the cutter compensation should be cancelled before G31 code.

Example:

The next block to G31 is 1 axis movement specified by an incremental value; refer to the Fig. **4-2-16-1:**

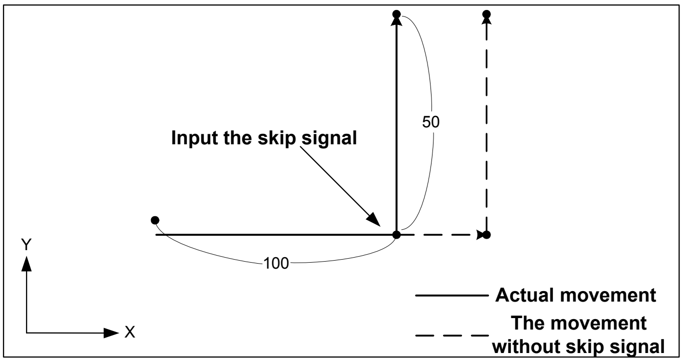


Fig. 4-2-16-1 The next block is 1 axis movement specified by an incremental value

The next block to G31 is 1 axis movement specified by an absolute value; refer to the Fig. **4-2-16-2:**

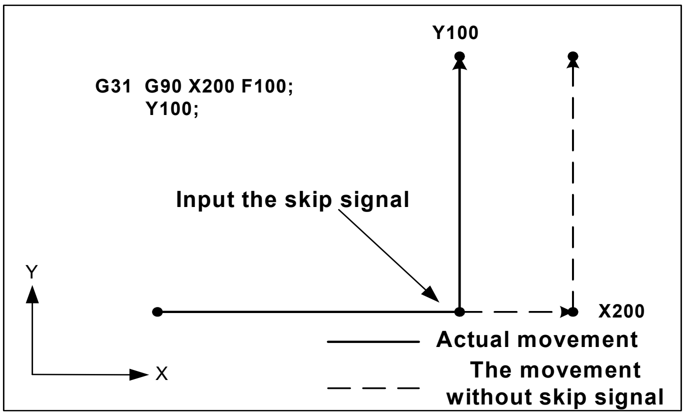


Fig. 4-2-16-2 The next block is 1 axis movement specified by an absolute value

The next block to G31 is an absolute command for 2 axes; refer to the **4-2-16-3:**

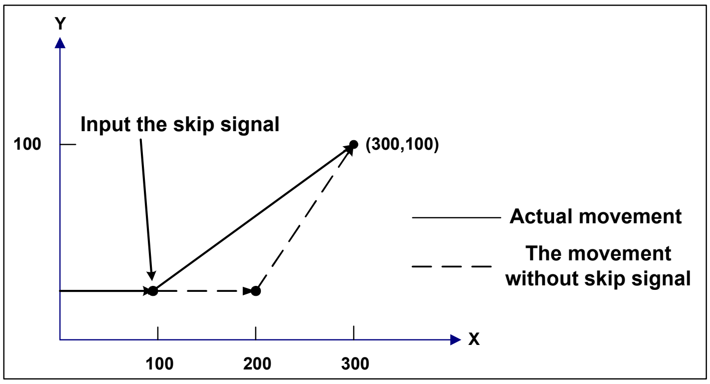


Fig. 4-2-16-3 The next block is an absolute value for 2 axes

Note: It can be set by the bit 6 of parameter No.: 02 (SKIP signal, it is regarded as the signal input when 0 is set to 1 and the 1 is set to 0).

4.2.17 Inch/Metric Conversion (G20/G21)

Code format: G20: Inch input

G21: Metric input

Function: Either inch or metric can be carried out the program input.

Explanations:

Change the unit of the following values after the inch/metric is converted.

The feedrate, position code, workpiece zero offset value, tool compensation value, the scale unit of MPG and the movement distance in the incremental feed are specified by F code.

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

Notice: 1. The tool compensation value should be preseted based upon the least input incremental unit when the inch shifts to metric or the metric converts into inch.

2. The 1st G28 code is identical with the same reference point return from the operation of the intermediate point when the inch shifts to metric or the metric converts into inch.

3. When the least input incremental unit is different with the least command incremental unit, the most error is an half of the least command unit, and this error does not accumulate.

4. Whether the program is input the inch or metric can be determined by the bit 2 of parameter No.: 00.

5. Whether the program is output the inch or metric can be determined by the bit 0 of parameter No.: 03.

6. It is essential to specify the G20 or G21 within the single block.

4.2.18 Optional Angle Chamfering/Corner Arc

Code format: **, L_: Chamfering**

, R_: Corner arc

Function: When the above-mentioned codes are added to the end of a block that specifies linear interpolation (G10) or circular interpolation (G02, G03), a chamfering or corner arc is automatically inserted during machining. Blocks specifying chamerfering and corner arc can be specified consecutively.

Explanations:

1. After L, the chamfering specifies the distance from the virtual corner point to the start and end points. The virtual corner point is the one that would exist if chamfering were not performed; refer to the following figure:

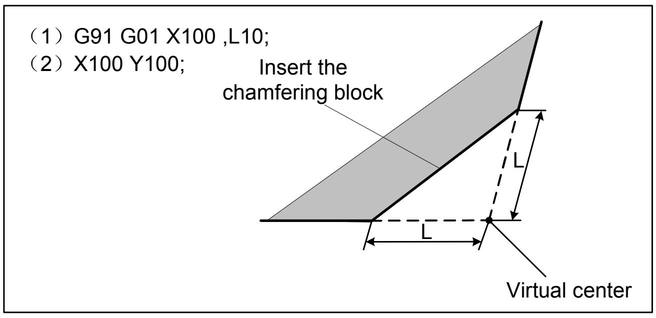


Fig. 4-2-18-1

2. After R, specify the radius for corner arc, refer to the following figure:

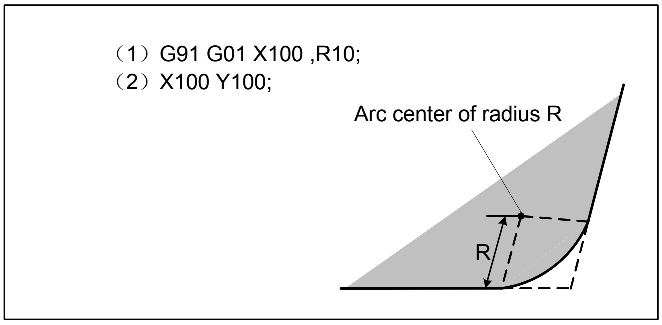


Fig. 4-2-18-2

Restrictions:

1. Chamfering and corner arc only can be performed within the specified plane, and these functions cannot be performed for parallel axes.

2. If the inserted chamfering or the block of the corner arc causes the tool to go beyond the original interpolation movement range, the alarm is issued.

3. The corner arc can not be specified in the screw machining block.

4. When the specified chamfering value and the corner value are negative, the system is assigned with the absolute value.

4.3 Reference Position G Code

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

There are 3 code operation methods for the reference position, refer to the Fig. **4-3-1,** the tool can be automatically moved to the reference position along with the specified axis in code via an intermediate point by G28; the tool can be automatically moved to the appointed point along with the specified axis in code via an intermediate point by G29.

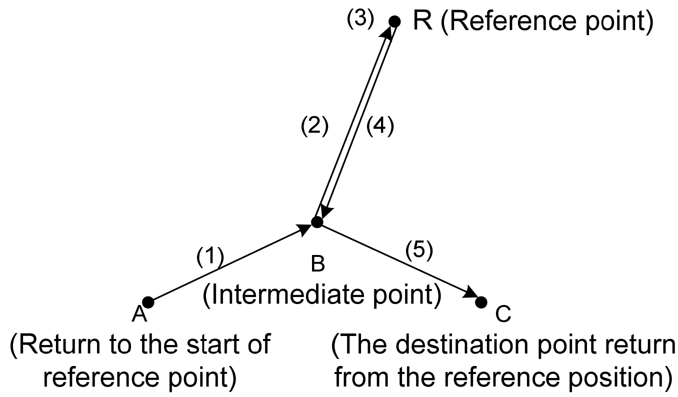


Fig. 4-3-1

4.3.1 Reference Position Return (G28)

Format: G28 X_ Y_ Z_

Function: G28 code is used for performing the operation, which returns to the reference position (some one special position on the machine tool) by the intermediate point.

Explanation:

Intermediate point:

The intermediate point is specified by the code parameter in G28, which can be expressed by the absolute value code or the incremental value code. The intermediate point coordinate value of the code axis is registered during the execution of the block for supporting the G29 (return from the reference position).

Notice:

The coordinate of the intermediate point is registered in the CNC, however, the coordinate value of the axis specified by G28 is only registered each time. The other axes are not specified, which are used the coordinate value specified by G28 before. And therefore, if you are not comprehend the intermediate point in the default system when using the G28 command, it is better to specify each axis. Consider it based upon the N5 block in the following example 1.

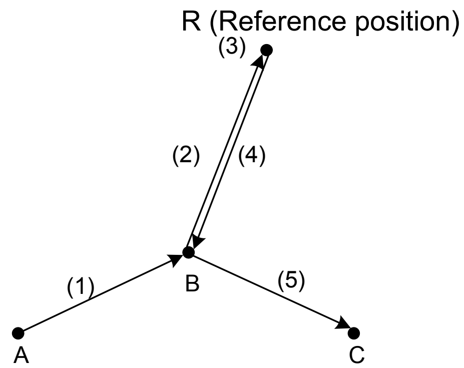


Fig. 4-3-1-1

1. The operation of the G28 block can be decomposed into following items (Refer to the Fig.

4-3-1-1):

(1) The current position positions to the intermediate point position (Point A → Point B) of the command axis at the rapid traverse rate.

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

2. G28 is one-shot modal code, which is only enabled to the current block.

3. It is supported the referernce position return of the composition both the single or multi-axis. The coordinate of the intermediate point is registered into CNC when the coordinate coordinate is converted.

Example:

N1 G90 G54 X0 Y10；

N2 G28 X40 ; Setting the intermediate point on X axis is X40 at the G54 workpiece coordinate system, return to the reference point by (40, 10); that is, the X axis is separately returned to the reference position.

N3 G29 X30 ; Return to the (30, 10) via (40, 10) from reference; that is, the X axis is returned to the destination alone.

N4 G01 X20；

N5 G28 Y60 ; Intermediate point Y60。

N6 G55； The workpiece coordinate system conversion, the intermediate point changes into the point (40, 10) at the G55 coordinate system from the point (40, 60) at the G54 workpiece coordinate system.

N7 G29 X60 Y20 ; Reference position returns to the point (60, 20) via the intermediate point (40, 60) at the G55 workpiece coordinate system.

G28 automatically cancels the tool compensations. This code, generally, is used (Tool-change is performed at the reference position after returning the reference point) in the automatical tool-change, therefore, when using this code, it is necessary to cancel the cutter compensation and tool length compensation firstly. Refer to the 1st reference position setting in the data parameter **P45～P48.**

4.3.2 2nd, 3rd, and 4th Reference Position Retrun (G30)

Set 4 reference positions in the machine coordinate system. In a system without an absolute-position detector, the 2nd, 3rd, and the 4th reference position return functions can be used only after the automatical reference position return (G28) or the manual reference position return is performed.

Format:

G30 P2 X_ Y_ Z_； the 2nd reference position return (P2 can be omitted)

G30 P3 X_ Y_ Z_; the 3rd reference position return

G30 P4 X_ Y_ Z_; the 4th reference position return

Function: G30 performs the operation that the intermediate point specified in G30 returns to the appointed reference position.

Explanations:

1. X_ Y_ Z_; Specify the code (Absolute value/incremental value code) at the intermediate position

2. The setting and restriction of the G30 code are identical with the G28, the 2nd, 3rd, and the 4th reference position setting are shown the data parameters **P50～P63.**

3. G30 code can be used together with the G29 code (return from the reference position), and its setting and restriction are same as the G28.

4.3.3 Automatically Return from Reference Position (G29)

Format: G29 X_ Y_ Z_

Function: G29 performs the operation that the appointed point returns from the reference position (or current point) via the intermediate point specified by G28, G30.

Explations:

1. The motion of G29 block can be divided into the following steps (Refer to the Fig. 4-3-1-1):

   (1) Positioning to the intermediate point (Point R → Point B) defined in G28, G30 from the reference position (current point) at the rapid traverse rate.

   (2) Positioning to the specified point (Point B → Point C) from a new intermediate point at the rapid traverse rate.

2. G29 is one-shot modal information, which is only enabled for the current block. Generally, it is necessary to immediately specify the code return from reference position following with the G28, G30.

3. The optional parameters X, Y and Z in the G29 code format are used for specifying the destination point (that is, the point C in Fig. 4-3-1-1) from the reference point return, which can be indicated by an absolute value code or an incremental value code. For an incremental, the code value specifies the incremental value departed from the intermediate point. When some axes are not specified, that is, failure movement value relative to the intermediate point for these axes. G29 is only followed by the command with one axis, which is the single axis return, the other axes will invariable.

Example:

G90 G0 X10 Y10；

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

G29 X30; Return from the reference position (60, 30) via an intermediate point (30. 30); it is note that it is in an incremental program method, and the component along with the X axis direction is 60.

The intermediate point specified by G29 is assigned by G28, G30 code. For the definition of the intermediate point, specification and system default, refer to the explantions in G28 code.

4.3.4 Reference Position Return Check (G27)

Format: G27 X_ Y_ Z_

Function: G27 performs the reference position detection

Explanations:

1. G27 code, tool positions at the rapid traverse rate. If the tool reaches to the reference position, the reference position return does not conduct. However, if the tool does not reach to the reference position, the alarm occurs.
2. Machine tool locking state, that is, specify the G27 code, the tool is already returned to the reference position automatically, and the completion signal return does not perform the breakover, too.
3. In the offset mode, the tool arrived position is performed by G27 command, which is gained for adding the offset value. Therefore, if the position adding to the offset value is not the reference position, the signal does not conduct, and the alarm occurs. Usually, it is necessary to cancel the tool offset before using the G27 code.
4. The X, Y or Z coordinate point position specified by G27 is the one below the machine tool coordinate system.

4.4 Canned Cycle G Code

Canned cycle, is a machining operation, can be performed based upon multiple block codes by a block with G function, so that the program can be simplified and the programming becomes easy for the programmer (This system is only owned the canned cycle of the G17 plane).

The general procedure of canned cycle:

A canned cycle consists of a sequence of 6 operations; refer to the Fig. **4-4-1.**

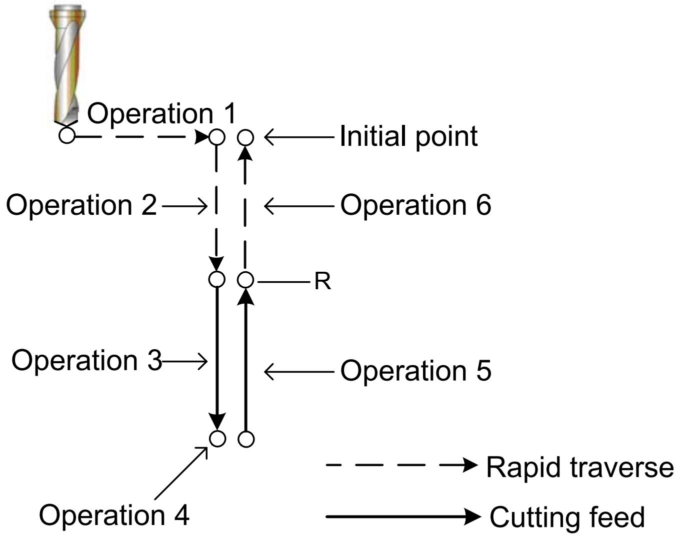


Fig. 4-4-1

Operation 1: The positioning both the X axis and Y axis (Another axis is considered)

Operation 2: Move to the point R at a rapid traverse rate

Operation 3: Hole machining

Operation 4: The operation at the bottom of hole

Operation 5: Return to point R

Operation 6: Move to the initial point at a rapid traverse rate

Position in the XY plane, hole machining performs along with the Z axis. Specifying a canned cycle operation is determined by 3 methods, which are separately specified by G codes.

1) Data form

G90 absolute value method; G91 Incremental value method

2) Return to the point plane

G98 Initial point plane; G99 Point R plane

3) Hole machining method

G73, G74, G76, G81～G89

Plane both in initial point Z and point R

Initial point plane: Tool's absolute position along with the Z axis direction before the canned cycle state.

Point R plane: It is also called Safety Plane, which is the position along with the Z axis direction in the canned cycle when the rapid traverse turns into cutting feed. Generally, position at some certain distance above a workpiece surface to prevent the tool from impacting to the workpiece; and guard the adequate distance to complete the acceleration procedure.

G73/G74 /G76/G81～G89 are specified the overall data (hole position data, hole machining data, repeated times) of the canned cycle to compose a block.

Z, R: Either the parameter Z at the bottom of a hole or the parameter R may be absent when performing the 1st drilling hole; the system is only altered the modal and performed the Z axis operation.

The format of the hole machining method is as follows:

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

Hole machining data

Hole position data

Hole machining method

Wherein, the basis meanings of hole position data and hole machining data are shown in the table **4-4-1:**

Table 4-4-1

Specified content	Parameter word	Explanation
Hole machining method	G	Refer to the table **4-4-3,** note above-mentioned restrictions
Hole position data	X，Y	Specify the hole position by an absolute value or an incremental value; the control is same as the G00 positioning.
Hole machining data	Z	Figure 4-4-2 shows that the incremental value specifies the distance to the bottom of a hole from point R or the absolute value commands the coordinate value at the bottom of a hole. Refer to the feedrate in the Fig. 4-4-1, the feedrate is specified by F in operation 3; in the operation 5, it is the rapid traverse rate or the feedrate by F code based upon different hole machining methods.
	R	The figure 4-4-2 shows that the distance from initial point plane to point R is specified by incremental value, or the coordinate value of the point R is specified by absolute value. The feedrate, in the operations 2 and 6, are performed at the rapid traverse rate; refer to the Fig. 4-4-1.
	Q	Specify the cutting value for each time in G73, G83 or the translation value (incremental value) in G76, G87.

Specified content	Parameter word	Explanation
	P	Specify a dwell time at the bottom of a hole. A canned cycle codes can be accompanied a parameter P_, in the P_ parameter value, perform the dwell operation time after the specified tool reaches to the plane Z; its unit is ms. The least value of parameter is determined by P281, and its most value is set by P282.
	F	Specify the cutting feedrate
	K	Change only the modal. Specify the repeated time in the K_ parameter value, and the K is only enabled with the specified block. If it omits, it regards as once bey default. The maximum drilling times is 99999; when the negative value is specified, and then perform it base upon the absolute value; when it is 0, the drilling hole operation is omitted.

Restrictions:

➢ Canned cycle G code is the modal one, which always keeps enabling until the G code of the specified canned cycle is cancelled.

➢ The G80 and the G code of group 01 is cancelled the G code of canned cycle.

➢ Machining data is always performed till the end of the canned cycle once it is specified in a canned cycle; therefore, the necessary hole machining data should be specified at the beginning of the canned cycle, and only the altered data should be specified in the following canned cycle.

Note 1: The cutting feedrate specified by F is held even if the canned cycle is cancelled.

Note 2: the scaling function along Z axis (cutting axle direction) is disabled when the canned cycle is performed.

Note 3: In the single mode, the canned cycle mostly uses 3 sections machining methods: positioning → R plane → Initial plane.

Note 4: In the canned cycle, when the bit 1 of parameter No.: 36 is set to 1, the hole machining data and the hole position data are eliminated in the REST or ESP. The examples of data hold and clean in the above-mentioned are shown below:

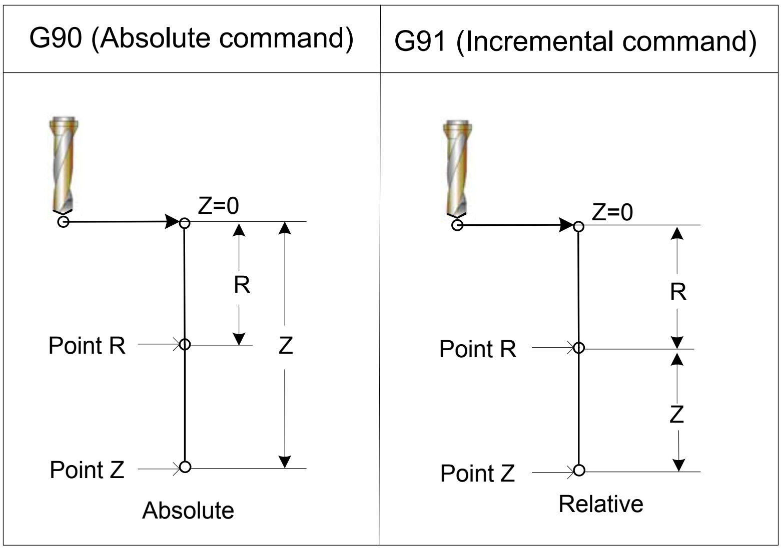
Table 4-4-2

Sequence	Data specification	Explanation
①	G00X_M3;	
②	G81X_Y_Z_R_F_;	Specify the required value for the Z, R and F because it is at the beginning.

③	Y_;	G81 and Z-R-F- can be omitted because the hole machining method specified in hole ② is identical with the hole machining data. The position movement Y of a hole is machined once by G81 method.
④	G82X_P_;	It is only moved along with X axis relative to the hole ③. Machining is performed by G82, and the hole machining data is performed by the Z, R and F specified in ② and the P specified in ④.
⑤	G80X_ Y_	Do not perform the hole machining. Cancel the overall hole machining data.
⑥	G85X_Z_R_P_;	Z and R should be specified again because the overall data are cancelled in ⑤. F can be omitted because it is same with the specified one in ②. P, is unavaliable in this block, is only saved.
⑦	X_Z_;	For ⑥, it is the hole machining with the different Z values, and the hole position is only enabled moved along with X axis.
⑧	G89X_Y_;	The specified Z in ⑦, the R, P in ⑥ and the F in ② are regarded as the hole machining data, and perform the hole machining for G89 method.
⑨	G01X_Y_;	Eliminate the hole machining method and data

A. The absolute value code and incremental value code G90/G91 of canned cycle

The G90 and G91 are changed along with the distance of the drilling axis; refer to the Fig. **4-4-2.** (Generally, program is performed using G90; if the G91 is used, the Z and R will be treated based upon the +/- of the command)



(A) (B)

Fig. 4-4-2

B. The plane return code G98/G99 of canned cycle

Tool can be returned to the point R plane or initial position plane after the tool is arrived to the bottom of a hole. The tool can be returned to the initial point plane or point R plane based upon the different of G98 and G99.

Usually, the G99 is used for drilling at the 1st time and the G98 is at the last. The initial plane invarible even if the machining is performed by G99. The operations of G98 and G99 are shown below:

G98 is regarded as the system default.

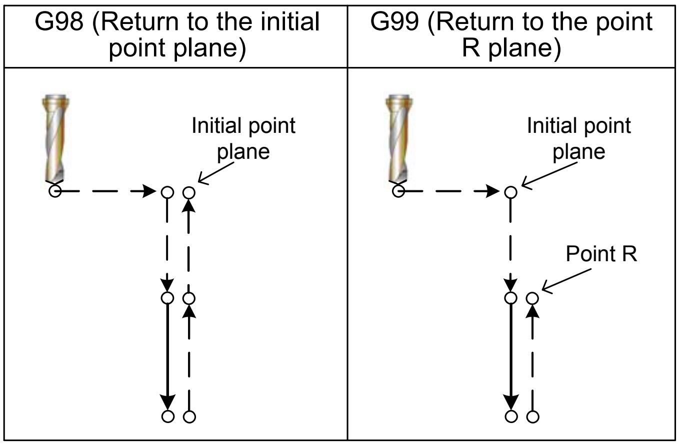


Fig. 4-4-3

The explanation symbols of each canned cycle, refer to the following figure.

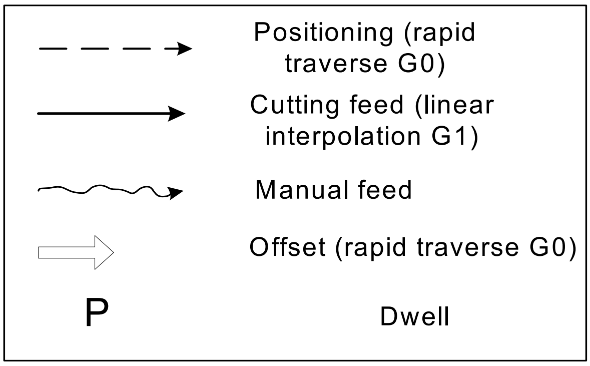


Fig. 4-4-4

The comparison table (G73～G89) of canned cycle

Table 4-4-3

G code	Drilling (-Z direction)	Operation at the bottom of a hole	Retraction (+ Z direction)	Application
G73	Intermittent feed		Rapid traverse	High-speed peck machining
G74	Feed	Dwell-spindle positive	Rapid traverse	Counter-tapping cycle
G76	Feed	Oriented spindle stop	Rapid traverse	Fine boring cycle
G80				Canned cycle cancel
G81	Feed		Rapid traverse	Drilling, spot drilling
G82	Feed	Dwell	Rapid traverse	Drilling, bore the stage hole
G83	Intermittent feed		Rapid traverse	Peck machining cycle
G84	Feed	Dwll → spindle negative	Feed	Tapping
G85	Feed		Feed	Boring
G86	Feed	Spindle stop	Rapid traverse	Boring
G87	Feed	Spindle positive	Rapid traverse	Boring
G88	Feed	Dwell → spindle stop	Manual → spindle positive	Boring
G89	Feed	Dwell	Feed	Boring

Restriction:

Tool radius offset (D) will be omitted during the canned cycle positioning.

4.4.1 High-speed Peck Machining Cycle G73

Code format: G73 X_Y_Z_R_Q_F_K_

Function: This cycle is specially set for performing the high-speed peck drilling, which is performed the intermittent cutting feed till to the bottom of a hole; simultaneously, it retreats from the hole at the rapid traverse during feed, and then eliminate the cutting chip. The operation schematic shows in the figure **4-4-1-1.**

Explanations:

X_Y_: Hole position data;

Z_: Incremental programming means specify the distance form point R to the bottom of a hole; absolute programming means the absolute coordinate value at the bottom of a hole;

R_: Incremental programming means the distance from the initial point plane to the point R; absolute programming means the absolute coordinate value of the point R;

Q_ Depth of cut for each cutting feed;

F_: Cutting feedrate

K_: Number of repeats.

G73（G98）	G73（G99）

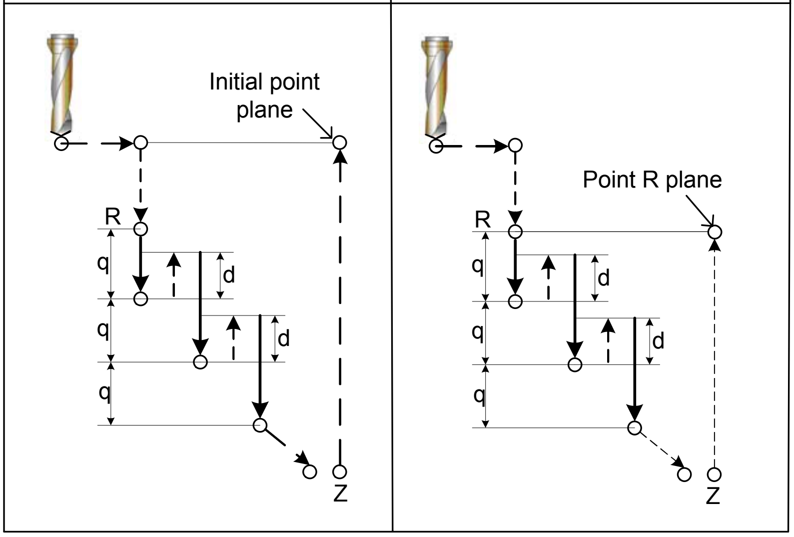


Fig. 4-4-1-1

Z, R: Parameters both the Z and R at the bottom of a hole will be simultaneously absented when performing the 1st drilling, and then the system executes the operation along Z axis.

Q: When the code parameter Q is specified, the above-mentioned intermittent feed is then performed. In this case, the system is retracted based upon the clearance d (Fig. 4-4-1-1) set in the data parameter **P270**. Each time for feed, the tool intermittently performs the retraction of that distance d at the rapid traverse rate.

When the G73 code 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: In the current version, M00, M01, M02, M06, M30, M98 and M99 are performed followed with the program; perform the above-mentioned M code after executing the current statement.

Note 2: When the bit 1 of parameter No.:43 is 0 and there is no cutting value in peck drilling (G73, G83), the alarm does not issue; in this case, fail to specify the code parameter Q or the Q is set to 0, the system performs the hole position at the X, Y plane instead of executing the drilling operation. When the bit 1 of parameter No.: 43 is 1, the alarm generates if there is no cutting value in peck drilling (G73, G83); that is, fail to specify the code parameter Q or the Q is set to 0; the system alarm prompts: "0045: fail to find address Q or the Q value is 0 (G73/G83)". If the Q value is specified as negative, the system then performs the intermittent feed based upon the absolute value.

Note 3: Tool length compensation: When the tool length compensation G43, G44 or G49 is commanded a same block with the canned cycle, add or cancel an offset value at the time of positioning to point R; in the canned cycle modal, if the tool compensation G43, G44 or G49 is placed at one block separately, the system is then performed the offset value addition or cancellation with real-time.

Restriction: Fail to specify the G code (From G00 to G03, G60 are the modal codes (bit 0 of parameter No.:48 is set to 1)) of group 01 in a same block when using G73 command, the system then performs the G60 modal.

Cutter compensation: In this canned cycle command, cutter compensation is ignored due to the command function is regardless of the cutter compensation.

Example:

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

Note: When the 2～6 hold machining are performed in the above-mentioned example; although the Q is omitted, the chip-removal operation still performs.

4.4.2 Drilling Cycle, Spot Drilling Cycle (G81)

Code format: G81 X_ Y_ Z_ R_ F_ K_

Function: This cycle is used for normal drilling cutting feed and performed to the bottom of the hole. The tool is then retreated from the bottom of the hole at the rapid traverse rate.

Explanations:

X_Y_: Hole position data;

Z_: Incremental program means specify the distance form point R to the bottom of a hole; absolute program means the coordinate value at the bottom of a hole;

R_: Incremental program means the distance from the initial point plane to the point R; absolute program means the coordinate value of the point R;

F_: Cutting feedrate

K_: Number of repeats (if required).

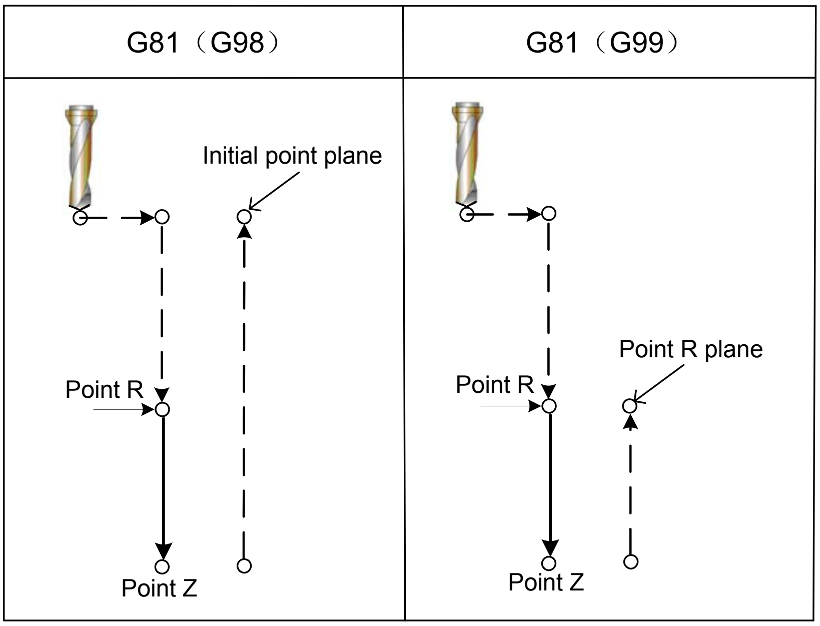


Fig. 4-4-2-1

Z, R: Either parameter Z or R at the bottom of a hole may be absent when performing the 1st drilling, the system changes the modal only instead of performing the operation along with Z axis.

Rapidly move to the point R after positioning along with the X and Y axes, perform the drilling machining from the point R to Z, and then tool retracts at the rapid traverse rate.

Rotate the spindle with the miscellaneous function M code before specifying the G81.

When the G81 code 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: In the current version, M00, M01, M02, M06, M30, M98 and M99 are performed followed with the program; perform the above-mentioned M code after executing the current statement.

Tool length compensation: When the tool length compensation G43, G44 or G49 is commanded a same block with the canned cycle, add or cancel an offset value at the time of positioning to point R; in the canned cycle modal, if the tool compensation G43, G44 or G49 is placed at one block separately, the system is then performed the offset value addition or cancellation with real-time.

Example:

M3 S2000	Spindle rotation
G90 G99 G81 X300 Y-250 Z-150 R-10 F120;	Positioning, drill hole 1, then return to point R
Y-550.;	Positioning, drill hole 2, then return to point R
Y-750.;	Positioning, drill hole 3, then return to point R
X1000.;	Positioning, drill hole 4, then return to point R
Y-550.;	Positioning, drill hole 5, then return to point R
G98 Y-750.;	Positioning, drill hole 5, then return to initial position plane
G80;	
G28 G91 X0 Y0 Z0 ;	Return to the reference position
M5;	Spindle rotation stop
M30;	

Restriction: Fail to specify the G code (From G00 to G03, G60 are the modal codes (bit 0 of parameter No.:48 is set to 1)) of group 01 in a same block when using G81 command, the system then performs the G60 modal.

Cutter compensation: In this canned cycle command, cutter compensation is ignored due to the command function is regardless of the cutter compensation.

4.4.3 Drilling Cydle, Counter Boring Cycle (G82)

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

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

Explanations:

X_Y_: Hole positioning data;

Z_: Incremental programming means the distance of the specified point R to the bottom of a hole; absolute programming means the absolute coordinate value of the bottom of a hole;

R_: Incremental programming means the distance from the initial point plane to point R; absolute programming means the absolute coordinate value of point R;

F_: Cutting feedrate;

P_: Dwell time at the bottom of a hole;

K_: Repeated times.

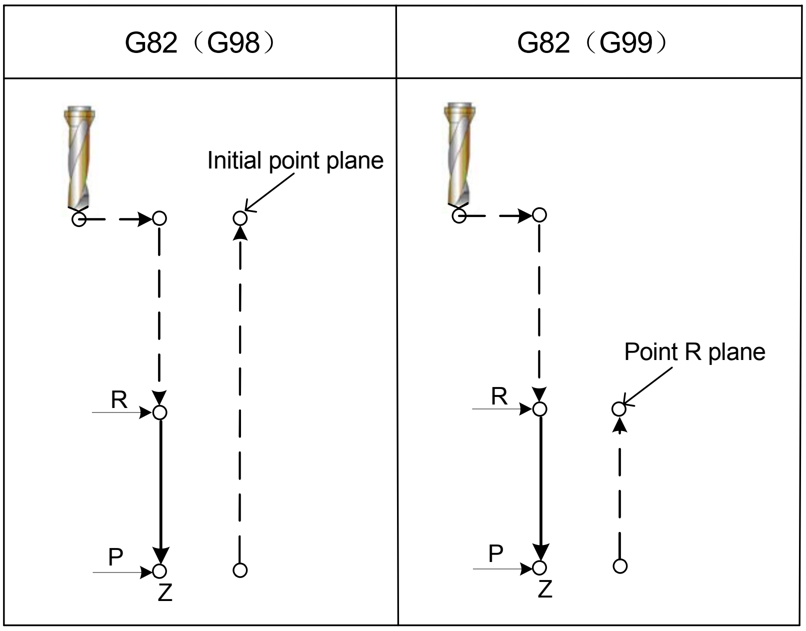


Fig. 4-4-3-1

After positioning along the X and Y axes, rapid traverse is performed to point R. Drilling is then performed 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.

The miscellaneous function M code rotates the spindle before specifying G82.

When the G82 and M codes are specified in the same block, the M code is executed at the time of the 1st positioning operation. The system then proceeds to the next drilling operation.

When the repeated times K is specified, the M code is executed for the 1st hole only instead of performing the second and subsequent holes.

Note: In the current version, M00, M01, M02, M06, M30, M98 and M99 are performed followed with the program; perform the above-mentioned M code after executing the current statement.

Tool length compensation: When the tool length compensation G43, G44 or G49 is commanded a same block with the canned cycle, add or cancel an offset value at the time of positioning to point R; in the canned cycle modal, if the tool compensation G43, G44 or G49 is placed at one block separately, the system is then performed the offset value addition or cancellation with real-time.

P is the modal code, the least value of parameter is set by data parameter P281 and the most value is set by p282. Value P is less than the P281 parameter setting value, which is operated based upon the least value; if it more than the P282 parameter setting value and it is operated based upon the most value.

Example:

M3 S2000 Spindle rotation

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

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

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

X1000.； Positioning, drill hole 4, dwell for 1s at thebottom of the hole, then return to the point R

Y-550； Positioning, drill hole 5, dwell for 1s at thebottom of the hole, then return to the point R

G98 Y-750； Positioning, drill hole 1, dwell for 1s at thebottom of the hole, then return to the initial position plane

G80； Canned cycle cancellation

G28 G91 X0 Y0 Z0 ； Reference position return

M5； Spindle rotation stop

M30；

Restriction: Fail to specify the G code (From G00 to G03, G60 are the modal codes (bit 0 of parameter No.:48 is set to 1)) of group 01 in a same block when using G81 command; otherwise, the G60 is replaced by G82.

Cutter compensation: In this canned cycle command, cutter compensation is ignored due to the command function is regardless of the cutter compensation.

4.4.4 Peck Drilling Cycle (G83)

Code 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.

Explanations:

X_Y_: Hole position data;

Z_: Incremental programming means the distance from the specified point R to the bottom of the hole; absolute programming means the absolute coordinate value of the bottom of the hole;

R_: Incremental programming means the distance from the initial point plane to the point R; absolute programming means the absolute coordinate value of the point R;

Q_: Cutting depth of each cutting feed;

F_: Cutting feedrate;

K_: Repeated times.

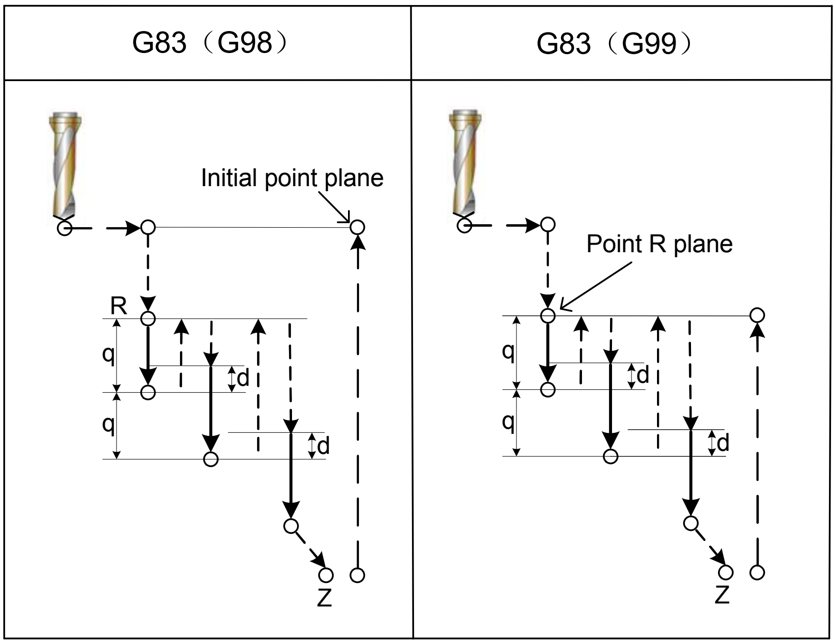


Fig. 4-4-4-1

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 point d just before where the last drilling ended, and the cutting feed is performed again. The value of d is set by parameter P271 as the Fig. **4-4-4-1** shows.

Be sure to specify a positive value in Q. Negative values are ignored and the system is still treated by positive.

Specify Q in the block of drilling, if it is not specified in the block without drilling; Q is registered as the modal data.

Use a miscellaneous function to rotate the spindle (M code) before specifying G83.

When G83 and M codes are specified in the same block, the M code is executed at the time of the 1st hole positioning operation. The system then proceeds to the next drilling operation.

When K is used to specify the repeated times, the M code is executed for the 1st hole only instead fo performing the subsequent holes.

Note 1: In the current version, M00, M01, M02, M06, M30, M98 and M99 are performed followed with the program; perform the above-mentioned M code after executing the current statement.

Note 2: When the bit 1 of parameter No.:43 equals to 0, there is no alarm without specifying the cutting value in peck drilling (G73, G83), in this case, the Q does not specify or sets to 0; the system performs the hole positioning along with X, Y plane instead of executing the drilling operation. When the bit 1 of parameter No.:43 equals to 1, the alarm occurs without specifying the cutting value in peck drilling (G73, G83); that is, the code parameter Q does not specified or sets to 0; the system alarms: "0045: the address Q does not find or sets to 0 (G73/G83)". If the Q value is specified as negative, the system then performs intermittent feed based upon its absolute value.

Tool length compensation: When the tool length compensation G43, G44 or G49 is commanded a same block with the canned cycle, add or cancel an offset value at the time of positioning to point R;

in the canned cycle modal, if the tool compensation G43, G44 or G49 is placed at one block separately, the system is then performed the offset value addition or cancellation with real-time.

Example:

M3 S2000；	Spindle rotation
G90 G99 G83 X300 Y-250 Z-150 R-100 Q15 F120;	Positioning, drill hole 1, then return to point R
Y-550；	Positioning, drill hole 2, then return to point R
Y-750；	Positioning, drill hole 3, then return to point R
X1000；	Positioning, drill hole 4, then return to point R
Y-550；	Positioning, drill hole 5, then return to point R
G98 Y-750；	Positioning, drill hole 5, then return to initial position plane
G80；	
G28 G91 X0 Y0 Z0 ；	Reference position return
M5；	Spindle rotation stop
M30；	

Restriction: Fail to specify the G code (From G00 to G03, G60 are the modal codes (bit 0 of parameter No.:48 is set to 1)) of group 01 in a same block when using G83 command, the system then performs the G60 modal.

Cutter compensation: In this canned cycle command, cutter compensation is ignored due to the command function is regardless of the cutter compensation.

4.4.5 Tapping Cycle G74 (or G84)

Code format: **G74/G84 X_ Y_ Z_ R_ P_ F_K_**

Function: This cycle performs tapping. In the tapping cycle, the dwell is performed when the tapping axis reaches to the bottom of the hole, then the spindle retracts the tapping axis with the negative rotation. (G74 is left-counter tapping cycle, G84 is right-counter tapping)

Explanations:

X_Y_: Hole positioning data;

Z_: Incremental programming means the distance from the specified point R to the bottom of the hole; absolute programming means the absolute coordinate value at the bottom of the hole;

R_: Incremental programming means the distance from the initial point plane to the point R; absolute programming means the absolute coordinate value of point R;

P_: Dwell time at the bottom of a hole;

F_: Tapping feedrate;

K_: Repeated times; (if required)

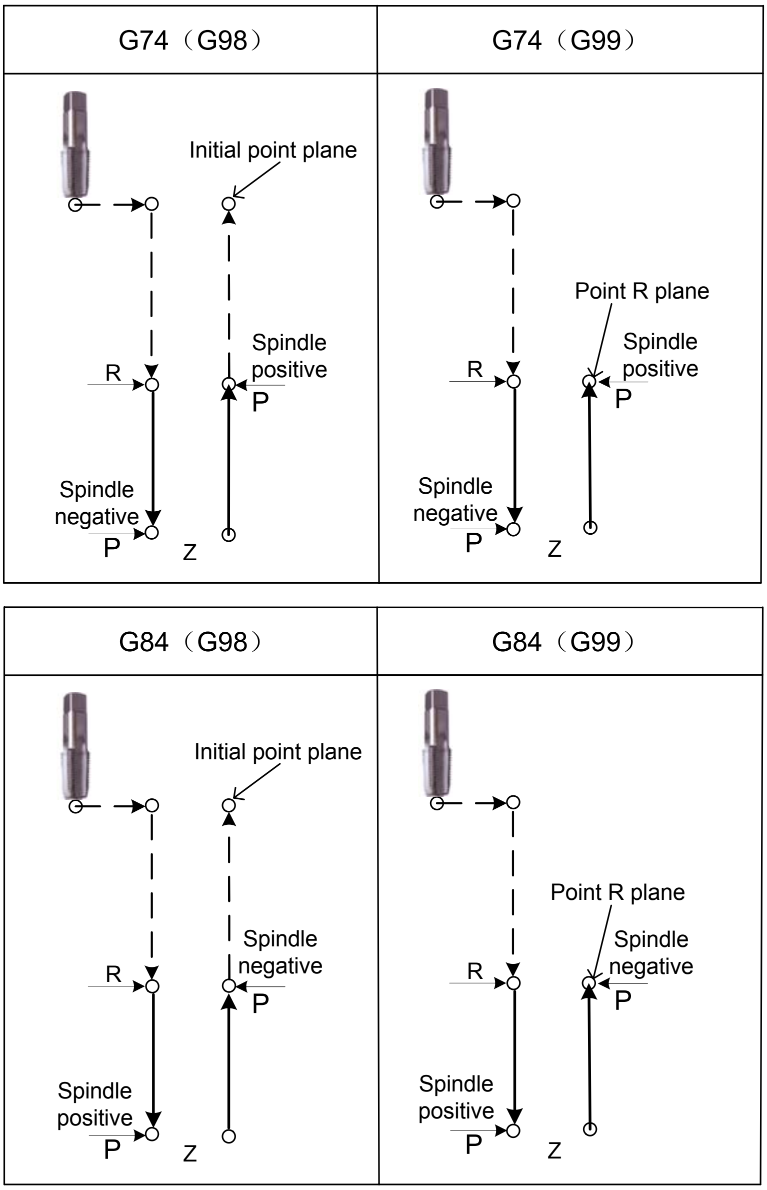


Fig. 4-4-5-1

When specifying the G74, spindle performs the tapping CW (When specifying G84, the spindle is CCW); the dwell is performed when it reaches to the bottom of the hole, then the spindle is rotated along with a reverse direction; simultaneously, the feedrate specified by program retracts the tapping axis, so that the screw occurs.

Example:

G94 Feed/min.;

M29 S1000 ； Spindle exact stop, specify the spindle speed;

G43 / G44 H10 ； Call the tool length compensation

G90 G99 G74 / G84 X100 Y110 Z -50 R5 P3000 F100； Positioning, tapping hole 1, then return topoint R;

Y150； Positioning, tapping hole 2, then return topoint R;

G91 X50 K5; It performs along with X axis based upon the reference point X100, Y150

50mm means that the incremental unit performs the tapping for 5 times;

G98 Y-750; Positioning, tapping hole 8, then return to the initial point;

G80; Cancel the tapping cycle;

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

M30; End of program;

Tool length compensation: When the tool length compensation G43, G44 or G49 is commanded a same block with the canned cycle, add or cancel an offset value at the time of positioning to point R; in the canned cycle modal, if the tool compensation G43, G44 or G49 is placed at one block separately, the system is then performed the offset value addition or cancellation with real-time.

Screw leading: In the feed/min. mode, the relationships among the screw leading, feedrate and spindle speed are shown below:

Feedrate F = screw tap pitch × spindle speed S

For example: tap the M12×1.5 screw hole in component, optional parameter;

S500=500 r /min; F=1.5×500=750mm/min;

In multiple thread, the F value can be calculated by multiplying by the numbers of thread head.

In feed/rev. mode, the thread leading equals to the feedrate.

Example:

Feed/min.:		Feed/rev.:	
Spindle speed 1000r/min;		Spindle speed 1000r/min;	
Thread leading 1.0mm;		Thread leading 1.0mm;	
Then, Z axis feedrate= 1000*1=1000mm/min;		Then, Z axis feedrate= thread leading =1mm / r;	
G94	Feed/min.	G95	Feed/rev.
G00 X120 Y100 ;	Positioning	G00 X120 Y100;	Positioning
M29 S1000 ;	Rigid method	M29 S1000 ;	Rigid method
G84 Z-100 R-20 F1000;	Right-counter rigid tapping	G84 Z-100 R-20 F1;	Right-counter rigid tapping
G80	Tapping cycle cancellation	G80	Tapping cycle cancellation
G28 G91 X0 Y0 Z0	Reference point return	G28 G91 X0 Y0 Z0	Reference point return
M30	End-of-program	M30	End-of-program

Restrictions:

G code: When using the G74/G84 command, fail to specify G code in 01 (G00 to G03, G60 is modal code (bit 0 of parameter No.:48 sets to 1)) group at the same block, the system then shifts to the G60 modal.

When the G74/G84 and M codes are specified at a same block, perform the M code while the 1st hole is performed the positioning operation; the next tapping operation is then disposed.

M code: To use the miscellaneous function M code rotates the spindle before specifying the G74/G84. If the spindle rotation does not specify, the system automatically rotates based upon the current spindle command speed in the R plane, then adjust as the CW (G74)/CCW (G84). When the G74/G84 and the M code are specified at a same block, perform the M code at the time of performing the 1st hole position operation. The system then proceeds next tapping.

When K is used to specify the repeated times, the M code is executed for the 1st hole only instead fo performing the subsequent holes.

Note: In the current version, M00, M01, M02, M06, M30, M98 and M99 are performed followed with the program; perform the above-mentioned M code after executing the current statement.

S command: If the specified spindle speed exceeds the top spindle speed in tapping (data parameter P257: the upper limit speed of spindle in tapping cycle), the system then alarms; the top speed level of the spindle in the rigid tapping is determined by P94~P296.

F command: If the specified F value exceeds the upper limit value of the cutting feedrate (data parameter: P96 sets the upper limit value), it is subject to the upper limit value.

P command: P is the modal code, the minimum value of parameter is set by data parameter **P281**; the maximum value of parameter is determined by data parameter **P282**. P is oprated based upon the minimum value when it is less than the parametr setting value of **P281** and it is operated based upon the maximum value when it is more than the parameter setting value of **P282**.

Axis shifting: Cancel the canned cycle before shifting the tapping axis. If the tapping axis is changed in rigid tapping mode, the system then shows the No.206 alarm.

Override: The feedrate and spindle rotation feedrate are 100% by default during the tapping; the machine tool still operates after pressing the feed hold button till the completion of the operation return.

Cutter compensation: In this canned cycle command, cutter compensation is ignored due to the command function is regardless of the cutter compensation.

Program restart: The program restart function is disabled in tapping cycle.

4.4.6 Fine Boring Cycle (G76)

Code format: G76 X_Y_Z_Q_R_P_F_K_

Function: The fine boring cycle is available for a hole with fine boring.

When reaching to the bottom of the hole, the spindle stops, and the tool is moved away from the machined surface of the workpiece and retracted.

To prevent the tool retraction path during retreating from impacting the smoothness on the machining surface, simultaneously, to avoid the damage of the tool.

Explanations:

X_Y_: Hole position data;

Z_: Incremental programming means the distance from the specified point R to the bottom of the hole; absolute programming means the absolute coordinate value at the bottom of the hole;

R_: Incremental programming means the distance from the initial point plane to the point R; absolute programming means the absolute coordinate value of point R;

Q_: Offset value at the bottom of a hoel

P_: Dwell time at the bottom of a hole;

F_: Cutting feedrate;

K_: Times of the fine boring.

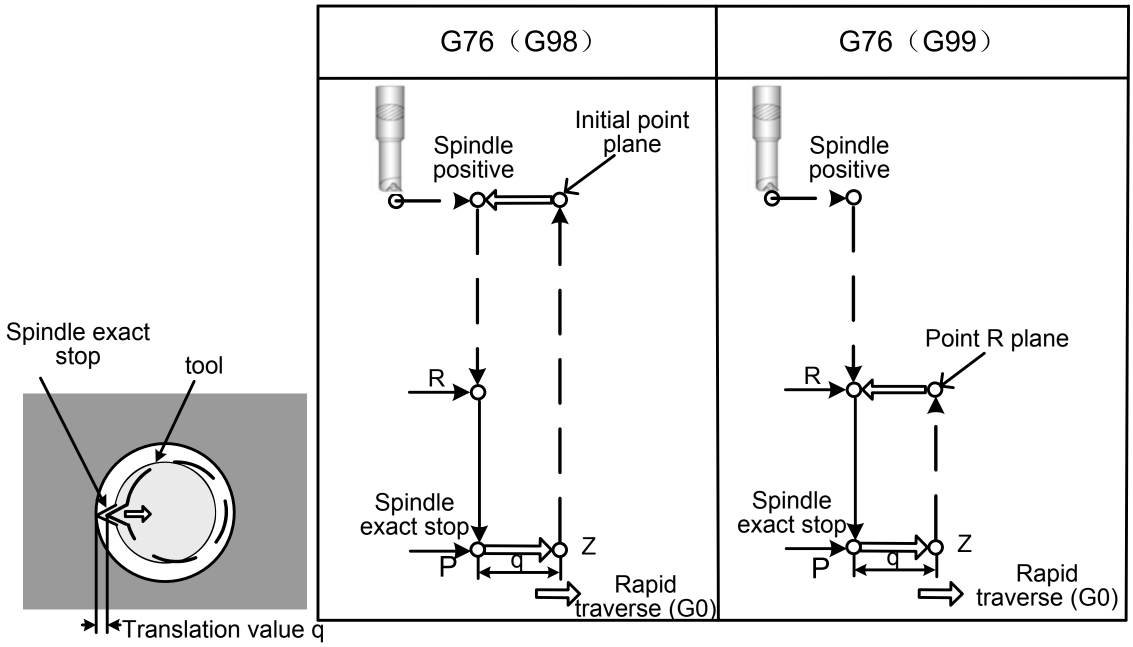


Fig. 4-4-6-1

The spindle stops on the canned revolving position when the tool reaches to the bottom of a hole, and tool moves and retracts based upon the reverse direction of tool nose. So that the machining surface will not be damage, and then the precise and enabled boring machining can be carried out. Parameter Q specifies the distance of tool-retraction. The tool-retraction direction and its axis can be specified by the bit 4 and 5 of parameter No.: 42; the Q value should be positive; and the sysmbol is still disabled even if using the negative. The offset value of the Q at the bottom of a hole is the registered modal value within the canned cycle, so it is important to specify it carefully. And therefore, it also uses for cutting depth of the G73 and G83.

To use the miscellandous function M code rotates the spindle before specifying the G76.

When the G74 and the M code are specified at a same block, perform the M code at the time of performing the 1st hole position operation. The system then proceeds the next operation.

When K is used to specify the repeated times, the M code is executed for the 1st hole only instead fo performing the subsequent holes.

Note: In the current version, M00, M01, M02, M06, M30, M98 and M99 are performed followed with the program; perform the above-mentioned M code after executing the current statement.

Tool length compensation: When the tool length compensation G43, G44 or G49 is commanded a same block with the canned cycle, add or cancel an offset value at the time of positioning to point R; in the canned cycle modal, if the tool compensation G43, G44 or G49 is placed at one block separately, the system is then performed the offset value addition or cancellation with real-time.

Axis shifting: Cancel the canned cycle before altering the drilling axis.

Boring machining: Fail to perform the boring maching in the block without X, Y and Z or othere axes.

Example:

M3 S500；Spindle rotation

G90 G99 G76 X300 Y-250 Z-150 R-100 Q5 P1000 F120; Positioning, bore hole 1, then return to point R, move 5mm when the orientation is performed at the bottom of ahole, and stops 1s.

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

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

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

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

G98 Y-750； Positioning, bore hole 6, and then return to the initial position plane

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

M5；Spindle stop

Restriction: Fail to specify the G code (From G00 to G03, G60 are the modal codes (bit 0 of parameter No.:48 is set to 1)) of group 01 in a same block when using G76 command, the system then performs the G60 modal.

Cutter compensation: In this canned cycle command, cutter compensation is ignored due to the command function is regardless of the cutter compensation.

Note 1: The in-feed axis and the in-feed direction are canned in this command, and the in-feed direction is regardless of the rotation of the G68 coordinate system.

4.4.7 Boring Cycle (G85)

Code format: G85 X_ Y_ Z_ R_ F_ K_

Function: This cycle is used to bore a hole.

Explanations:

X_Y_: Hole position data;

Z_: Incremental programming means the distance from the specified point R to the bottom of the hole; absolute programming means the absolute coordinate value at the bottom of the hole;

R_: Incremental programming means the distance from the initial point plane to the point R; absolute programming means the absolute coordinate value of point R;

F_: Cutting feedrate;

K_: Repeated times.

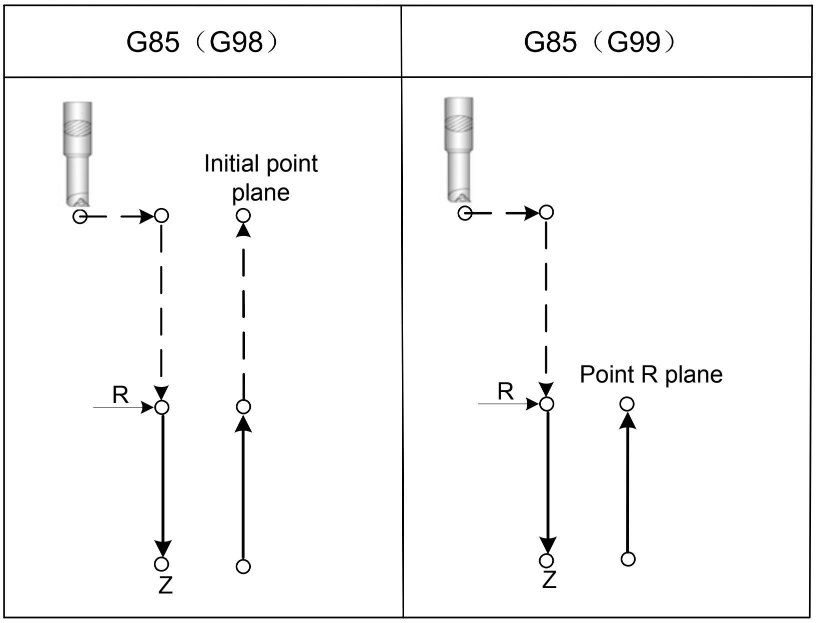


Fig. 4-4-7-1

Rapid traverse to point R after positioning along the X and Y axes, and then perform a boring from point R to Z; perform the cutting feed to return point R when reaching to the bottom of a hole.

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

When the G85 and the M code are specified at a same block, perform the M code at the time of performing the 1st hole position operation. The system then proceeds the next operation.

When K is used to specify the repeated times, the M code is executed for the 1st hole only instead fo performing the subsequent holes.

Note: In the current version, M00, M01, M02, M06, M30, M98 and M99 are performed followed with the program; perform the above-mentioned M code after executing the current statement.

Tool length compensation: When the tool length compensation G43, G44 or G49 is commanded a same block with the canned cycle, add or cancel an offset value at the time of positioning to point R; in the canned cycle modal, if the tool compensation G43, G44 or G49 is placed at one block

VOLUME I PROGRAMMING

separately, the system is then performed the offset value addition or cancellation with real-time.

Axis switching: Cancel the canned cycle before the drilling axis can be changed.

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

Example

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

Restriction: Fail to specify the G code (From G00 to G03, G60 are the modal codes (bit 0 of parameter No.:48 is set to 1)) of group 01 in a same block when using G85 command, the system then performs the G60 modal.

Cutter compensation: In this canned cycle command, cutter compensation is ignored due to the command function is regardless of the cutter compensation.

4.4.8 Boring Cycle (G85)

Code format: G86 X_ Y_ Z_ R_ F_ K_；

Function: This cycle code is used for boring machining cycle. (omit the dwell operation at the bottom of a hole)

Explanation:

X_Y_: Hole positioning data;

Z_: Incremental programming means the distance from the specified point R to the bottom of the hole; absolute programming means the absolute coordinate value at the bottom of the hole;

R_: Incremental programming means the distance from the initial point plane to the point R; absolute programming means the absolute coordinate value of point R;

F_: Cutting feedrate;

K_: Repeated machining times.

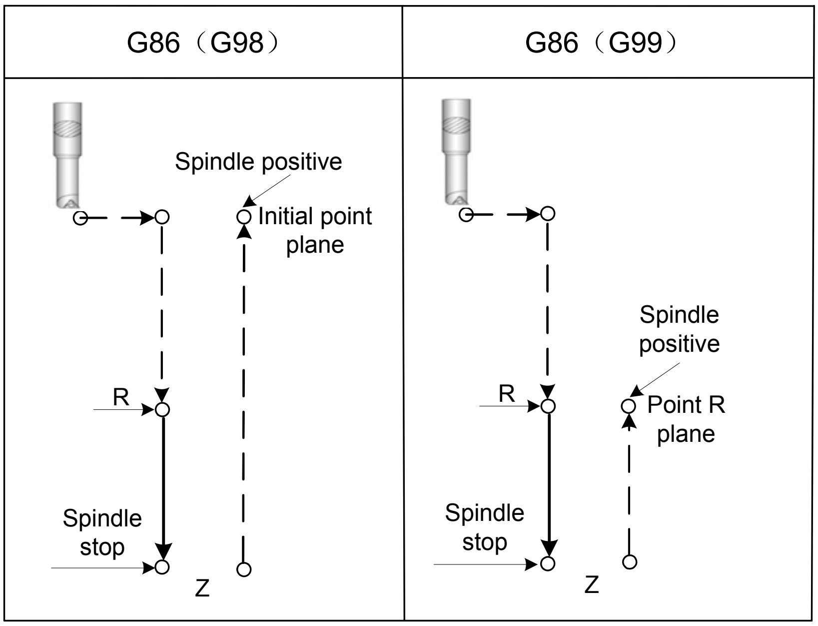


Fig. 4-4-8-1

Rapid traverse to point R after positioning along the X and Y axes, and then perform a boring from point R to Z; tool retracts at a rapid traverse rate when reaching to the bottom of a hole.

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

When the G86 and the M code are specified at a same block, perform the M code at the time of performing the 1st hole position operation, and then, the system then proceeds the next operation. When K is used to specify the repeated times, the M code is executed for the 1st hole only instead fo performing the subsequent holes.

Note: In the current version, M00, M01, M02, M06, M30, M98 and M99 are performed followed with the program; perform the above-mentioned M code after executing the current statement.

Tool length compensation: When the tool length compensation G43, G44 or G49 is commanded a same block with the canned cycle, add or cancel an offset value at the time of positioning to point R; in the canned cycle modal, if the tool compensation G43, G44 or G49 is placed at one block separately, the system is then performed the offset value addition or cancellation with real-time.

Axis switching: Cancel the canned cycle before the drilling axis can be changed.

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

For example:

M3 S2000； Spindle rotation

G90 G99 G86 X300 Y-250 Z-150 R-100 F120 Positioning, bore hole 1, then return to point R

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

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

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

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

G98 Y-750； Positioning, bore hole 5, then return to the initial position plane

G80；

G28 G91 X0 Y0 Z0 ； Return to the reference point

M5； Spindle rotation stop

M30；

Restriction: Fail to specify the G code (From G00 to G03, G60 are the modal codes (bit 0 of parameter No.:48 is set to 1)) of group 01 in a same block when using G86 command, the system then performs the G60 modal.

Cutter compensation: In this canned cycle command, cutter compensation is ignored due to the command function is regardless of the cutter compensation.

4.4.9 Hole Cycle, Back Boring Cycle (G87)

Code format: G87 X_Y_Z_R_Q_P_ F_；

Function: This cycle performs accurate boring

Explanation:

X_Y_: Hole positioning data;

Z_: Incremental programming means the distance from the specified point R to the bottom of the hole; absolute programming means the absolute coordinate value at the bottom of the hole;

R_: Incremental programming means the distance from the initial point plane to the point R; absolute programming means the absolute coordinate value of point R; (the bottom of the hole)

Q_: Offset value at the bottom of a hole

P_ Dwell time at the bottom of a hole;

F_: Cutting feedrate;

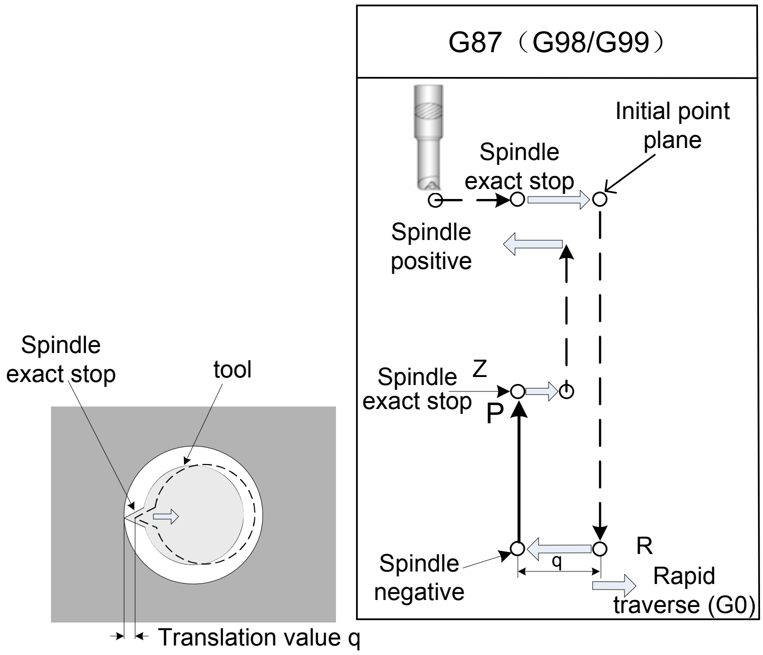


Fig. 4-4-9-1

Spindle stops the tool after orientation along with the X and Y axes and moves at the opposite direction along the tool nose, and movement is performed to the bottom of the hole (point R). The tool is then shofted in the direction of tool nose and the spindle is rotated negatively. Boring is performed in the positive direction along the Z axis until the point Z is reached. 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 CW to proceed to the next block operation.

Parameter Q specifies the distance of tool-retraction. The tool-retraction direction and its axis can be specified by the **bit 4 and 5 of parameter No.: 42**; the Q value should be positive; and the sysmbol is still disabled even if using the negative. The offset value of the Q at the bottom of a hole is the registered modal value within the canned cycle, so it is important to specify it carefully. And therefore, it also uses for cutting depth of the G73 and G83.

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

When the G87 and the M code are specified at a same block, perform the M code at the time of performing the 1st hole position operation, and then, the system then proceeds the next tapping.

When K is used to specify the repeated times, the M code is executed for the 1st hole only instead fo performing the subsequent holes.

Note: In the current version, M00, M01, M02, M06, M30, M98 and M99 are performed followed with the program; perform the above-mentioned M code after executing the current statement.

Tool length compensation: When the tool length compensation G43, G44 or G49 is commanded a same block with the canned cycle, add or cancel an offset value at the time of positioning to point R; in the canned cycle modal, if the tool compensation G43, G44 or G49 is placed at one block separately, the system is then performed the offset value addition or cancellation with real-time.

Canned cycle is only performed on the G17 plane.

Boring machining: Fail to perform the boring machining in the block without the X, Y, Z or other miscellaneous axis.

Prompt: When performing the back boring cycle, it is necessary to remember that the values Z and R should be specified. Generally, the position Z is above the position R; otherwise, the system alarm may occur.

Example:

M3 S500；Spindle rotation

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

(Positioning, bore hole 1, positioning at the initial position and then offset 5mm to stop at point Z for 1 second)

Y-550； Positioning, bore hole 2, then return to initial position plane

Y-750； Positioning, bore hole 3, then return to initial position plane

X1000； Positioning, bore hole 4, then return to initial position plane

Y-550； Positioning, bore hole 5, then return to initial position plane

G98 Y-750； Positioning, bore hole 6, then return to initial position plane

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

M5； Spindle rotation stop

M30；

Restriction: Fail to specify the G code (From G00 to G03, G60 are the modal codes (bit 0 of parameter No.:48 is set to 1)) of group 01 in a same block when using G87 command; otherwise, the G87 is replaced by G code of group 01.

Cutter compensation: In this canned cycle command, cutter compensation is ignored due to the command function is regardless of the cutter compensation.

Note 1: In this command, in-feed axis and direction are fixed, and the in-feed direction is regardless of the rotation of the G68 coordinate system.

4.4.10 Boring Cycle (G88)

Code format: G88 X_Y_Z_R_ P_F_

Function: This cycle is used for boring a hole.

Explanation:

X_Y_: Hole positioning data;

Z_: Incremental programming means the distance from the specified point R to the bottom of the hole; absolute programming means the absolute coordinate value at the bottom of the hole;

R_: Incremental programming means the distance from the initial point plane to the point R; absolute programming means the absolute coordinate value of point R;

P_ Dwell time at the bottom of the hole;

F_: Cutting feedrate.

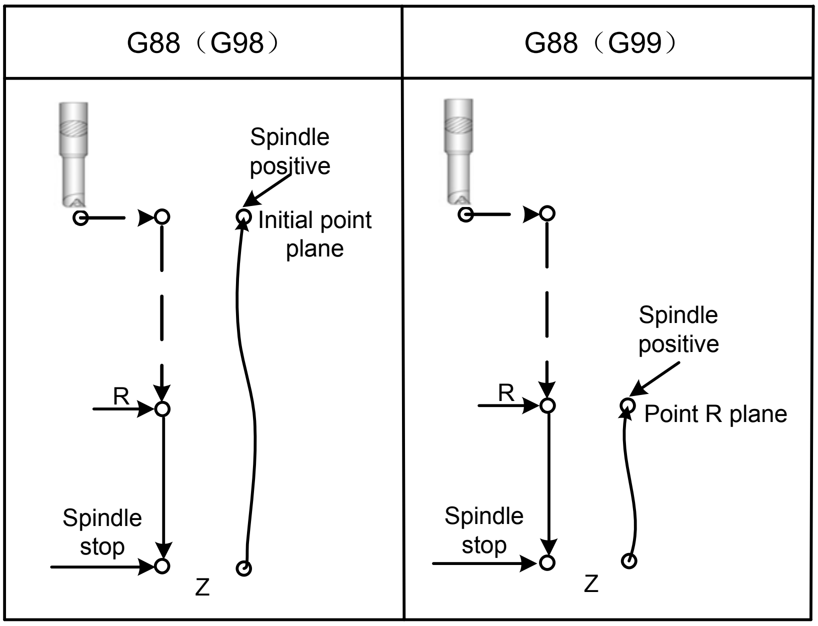


Fig. 4-4-10-1

Rapidly move to point R, and then the boring is performed from point R to Z after positioning is carried out along with the X and Y axes. Dwell is performed after the boring is executed, the the spindle stops; tool returns to the point R (G99) or initial point (G98) manually from the point Z where places at the bottom of the hole, and then spindle rotates positively.

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

When the G88 and the M code are specified at a same block, perform the M code at the time of performing the 1st hole position operation, and then, the system then proceeds the next drilling operation.

When K is used to specify the repeated times, the M code is executed for the 1st hole only instead fo performing the subsequent holes.

Note: In the current version, M00, M01, M02, M06, M30, M98 and M99 are performed followed with the program; perform the above-mentioned M code after executing the current statement.

P is the modal code, the least value of parameter is set by data parameter P281 and the most value is set by p282. Value P is less than the P281 parameter setting value, which is operated based upon the least value; if it more than the P282 parameter setting value and it is operated based upon the most value.

Tool length compensation: When the tool length compensation G43, G44 or G49 is commanded a same block with the canned cycle, add or cancel an offset value at the time of positioning to point R; in the canned cycle modal, if the tool compensation G43, G44 or G49 is placed at one block separately, the system is then performed the offset value addition or cancellation with real-time.

Axis switching: Cancel the canned cycle before the boring axis can be shifted.

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

Example:

M3 S2000; Spindle rotation

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

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

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

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

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

G98 Y-750; Positioning, bore hole 6, then return to initial position plane

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

M5; Spindle rotation stop

Restriction: Fail to specify the G code (From G00 to G03, G60 are the modal codes (bit 0 of parameter No.:48 is set to 1)) of group 01 in a same block when using G88 command, the system then performs the G60 modal.

Cutter compensation: In this canned cycle command, cutter compensation is ignored due to the command function is regardless of the cutter compensation.

4.4.11 Boring Cycle (G89)

Code format: **G89 X_ Y_ Z_ R_ P_ F_ K_**

Function: This cycle is used to bore a hole

Explanation:

X_Y_: Hole positioning data;

Z_: Incremental programming means the distance from the specified point R to the bottom of the hole; absolute programming means the absolute coordinate value at the bottom of the hole;

R_: Incremental programming means the distance from the initial point plane to the point R; absolute programming means the absolute coordinate value of point R;

P_ Dwell time at the bottom of a hole;

F_: Cutting feedrate;

K_: Repeated times.

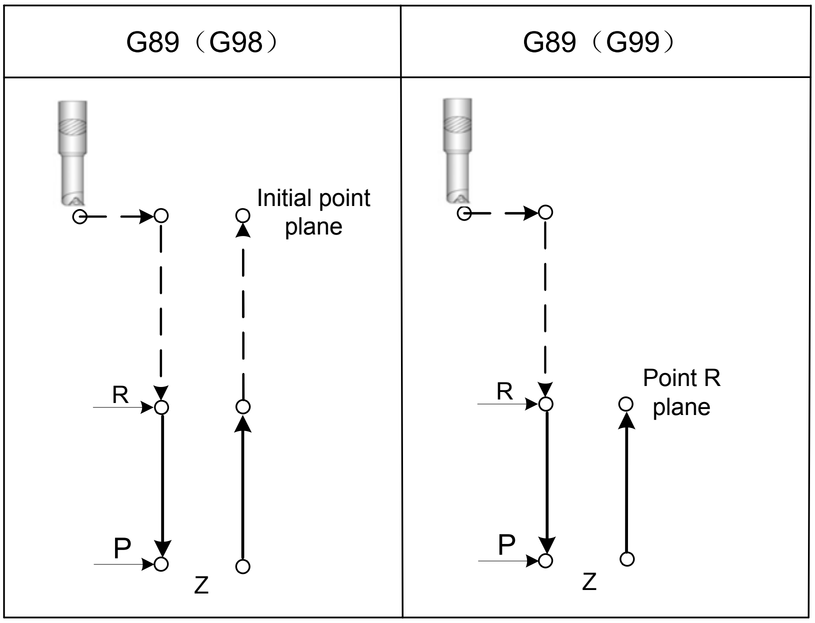


Fig. 4-4-11-1

This cycle, almost, is absolute indentical with the G85. The different is that this cycle performs a dwell at the bottom of the hole.

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

When the G89 and the M code are specified at a same block, perform the M code at the time of performing the 1st hole position operation, and then, the system then proceeds the next drilling operation.

When K is used to specify the repeated times, the M code is executed for the 1st hole only instead fo performing the subsequent holes.

Note: In the current version, M00, M01, M02, M06, M30, M98 and M99 are performed followed with the program; perform the above-mentioned M code after executing the current statement.

P is the modal code, the least value of parameter is set by data parameter P281 and the most value is set by p282. Value P is less than the P281 parameter setting value, which is operated based upon the least value; if it more than the P282 parameter setting value and it is operated based upon the most value.

Tool length compensation: When the tool length compensation G43, G44 or G49 is commanded a same block with the canned cycle, add or cancel an offset value at the time of positioning to point R; in the canned cycle modal, if the tool compensation G43, G44 or G49 is placed at one block separately, the system is then performed the offset value addition or cancellation with real-time.

Axis switching: Cancel the canned cycle before the boring axis can be shifted.

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

Example:

M3 S100； Spindle rotation

G90 G99 G89 X300 Y-250 Z-150 R-120 P1000 F120； Positioning, bore hole 1, then return to

	point R then stop that the bottom of the hole for 1s.
Y-550；	Positioning, bore hole 2, then return to point R
Y-750；	Positioning, bore hole 3, then return to point R
X1000；	Positioning, bore hole 4, then return to point R
Y-550；	Positioning, bore hole 5, then return to point R
G98 Y-750；	Positioning, bore hole 6, then return to initial positon plane
G80；	
G28 G91 X0 Y0 Z0 ；	Return to the reference position
M5；	Spindle rotation stop
M30；	

Restriction: Fail to specify the G code (From G00 to G03, G60 are the modal codes (bit 0 of parameter No.:48 is set to 1)) of group 01 in a same block when using G89 command, the system then performs the G60 modal.

Cutter compensation: In this canned cycle command, cutter compensation is ignored due to the command function is regardless of the cutter compensation.

4.5 Rigid Cycle G Code

4.5.1 Left-handed Rigid Tapping Cycle G74

Code format: G74 X_Y_Z_R_P_F_K_

Function: In the rigid method, the working of spindle motor is a servo motor, and this code can be carried out the left-handed tapping with high speed and high accuracy.

Explanation:

X_Y_: Hole position data

Z_: Incremental programming means the distance from the specified point R to the bottom of the hole; absolute programming means the absolute coordinate value at the bottom of the hole;

R_: Incremental programming means the distance from the initial point plane to the point R; absolute programming means the absolute coordinate value of point R;

P_: The dwell time at the bottom of a hole or the dwell time at the point R.

F_: Cutting feedrate.

K_: Repeated times. (If required)

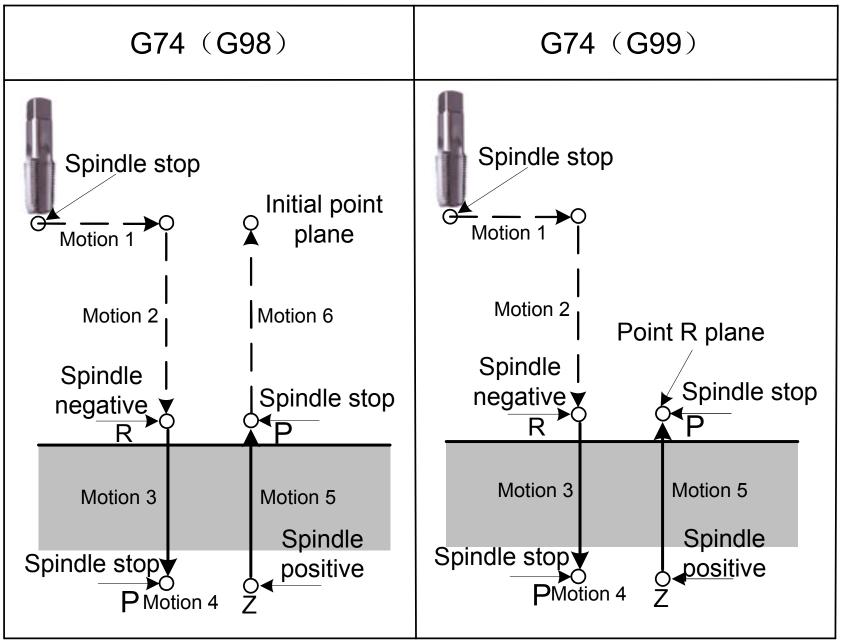


Fig. 4-5-1-1

Z axis moves to point R at the rapid traverse and performs G74 along with X and Y axes after positioning, and then spindle reverse. The tapping is performed from point R to Z, when the tapping is completed, the spindle stops and performs the dwell, then spindle rotates tool retreating from point R with reverse direction, spindle then stops; consequently, spindle moves to the initial position at a rapid traverse rate. The feedrate and spindle override are regarded as 100% when tapping is being performed.

Rigid method: In the position mode (bit 1 of parameter No.:46 is set to 1, K parameter No. :7#7 sets to 1), to perform M29 S***** before the tapping code can be specified the rigid method.

Tool length compensation: When the tool length compensation G43, G44 or G49 is commanded a same block with the canned cycle, add or cancel an offset value at the time of positioning to point R; in the canned cycle modal, if the tool compensation G43, G44 or G49 is placed at one block separately, the system is then performed the offset value addition or cancellation with real-time.

Thread leading: In the feed/min. method, the relationships among the thread leadind, feedrate and spindle speed as follows:

Feedrate F = screw tap pitch × spindle speed S

For example: Tap a thread hole M12×1.5 on component, the parameter can be optioned;

S500=500 r /min； F=1.5×500=750mm/min；

Multi-head thread x head numbers = F value

In the feed method per revolution, thread leading equals to feedrate.

For example:

Feed/min.:		Fee/rev.:	
Spindle speed 1000r/min;		Spindle speed 1000r/min;	
Thread leading 1.0mm;		Thread leading 1.0mm;	
Then feedrate of Z axis= 1000*1=1000mm/min;		Then feedrate of Z axis=thread leading 1=1mm / r;	
G94	Feed/min.	G95	Feed/rev.
G00 X120 Y100;	Positioning	G00 X120 Y100;	Positioning
M29 S1000 ;	Specify a rigid tapping	M29 S1000 ;	Specify a rigid tapping
G74 Z-100 R-20 F1000;	Left-handed rigid tapping	G74 Z-100 R-20 F1;	Left-handed rigid tapping
G80	Cancel the tapping cycle	G80	Cancel the tapping cycle
G28 G91 X0 Y0 Z0	Reference position return	G28 G91 X0 Y0 Z0	Reference position return
M30	End-of-program	M30	End-of-program

Restriction:

G code: Fail to specify the G code (From G00 to G03, G60 are the modal codes (bit 0 of parameter No.:48 is set to 1)) of group 01 in a same block when using G74 command, the system then performs the G60 modal.

M code: Spindle is rotated by miscellaneous function M code before specifying G74. If there is no specified the spindle rotation, the system then adjusts as CW based upon the current spindle command speed automatically.

When the G74 code 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 repeated times, the M code is executed for the 1st hole only instead fo performing the subsequent holes.

Note: In the current version, M00, M01, M02, M06, M30, M98 and M99 are performed followed with the program; perform the above-mentioned M code after executing the current statement.

S command: If the specified spindle speed exceeds the top spindle speed in tapping (data parameter P257: the upper limit speed of spindle in tapping cycle), the system then alarms; the top speed level of the spindle in the rigid tapping is determined by P94~P296.

F command: If the specified F value exceeds the upper limit value of the cutting feedrate (data parameter: P96 sets the upper limit value of cutting feed), it is subject to the upper limit value.

P command: P is the modal code, the least value of parameter is set by data parameter P281 and the most value is set by p282. Value P is less than the P281 parameter setting value, which is operated based upon the least value; if it more than the P282 parameter setting value and it is operated based upon the most value.

Axis shifting: Cancel the canned cycle before shifting the tapping axis. If the tapping axis is changed in rigid tapping mode, the system then shows the No.206 alarm.

Override: The feedrate and spindle rotation feedrate are 100% by default during the rigid tapping; the machine tool still operates after pressing the feed hold and reset buttons till the completion of the operation return.

Cutter compensation: In this canned cycle command, cutter compensation is ignored due to the command function is regardless of the cutter compensation.

Program restart: The program restart function is disabled in tapping cycle.

Note: When the machining, such as the flexible tapping, rigid tapping or peck tapping, etc. are performed, it is necessary to firstly cancel the constant surface cutting feed by G97; otherwise, the disorder gear or broken screw tap, etc. will occur.

4.5.2 Right-handed Rigid Tapping G84

Format: **G84 X_Y_Z_R_P_F_K_**

Function: In the rigid method, the control of spindle motor is a servo motor, which can be carried out the high-speed accuracy tapping. Also, it can be guarantee that the start position of tapping is consistent in the case of invariable Point R. Namely, the thread still keep order even if repeated perform the tapping in a position.

Explanation:

X_Y_: Hole positioning data;

Z_: Incremental programming means the distance from the specified point R to the bottom of the hole; absolute programming means the absolute coordinate value at the bottom of the hole;

R_: Incremental programming means the distance from the initial point plane to the point R; absolute programming means the absolute coordinate value of point R;

P_: The dwell time at the bottom of a hole or the dwell time at point R when retracting;

F_: Cutting feedrate;

K_: Repeated times. (If required)

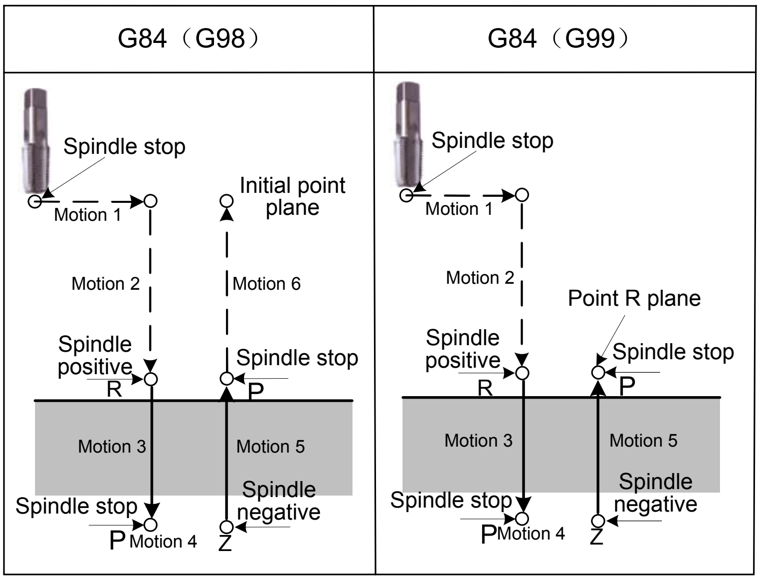


Fig. 4-5-2-1

Z axis moves to point R at the rapid traverse and performs G84 along with X and Y axes after positioning, and then spindle reverse. The tapping is performed from point R to Z, when the tapping is completed, the spindle stops and performs the dwell, then spindle rotates tool retreating from point R with reverse direction, spindle then stops; consequently, spindle moves to the initial position at a rapid traverse rate. The feedrate and spindle override are regarded as 100% when tapping is being performed.

Rigid method: In the position mode (bit 1 of parameter No.:46 is set to 1, K parameter No. :7#7 sets to 1), to perform M29 S***** before the tapping code can be specified the rigid method.

Tool length compensation: When the tool length compensation G43, G44 or G49 is commanded a same block with the canned cycle, add or cancel an offset value at the time of positioning to point R; in the canned cycle modal, if the tool compensation G43, G44 or G49 is placed at one block separately, the system is then performed the offset value addition or cancellation with real-time.

Screw leading: In the feed/min. mode, the relationships among the screw leading, feedrate and spindle speed are shown below:

Feedrate F = screw tap pitch × spindle speed S

For example: tap the M12×1.5 screw hole in component, optional parameter;

S500=500 r /min；　　F=1.5×500=750mm/min；

In multiple thread, the F value can be calculated by multiplying by the numbers of thread head.

In feed/rev. mode, the thread leading equals to the feedrate.

For example:

Feed/min.:		Fee/rev.:	
Spindle speed 1000r/min；		Spindle speed 1000r/min；	
Thread leading 1.0mm；		Thread leading 1.0mm；	
Then feedrate of Z axis= 1000*1=1000mm/min;		Then feedrate of Z axis=thread leading 1=1mm / r；	
G94	Feed/min.	G95	Feed/rev.
G00 X120 Y100；	Positioning	G00 X120 Y100；	Positioning
M29 S1000 ；	Specify a rigid tapping	M29 S1000 ；	Specify a rigid tapping
G84 Z-100 R-20 F1000;	Left-handed rigid tapping	G84 Z-100 R-20 F1;	Left-handed rigid tapping
G80	Cancel the tapping cycle	G80	Cancel the tapping cycle
G28 G91 X0 Y0 Z0	Reference position return	G28 G91 X0 Y0 Z0	Reference position return
M30	End-of-program	M30	End-of-program

Restrictions:

G code: Fail to specify the G code (From G00 to G03, G60 are the modal codes (bit 0 of parameter No.:48 is set to 1)) of group 01 in a same block when using G84 command, the system then performs the G60 modal.

M code: Spindle is rotated by miscellaneous function M code before specifying G84. If there is no specified the spindle rotation, the system then adjusts as CW based upon the current spindle command speed automatically.

When the G84 and an M codes 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 repeated times, the M code is executed for the 1st hole only instead fo performing the subsequent holes.

Note: In the current version, M00, M01, M02, M06, M30, M98 and M99 are performed followed with the program; perform the above-mentioned M code after executing the current statement.

S command: If the specified spindle speed exceeds the top spindle speed in tapping (data parameter P257: the upper limit speed of spindle in tapping cycle), the system then alarms; the top speed level of the spindle in the rigid tapping is determined by P94~P296.

F command: If the specified F value exceeds the upper limit value of the cutting feedrate (data parameter: P96 sets the upper limit value), it is subject to the upper limit value.

P command: P is the modal code, the least value of parameter is set by data parameter P281 and the most value is set by p282. Value P is less than the P281 parameter setting value, which is operated based upon the least value; if it more than the P282 parameter setting value and it is operated based upon the most value.

Axis shifting: Cancel the canned cycle before shifting the tapping axis. If the tapping axis is changed in rigid tapping mode, the system then shows the No.206 alarm.

Override: The feedrate and spindle rotation feedrate are 100% by default during the tapping; the machine tool still operates after pressing the feed hold and reset buttons till the completion of the operation return.

Cutter compensation: In this canned cycle command, cutter compensation is ignored due to the command function is regardless of the cutter compensation.

Program restart: The program restart function is disabled in tapping cycle.

Note: When the machining, such as the flexible tapping, rigid tapping or peck tapping, etc. are performed, it is necessary to firstly cancel the constant surface cutting feed by G97; otherwise, the disorder gear or broken screw tap, etc. will occur.

4.5.3 Peck Tapping (Chip-removal) Cycle

Code format: G84（or G74） X_Y_Z_R_P_Q_F_K_

Function: In the peck tapping, cutting is performed for several times until the bottom of the hole.

Explanation:

X_Y_: Hole positioning data;

Z_: Incremental programming means the distance from the specified point R to the bottom of the hole; absolute programming means the absolute coordinate value at the bottom of the hole;

R_: Incremental programming means the distance from the initial point plane to the point R; absolute programming means the absolute coordinate value of point R;

P_: The dwell time at the bottom of a hole or at point R when retracting;

Q_: The cutting depth of cutting feed each time;

F_: Cutting feedrate;

V_: Retraction distance d; if it does not specify, which is set by parameter P284.

K_: Repeated times. (If required)

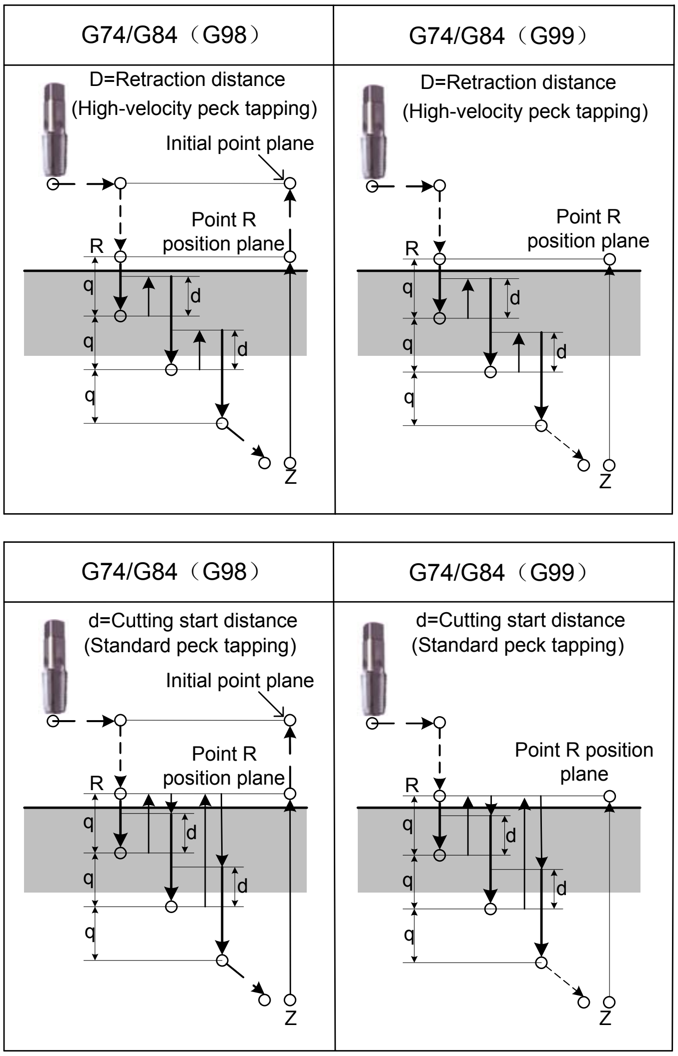


Fig. 4-5-3-1

Table 4-5-3-1

Peck tapping cycle	Parameter setting	Adoptive tapping method
Peck flexible tapping cycle	NO:46#1=0 and NO:K007#7=0	It is high-speed peck tapping cycle when NO:44#5=1. It is standard peck tapping cycle when NO:44#5=0.
Peck rigid tapping cycle	NO:46#1=1 and NO:K007#7=1	It is high-speed peck tapping cycle when NO:44#5=1. It is standard peck tapping cycle when NO:44#5=0.

Two peck tapping cycles are available: High-speed peck tapping cycle and standard peck tapping cycle, which are determined by **bit 1 of parameter No.: 46.**

Peck flexible tapping cycle:

When **bit 1 of parameter No.: 46 equals to 0** and **bit 7 of parameter No.: K007 equals to 0,** which are the flexible peck tapping cycle. In the peck flexible tapping method, the high-speed peck tapping cycle and standard peck flexible tapping cycle are available, which can be set by **bit 5 of parameter No.: 44** .

High-speed peck tapping cycle:

It is the high-speed peck tapping cycle when **bit 5 of parameter No.: 44 sets to 1**: rapid traverse to point R after positioning along with the X and Y axes. From point R, cutting is performed with depth Q (depth of cut for each cutting feed) , then the tool is retracted by distance d (It is specified by canned parameter V, if it does not specify, it then sets by data parameter P284), which is set the rigid tapping tool-retraction by the bit 4 of parameter No.: 44, the override is enabled, is specified retraction speed override by bit 3 of parameter No.: 45, is set the rigid tapping in-feed, tool-retraction whether using the same time constant by bit 2 of parameter No.: 45, is set whether the feedrate selection and override cancellantion signal are enabled in the rigid tapping by bit 4 of No.: 45. When point Z has been reached, the spindle is stopped, then rotated in the reverse direction for retration.

Standard peck (flexible) tapping cycle:

It is the standard peck tapping cycle when **bit 5 of parameter No.: 44 sets to 0:** rapid traverse to point R after positioning along with the X and Y axes. From point R, cutting is performed with depth Q (depth of cut for each cutting feed), and then return to point R; whether the overrider in rigid tapping is enabled which is set by bit 4 of parameter No.44; specify the retraction feedrate by bit 3 of parameter No.:45; the distance d (it is set by data parameter P284) from the point R to the end of cutting, then perform the cutting again based upon the value of the cutting feedrate F; whether using the same time constant in the in-feed, tool-retraction, which is set by bit 2 of parametr No.45. When point Z has been reached, the spindle is stopped, then rotated in the reverse direction for retration.

Standard peck (rigid) tapping cycle:

In the position mode (bit 1 of parameter No.:46 is set to 1, K parameter No. :7#7 sets to 1), specify M29 S***** as the peck rigid tapping cycle before the tapping code; here, use the standard peck tapping cycle method, and its setting method is indentical with the flexible standard peck tapping one.

Tool length compensation: When the tool length compensation G43, G44 or G49 is commanded a same block with the canned cycle, add or cancel an offset value at the time of positioning to point R; in the canned cycle modal, if the tool compensation G43, G44 or G49 is placed at one block separately, the system is then performed the offset value addition or cancellation with real-time.

Restrictions:

G code: Fail to specify the G code (From G00 to G03, G60 are the modal codes (bit 0 of parameter No.:48 is set to 1)) of group 01 in a same block when using G74/G84 command, the system then performs the G60 modal.

M code: Use the miscellaneous function M code to rotat the spindle before specifying the G74/G84. if the spindle rotation does not perform, the system is then automatically rotated based upon the current spindle in R plane, and then adjust to the CCW.

When the G74/G84 and an M codes 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 repeated times, the M code is executed for the 1st hole only instead fo performing the subsequent holes.

(In the current version, M00, M01, M02, M06, M30, M98 and M99 are performed followed with the program; perform the above-mentioned M code after executing the current statement.)

S command: If the specified spindle speed exceeds the top spindle speed in tapping (data parameter P257: the upper limit speed of spindle in tapping cycle), the system then alarms; the top speed level of the spindle in the rigid tapping is determined by P94~P296.

F command: If the specified F value exceeds the upper limit value of the cutting feedrate (data parameter: P96 sets the upper limit value), it is subject to the upper limit value.

P command: P is the modal code, the least value of parameter is set by data parameter P281 and the most value is set by p282. Value P is less than the P281 parameter setting value, which is operated based upon the least value; if it more than the P282 parameter setting value and it is operated based upon the most value.

Axis shifting: Cancel the canned cycle before shifting the tapping axis. If the tapping axis is changed in rigid tapping mode, the system then shows the No.206 alarm.

Override: The feedrate and spindle rotation feedrate are 100% by default during the tapping; the machine tool still operates after pressing the feed hold and reset buttons till the completion of the operation return.

Cutter compensation: In this canned cycle command, cutter compensation is ignored due to the command function is regardless of the cutter compensation.

Program restart: The program restart function is disabled in tapping cycle.

Note: When the machining, such as the flexible tapping, rigid tapping or peck tapping, etc. are performed, it is necessary to firstly cancel the constant surface cutting feed by G97; otherwise, the disorder gear or broken screw tap, etc. will occur.

4.6 Compound Cycle G Code

Compound comparison table (G22～G38)

Table 4-6-1

G code	Drilling (-Z direction)	Operation at the bottom of a hole	Tool-retraction operation (-Z direction)	Purpose
G22	Cutting feed		Rapid traverse	Groove rough-milling of inner circle along CCW
G23	Cutting feed		Rapid traverse	Groove rough-milling of inner circle along CW
G24	Cutting feed		Rapid traverse	Fine-milling cycle of full circle along CCW
G25	Cutting feed		Rapid traverse	Fine-milling cycle of full circle along CW
G26	Cutting feed		Rapid traverse	Excircle fine-milling cycle along CCW
G32	Cutting feed		Rapid traverse	Excircle fine-milling cycle along CW
G33	Cutting feed		Rapid traverse	Rectangluar groove rough-milling along CCW
G34	Cutting feed		Rapid traverse	Rectangluar groove rough-milling along CW
G35	Cutting feed		Rapid traverse	Fine-milling cycle of the inner rectangular groove along CCW
G36	Cutting feed		Rapid traverse	Fine-milling cycle of the inner rectangular groove along CW
G37	Cutting feed		Rapid traverse	Fine-milling cycle ex-rectangular CCW
G38	Cutting feed		Rapid traverse	Fine-milling cycle ex-rectangular CW

Restriction:

The tool radius offset (D) will be ignored during the canned cycle position.

4.6.1 Groove Rough-Milling of Inner Circle (G22/G23)

Code format:

G22

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

G23

Function: The multiple circular arc interpolation can be performed by helical method from the beginning of the center till the round groove is machined by the program dimension.

Explanations:

G22: Groove rough-milling of inner circle along CCW;

G23: Groove rough-milling of inner circle along CW;

X, Y: The start position of X, Y plane;

Z: Machining depth, its an absolute position in G90; it is a position related to the R reference surface in G91;

R: R reference surface position, it is absolute position in G90, the start position related to this block in G91;

I: Incircle groove radius;

L: The width increment by cutting within XY plane.

W: Cutting depth along with the Z axis at the 1st time, it is downward from R reference surface, which should be more than 0 (If the cutting depth is more than the position at the bottom of the groove, then it is machined directly based upon the groove position);

Q: Cutting depth of each cutting feed;

V: The distance from the unprocessed surface during cutting at the rapid traverse rate;

D: Tool compensation number, take out the corresponding tool compensation value based upon the provided series number;

K_: Repeated times.

Cycle processes:

(1) Rapid positioning to the specified point (X, Y), offset a tool radius D along with the X axis negative direction, and then multiple the cutting coefficient position of the helix;

(2) Move to point R plane at the rapid traverse rate;

(3) Downward the cutting W distance depth with helical method at the rapid traverse rate → feed to the circle center

(4) The circle surface with radius I is milled with helix step by step by L value for each time where from center to outside;

(5) Z axis returns to R reference surface at the rapid traverse rate;

(6) Rapid positioning to the start position along with the X and Y axes;

(7) Z axis descends to the distance from the unproceeded surface V at the rapid traverse rate;

(8) Cutting (Q+V) depth downward along Z axis;

VOLUME I PROGRAMMING

(9) Repeat the operations from (4) to (8) till to the total cutting depth on the surface is machined;

(10) Return to initial point plane or point R plane based upon the different specifications of G98 or G99.

Code path:

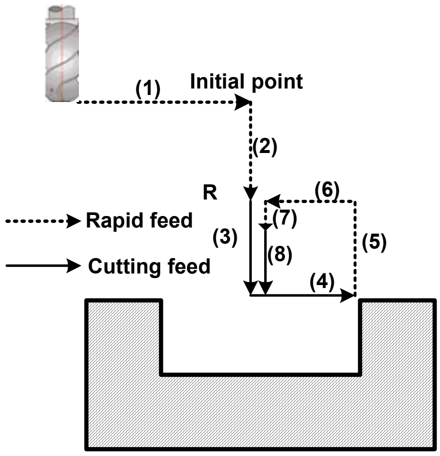


Fig. 4-6-1-1

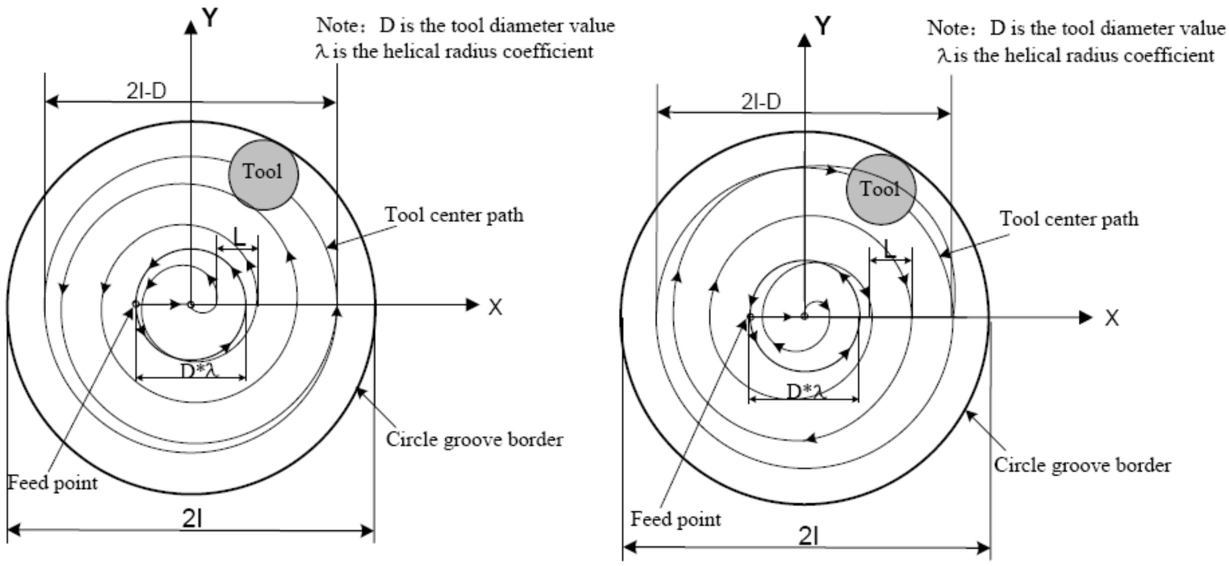


Fig. 4-6-1-2

Notice:

1. It is surposed to change the bit 1 of parameter No.12 into 1 when using this code.

2. The coefficient setting of helical cutting radius in the groove cycle should be more than 0, which is set by data parameter P269.

For example: An inner-circle groove is performed by rough-milling by canned cycle G22 code, refer to the following figure:

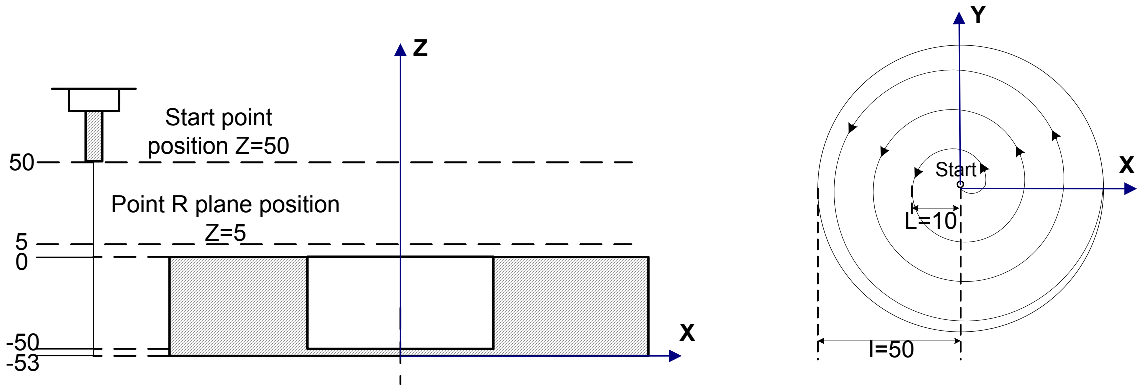


Fig. 4-6-1-3

G90 G00 X50 Y50 Z50; （G00 rapid positioning）

G99 G22 X25 Y25 Z-50 R5 I50 L10 W20 Q10 V10 D1 F800; (Incircle groove rough-millingcycle is performed)

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

M30;

Restriction: Fail to specify the G code (From G00 to G03, G60 are the modal codes (bit 0 of parameter No.:48 is set to 1)) of group 01 in a same block when using G22/G23 command, the system then performs the G60 modal.

Cutter compensation: In this canned cycle command positioning, tool radius offset is ignored; call the specified cutter compensation by program in the procedure of in-feed.

4.6.2 Fine-milling Cycle of the Full Incircle (G24/G25)

Code format:

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

Function: Tool performs a whole circle with fine-milling inside the circle based upon the specified radius value I and the direction, and then return after fine-milling is executed.

Explanations:

G24: Fine-milling cycle inside the whole circle along CCW.

G25: Fine-milling cycle inside the whole circle along CW.

X, Y: The start position of the X, Y plane;

Z: Machining depth, it is absolute position in G90; it is a position related to the R reference surface in G91;

R: R reference surface position, it is the absolute position in G90; it is a position related to the start of this block in G91;

I: Fine-milling circle radius;

J: The distance between the fine-milling start and the center of the fine-milling;

D: Tool compensation number, take out the corresponding tool compensation value based upon the provided series number;

K_: Repeated times.

Cycle processes:

(1) Rapid positioning to the postion along with XY plane;

(2) Descend to point R plane at the rapid traverse rate;

(3) Cutting feed to the start of the bottom of a hole;

(4) The transition arc 1 is regarded as path to perform an arc interpolation from the start;

(5) The fine-milling incircle is treated as a path to perform the whole-circle interpolation;

(6) The transition arc 4 is regarded as a ptht to perform the arc interpolation and then return to the start;

(7) Return to initial point plane or point R plane based upon the different specifications of G98 or G99.

Code path:

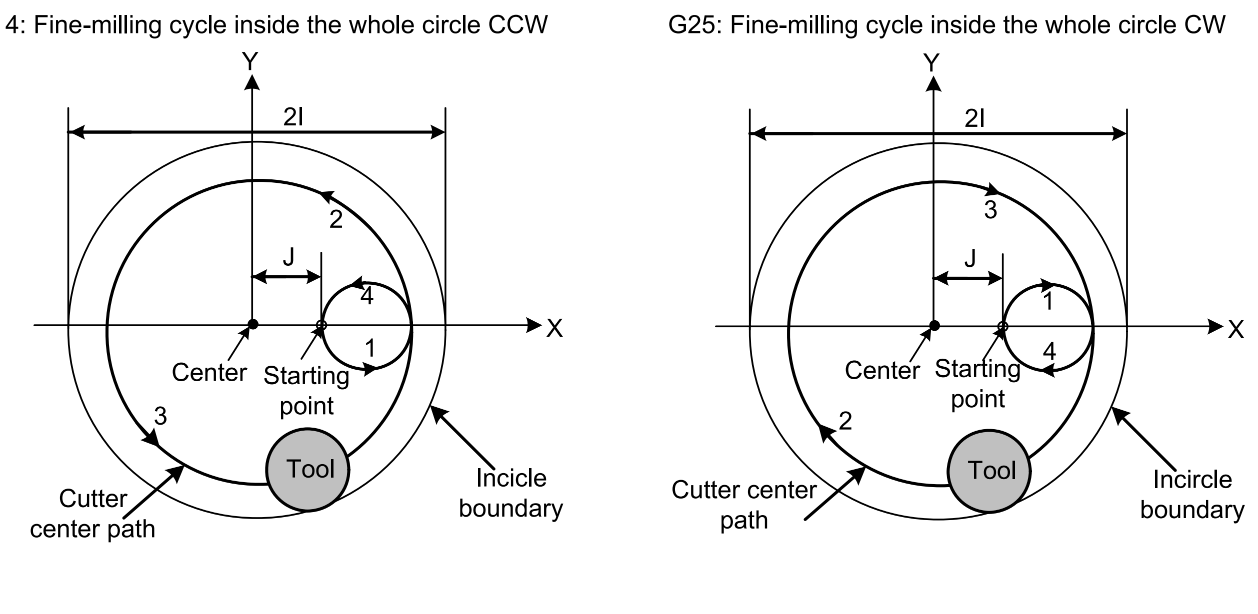


Fig. 4-6-2-1

Notice: It is suggested to change the bit 1 of parameter No.12 into 1 when using this code.

For example: The fine-milling is performed for the roughed circle groove by canned cycle G24 code; refer to the following figure.

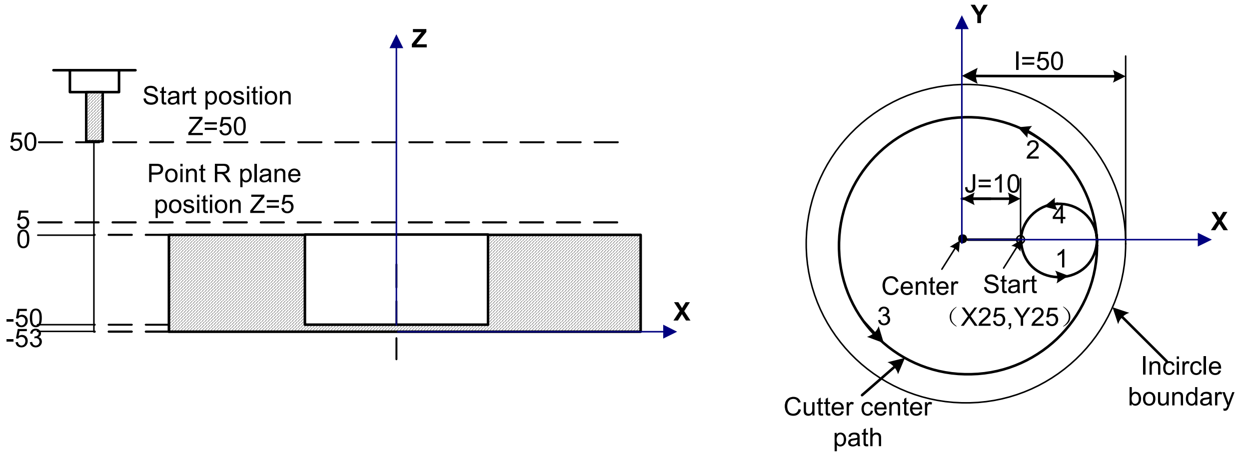


Fig. 4-6-2-2

G90 G00 X50 Y50 Z50； (G00 rapid positioning)

G99 G24 X25 Y25 Z-50 R5 I50 J10 D1 F800； (Canned cycle start, perform the incircle fine-milling cycle at the bottom of the hole)

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

M30；

Restriction: Fail to specify the G code (From G00 to G03, G60 are the modal codes (bit 0 of parameter No.:48 is set to 1)) of group 01 in a same block when using G24/G25 command; otherwise, G24/G25 will be replaced by the G code of group 01.

Cutter compensation: In this canned cycle command positioning, tool radius offset is ignored; call the specified cutter compensation by program in the procedure of in-feed.

4.6.3 Excircle Fine-milling Cycle (G26/G32)

Code format:

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

Function: Tool performs a whole circle at the excircle with fine-milling based upon the specified radius value and its direction, and then return.

Explanations:

G26: Excircle fine-milling cycle CCW

G32: Excircle fine-milling cycle CW

X, Y: The start position of X, Y plane;

Z: Machining depth, it is absolute position in G90; it is the position related to the R reference surface in G91;

R: R reference surface position, it is the absolute position in G90; it is the position related to the start of this block in G91;

I: Fine-milling circle radius;

J: The distance between the fine-milling start and the circle edge;

D: Tool compensation number, take out the corresponding tool compensation value based upon the offered series number;

K: Repeated times. (If required)。

Cycle processes:

(1) Positioning to the XY plane at the rapid traverse rate;

(2) Descend to point R plane at the rapid traverse rate;

(3) Cutting feed to the bottom of a hole;

(4) The transition arc 1 is regarded as path to perform arc interpolation from the start;

(5) Perform the whole circle interpolation based upon the path of the arc 2 and 3;

(6) The transition arc 4 is regarded as path to perform the arc interpolation and then return to the start;

(7) Return to initial point plane or point R plane based upon the different specifications of G98 or G99

Code path:

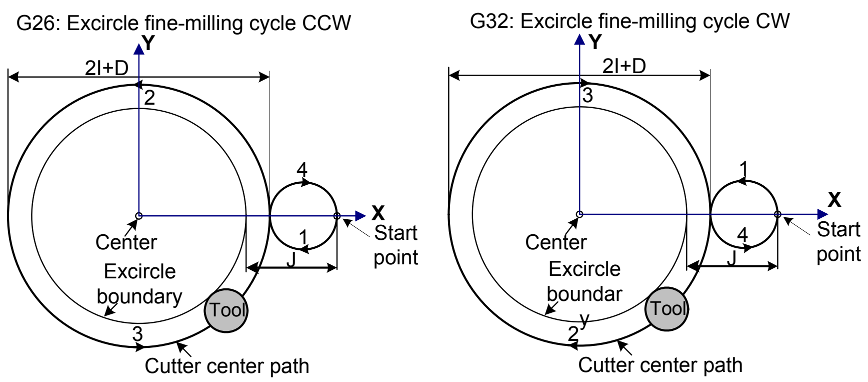


Fig. 4-6-3-1

Explanation:

The interpolaton both the transition arc and the fine-milling arc are inconsistent when the finish-milling is perfirmed at the excircle; the interpolation direction in the code explanation is the interpolation direction of the fininsh-milling arc.

For example: The fine-milling is performed for the roughed circle groove by canned cycle G26 code; refer to the following figure.

VOLUME I PROGRAMMING

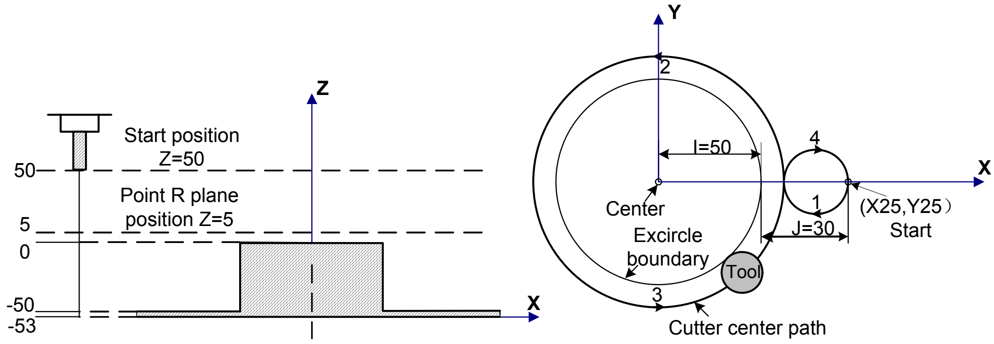


Fig. 4-6-3-2

G90 G00 X50 Y50 Z50; (G00 rapid positioning)

G99 G26 X25 Y25 Z-50 R5 I50 J30 D1 F800; (Canned cycle start, Excircle fine-milling cycle performs at the bottom of a hole)

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

M30;

Restriction: Fail to specify the G code (From G00 to G03, G60 are the modal codes (bit 0 of parameter No.:48 is set to 1)) of group 01 in a same block when using G26/G32 command, the system then executes G60 modal.

Cutter compensation: In this canned cycle command positioning, tool radius offset is ignored; call the specified cutter compensation by program in the procedure of in-feed.

4.6.4 Rectangular Groove Rough-milling (G33/G34)

Code format:

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

Function: Start from the center of the rectangle, the specified parameter data is regarded as the linear cutting cycle till the rectangular groove is machinied based upon the programmed dimension.

Explanations:

G33: Rectangular groove rough-milling CCW;

G34: Rectangular groove rough-milling CW;

X, Y: The start position on the X,Y plane;

Z: Machining depth, it is the absolute position in G90; it is the positon related to the R reference surface in G91;

R: R reference surface position, it is the absolute position in G90; it is the position related to the start in this block in G91;

I: The width of the rectangular groove along X axis direction;

J: The width of the rectangular groove along Y axis direction;

L: The cutting width increment within the specified plane;

W: Cutting along the Z axis at the 1st time is the downward distance from the R reference surface, which should be more than 0 (If the cutting depth for the 1st time exceeds the position at the bottom of the groove, it will be directly machined based upon the position of groove bottom);

Q: Cutting depth of each cutting feedrate;

V: The distance from the unprocessed surface during the cutting at the rapid traverse rate;

U: Corner arc radius; without corner arc transition if it omits;

D: Tool compensation number, take out the corresponding tool compensation value based upon the offered series number;

K_: Repeated times.

Cycle processes:

(1) Positioning to the helical cutting start position along XY plane at the rapid traverse rate;

(2) Descend to the point R plane at the rapid traverse rate;

(3) Helical cutting W distance based upon that the radius compensation value miltiples the value of data parameter No.269 is regarded as the diameter;

(4) Feed to the rectangular center position;

(5) The rectangular surface milled increasing by L value for each time where from center to outside;

(6) Z axis returns to R refernce surface at the rapid traverse rate;

(7) Positioning to helix cutting start position along the XY plane at the rapid traverse rate;

(8) Z axis descends to the distance from the unprocessed plane V;

(9) The cutting (Q+V) depth downward along Z axis;

(10) Repeat the operations from (4) to (9) till to the total cutting depth on the surface is machined;

(11) Return to initial point plane or point R plane based upon the different specifications of G98 or G99.

Code path:

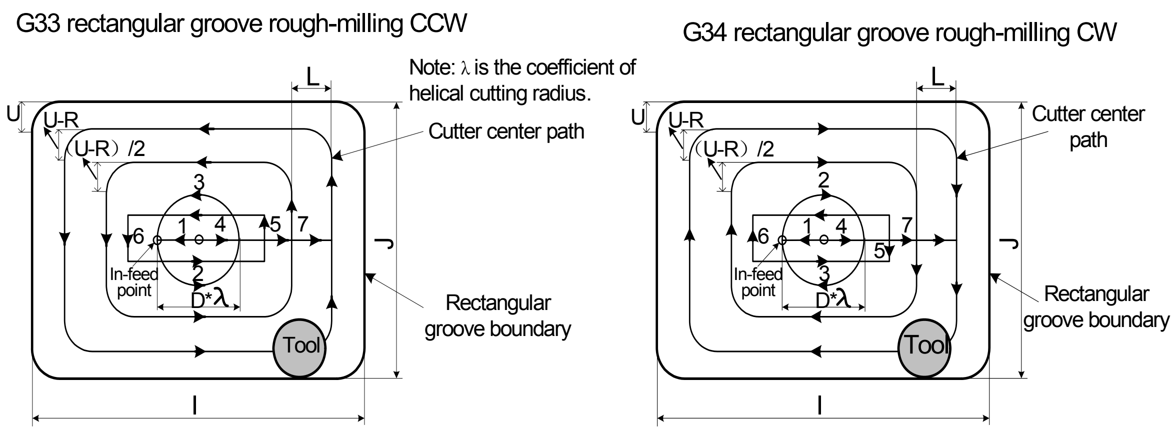


Fig. 4-6-4-1

Notice: It is suggested to change the bit 1 of parameter No.12 into 1 when using this code.

For example: The rough-milling is performed within a rectangle groove by canned cycle G33 code; refer to the following figure:

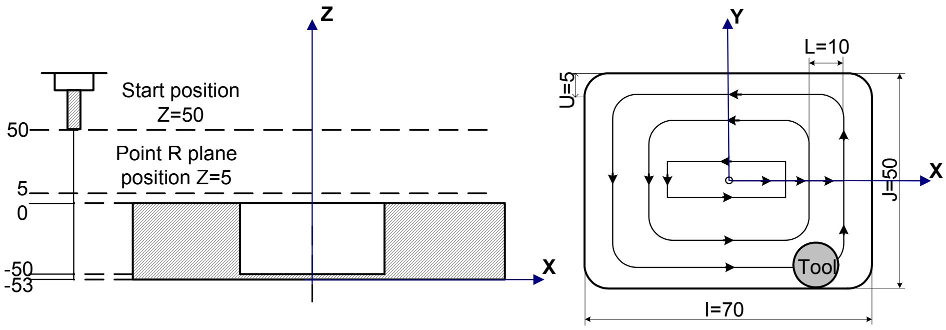


Fig. 4-6-4-2

G90 G00 X50 Y50 Z50; (G00 rapid positioning)

G99 G33 X25 Y25 Z-50 R5 I70 J50 L10 W20 Q10 V10 U5 D1 F800; (Perform the groove rough-milling cycle within rectangle)

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

M30;

Restriction: Fail to specify the G code (From G00 to G03, G60 are the modal codes (bit 0 of parameter No.:48 is set to 1)) of group 01 in a same block when using G33/G34 command; otherwise, G33/G34 will be replaced by the G code of group 01.

Cutter compensation: In this canned cycle command positioning, tool radius offset is ignored; call the specified cutter compensation by program in the procedure of in-feed.

4.6.5 Fine-milling Cycle within Rectangular Groove (G35/G36)

Code format:

G35

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

G36

Function: Tool is performed the fine-milling with the specified width and direction, and then return after the fine-milling is completed.

Explanations:

G35: Fine-milling cycle inside the rectangular groove CCW.

G36: Fine-milling cycle inside the rectangular groove CW.

X, Y: The start position on X, Y plane;

Z: Machining depth, it is the absolute position inG90; it is the position related to the R reference surface in G91;

R: R reference surface position, it is the absolute position in G90; it is the position related to the start of this block in G91;

I: The width along with rectangular X axis direction;

J: The width along with rectangular Y axis direction;

L: The distance from the fine-milling start to the rectangular edge positive direction along X axis;

U: Corner arc radius; there is no corner arc transition if it omits. When 0＜｜U｜＜tool radius, the alarm then occurs;

D; Cutter compensation number, take out the corresponding cutter compensation value based upon the provided series number;

K: Repeated times.

Cycle processes:

(1) Positioning to the start position on XY plane at the rapid traverse rate;

(2) Descend to the point R plane at the rapid traverse rate;

(3) Cutting feed to the bottom of a hole;

(4) Perform the arc interpolation from the start based upon that the transition arc 1 is regarded as the path;

(5) Perform the linear and arc interpolation based that the 2-3-4-5-6 are treated as path;

(7) Return to start to perform the arc interpolation based upon that the transition arc 7 is regarded as path;

(7) Return to initial point plane or point R plane based upon the different specifications of G98 or G99.

Code path:

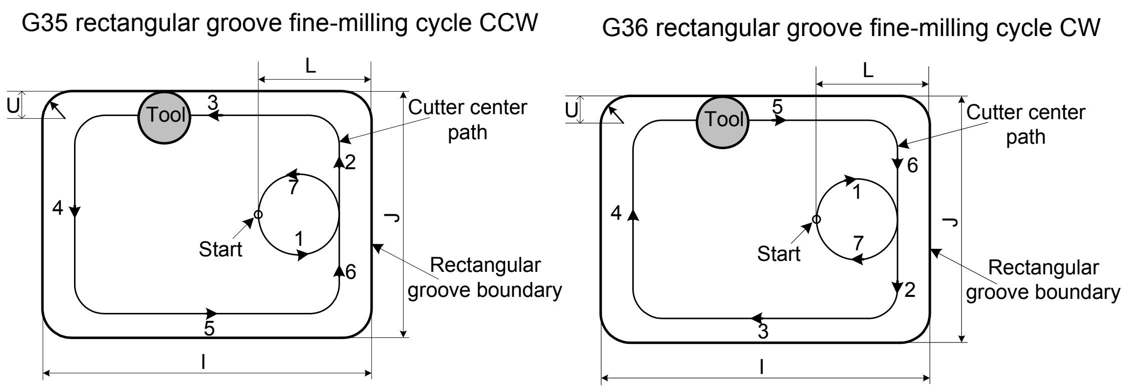


Fig. 4-6-5-1

Notice: It is suggested to change the bit 1 of parameter No.12 into 1 when using this code.

For example: The fine-milling is performed for the roughed groove by canned cycle G35 code; refer to the following figure.

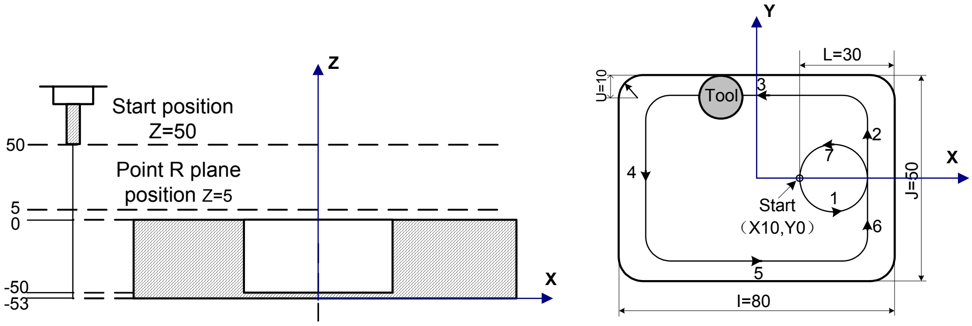


Fig. 4-6-5-2

G90 G00 X50 Y50 Z50; (G00 rapid positioning)

G99 G35 X10 Y0 Z-50 R5 I80 J50 L30 U10 D1 F800; (Perform the rectangular groove inner milling to the bottom of a hole in the canned cycle method)

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

M30;

Restriction: Fail to specify the G code (From G00 to G03, G60 are the modal codes (bit 0 of parameter No.:48 is set to 1)) of group 01 in a same block when using G35/G36 command, the system then executes G60 modal.

Cutter compensation: In this canned cycle command positioning, tool radius offset is ignored; call the specified cutter compensation by program in the procedure of in-feed.

4.6.6 Fine-milling ccle Outside the Rectangle (G37/G38)

Code format:

G37

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

G38

Function: Tool is performed the fine-milling with the specified width and direction, and then return after the fine-milling is completed.

Explanations:

G37: Fine-milling cycle outside the rectangular groove CCW.

G38: Fine-milling cycle outside the rectangular groove CW.

X, Y: The start position on X, Y plane;

Z: Machining depth, it is the absolute position inG90; it is the position related to the R reference surface in G91;

R: R reference surface position, it is the absolute position in G90; it is the position related to the start of this block in G91;

I: The width along with rectangular X axis direction;

J: The width along with rectangular Y axis direction;

L: The distance from the fine-milling start to the rectangular edge positive direction along X axis;

U: Corner arc radius; there is no corner arc transition if it omits;

D; Cutter compensation number, take out the corresponding cutter compensation value based upon the provided series number;

K: Repeated times.

Cycle processes:

(1) Positioning to the start position on XY plane at the rapid traverse rate;

(2) Descend to the point R plane at the rapid traverse rate;

(3) Cutting feed to the bottom of a hole;

(4) Perform the arc interpolation from the start based upon that the transition arc 1 is regarded as the path;

(5) Perform the linear and arc interpolation based that the 2-3-4-5-6 are treated as path;

(7) Return to start to perform the arc interpolation based upon that the transition arc 7 is regarded as path;

(8) Return to initial point plane or point R plane based upon the different specifications of G98 or G99.

Code path:

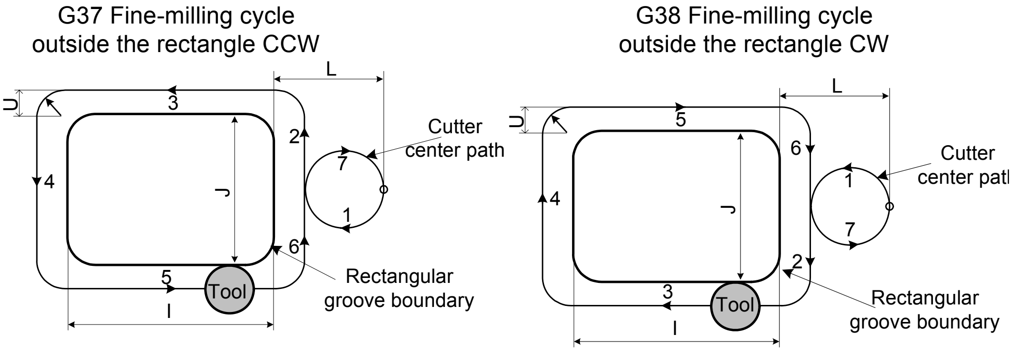


Fig. 4-6-6-1

Explanation:

The transition arc is inconsistent with the interpolation direction of the fine-milling arc when the finish-milling is performed outside the rectangle. The interpolation direction in the code explanation is the one of the fine-milling arc.

For example: Fine-milling outside the rectangle is performed by canned cycle G37 code.

G90 G00 X50 Y50 Z50; (G00 rapid positioning)

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

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

M30;

Restriction: Fail to specify the G code (From G00 to G03, G60 are the modal codes (bit 0 of parameter No.:48 is set to 1)) of group 01 in a same block when using G37/G38 command, the system then executes G60 modal.

Cutter compensation: In this canned cycle command positioning, tool radius offset is ignored; call the specified cutter compensation by program in the procedure of in-feed.

4.6.7 Canned Cycle Cancel (G80)

Code format: G80

Function: Canned cycle canel

Explanation:

Cancel the overall canned cycles, and then perform the normal operation, at the same time, the point R and Z are cancelled, as well the other drilling, boring data are eliminated, too.

Example:

M3 S100; Spindle rotation

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

Positioning, bore hole 1, then return to the point R

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

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

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

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

G98 Y-750; Positioning, bore hole 1, then return to the initial position plane

G80;

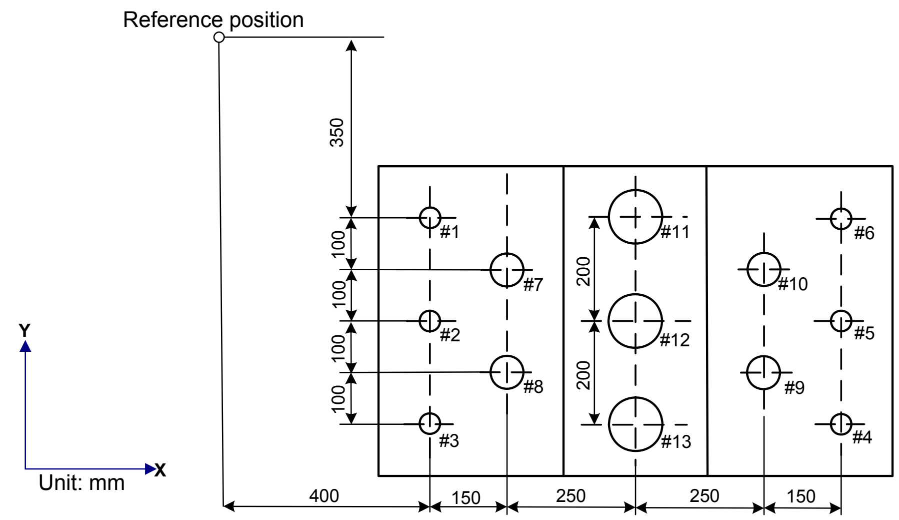
G28 G91 X0 Y0 Z0; Return to the reference position and then cancel the canned cycle.

M5; Spindle rotation stop

M30;

For example:

The following figure uses the tool length compensation, which are expressed the usage of the canned cycle totally.



 # 1～ 6... drill Φ10 hole

 # 7～10...drill Φ20 hole

#11～13..bore Φ95 hole

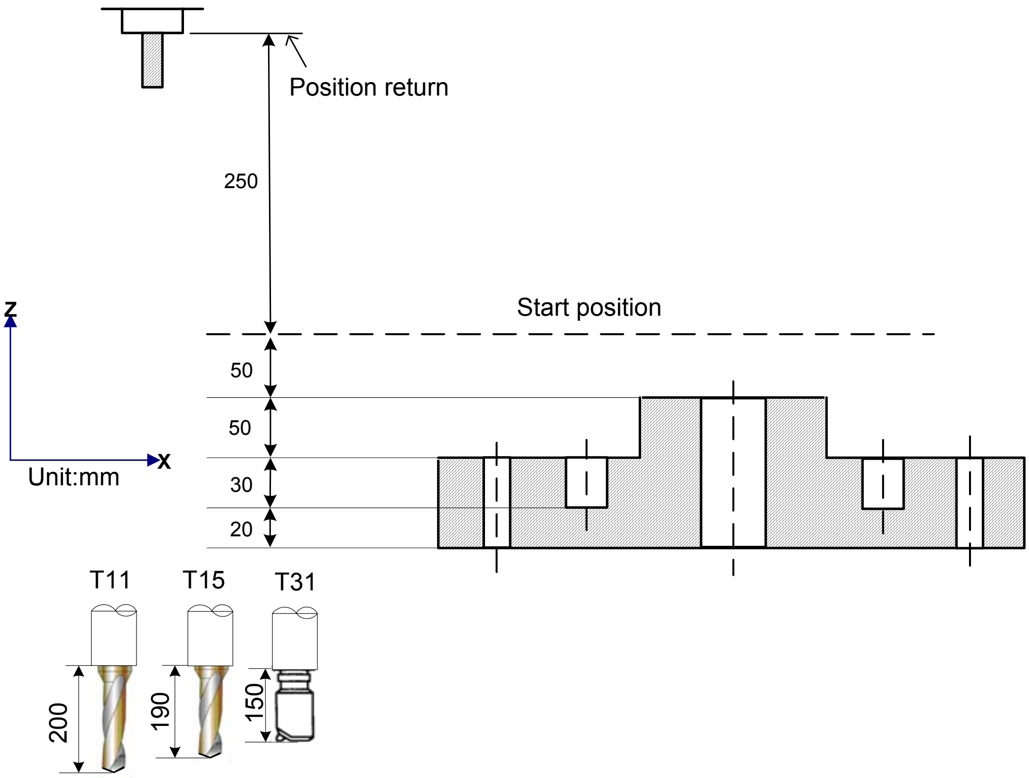


Fig. 4-6-7-1

The value of offset No.11 is 200, No.15 is 190 and No.31 is 150, which are separately set as the offset. The program are shown below:

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

	at the initial point
N004 S300 M3 ；	Spindle start.
N005 G99 G81 X400 Y-350 ； Z-153 R-97 F120 ；	Machine #1 hole after positioning.
N006 Y-550 ；	Machine #2 hole after positioning, then return to point R plane.
N007 G98 Y-750 ；	Machine #3 hole after positioning, then return to point initial point plane.
N008 G99 X1200 ；	Machine #4 hole after positioning, then return to point R plane.
N009 Y-550 ；	Machine #5 hole after positioning, then return to point R plane.
N010 G98 Y-350 ；	Machine #6 hole after positioning, then return to point initial point plane.
N011 G00 X0 Y0 M5 ；	Return to reference position, spindle then stops
N012 G49 Z250 T15 M6 ；	Cancel the tool length compensation, then tool-change performs.
N013 G43 Z0 H15 ；	Initial point plane, tool length compensation
N014 S200 M3 ；	Spindle start.
N015 G99 G82 X550 Y-450 ； Z-130 R-97 P30 F70 ；	Machine #7 hole after positioning, then return to point R plane.
N016 G98 Y-650 ；	Machine #8 hole after positioning, then return to point initial point plane.
N017 G99 X1050 ；	Machine #9 hole after positioning, then return to point R plane.
N018 G98 Y-450 ；	Machine #10 hole after positioning, then return to point initial point plane.
N019 G00 X0 Y0 M5 ；	Return to reference point, spindle stops.
N020 G49 Z250 T31 M6 ；	Cancel the tool length compensation, then prform the tool-change.
N021 G43 Z0 H31 ；	Initial point plane tool length compensation.
N022 S100 M3 ；	Spindle start.
N023 G85 G99 X800 Y-350 ； Z-153 R47 F50 ；	Machine #11 hole after positioning, then return to point R plane.
N024 G91 Y-200 ； Y-200 ；	Machine #12, #13 holes after positioning, then return to point R plane.

N025 G00 G90 X0 Y0 M5 ；	Return to reference point, then spindle stops
N026 G49 Z0 ；	Cancel the tool length compensation
N027 M30 ；	Program stops

4.7 Tool Compensation G Code

4.7.1 Tool Length Compensation G43, G44 and G49

Function:

G43 specifies the positive compensation of tool length.

G44 specifies the negative compensation of tool length.

G49 cancels tool length compensation.

Format:

System supports A/B tool length offset methods, which can be set by **bit 0 of parameter No.:39**.

Method A:

$\left.\begin{array}{l}\text{G43}\\\text{G44}\end{array}\right\}$ Z_ H_ ；

Method B:

G17 G43 Z_H;

G17 G44 Z_H;

G18 G43 Y_H;

G18 G44 Y_H;

G19 G43 X_H;

G19 G44 X_H;

Tool length offset method cancel: G49 or H0.

Explanation:

The purose of the above-mentioned codes is specified a end position of the specified axis and then moves an offset. The D-value of imaged tool length value in programming and the tool length value used in the actual machining are set in the offset register in advance; and therefore, the component machining can be performed by the tool with different length based upon changing the tool length compensation value, instead of altering the program.

G43 and G44 are specified different offset direction; and the offset number can be specified by H code.

1. Offset direction

G43: Positive offset (It is the most common offset method)

G44: Negative offset

The specified axis in the program moving with the command end coordinate value addes the offset value (It is set at the offset register) specified by H code regardless of the absolute value code or incremental value code. In G44, deduct the offset value specified in H code, and then its coordinate value calculated is regarded as the end coordinate value.

G43 and G44 are modal G codes, which are enabled before encountering other G codes in the same group.

2. The specification of offset value

The offset length offset number is specified by H code, the corresponding offset value of the offset number adds or subtracts to the code value moving with Z axis in program, and then becomes the new Z axis movement code. H00～H255 can be specified according to the offset numbers.

The setting range of the offset values are shown below:

Table 4-7-1-1

	Range
Compensation value H (input by mm)	-999.999 mm～+999.999mm
Compensation value H (input by inch)	-39.3700 inch～+39.3700 inch

Offset No.00 means that H00 is corresponding with the offset value 0, which can not be set in system.

Notice: When the offset value is changed due to the offset number, only the new offset value replaces the old one instead of adding. For example:

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

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

G90 G43 Z100 H01 ； .... Z moves to 120

G90 G43 Z100 H02 ； .... Z moves to 130

3. The effective sequence of offset number

The current offset number will be immediately enabled once the length offset modal is set up. However, the new offset value will immediately replace the old one if the offset number is changed.

O×××××；

G43 Z10 H01； (1) Offset number H01 is enabled

G44 Z20 H02； (2) Offset number H02 is enabled

Z30 H03； (3) Offset number H03 is enabled

G49； (4) Cancel the tool offset at the end of this block

M30；

4. Cancel the tool length compensation

Cancel the tool compensation by G49 or H00. The system will immediately cancel the tool length compensation after G49 is specified. The compensation axis address and compensation should be compiled followed with the H00; otherwise, the tool length compensation can not be cancelled.

Notice: 1. The method B of tool length offset is performed after along more than two axes, cancel the offset of overall axes by G49, and the offset value of axis vertical to the specified plane is only cancelled by H00.

2. It is suggested that the establishment and cancellation of length compensation adds to the movement code along Z axis; otherwise, the establishment and length compensation cancellation will be performed at the current point; and therefore, it is better to confirm that the Z axis is on the safety height when using G49, prevent the the workpiece from impacting or damaging.

5. The concrete example of tool length compensation

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

(B) H01= offset value -4

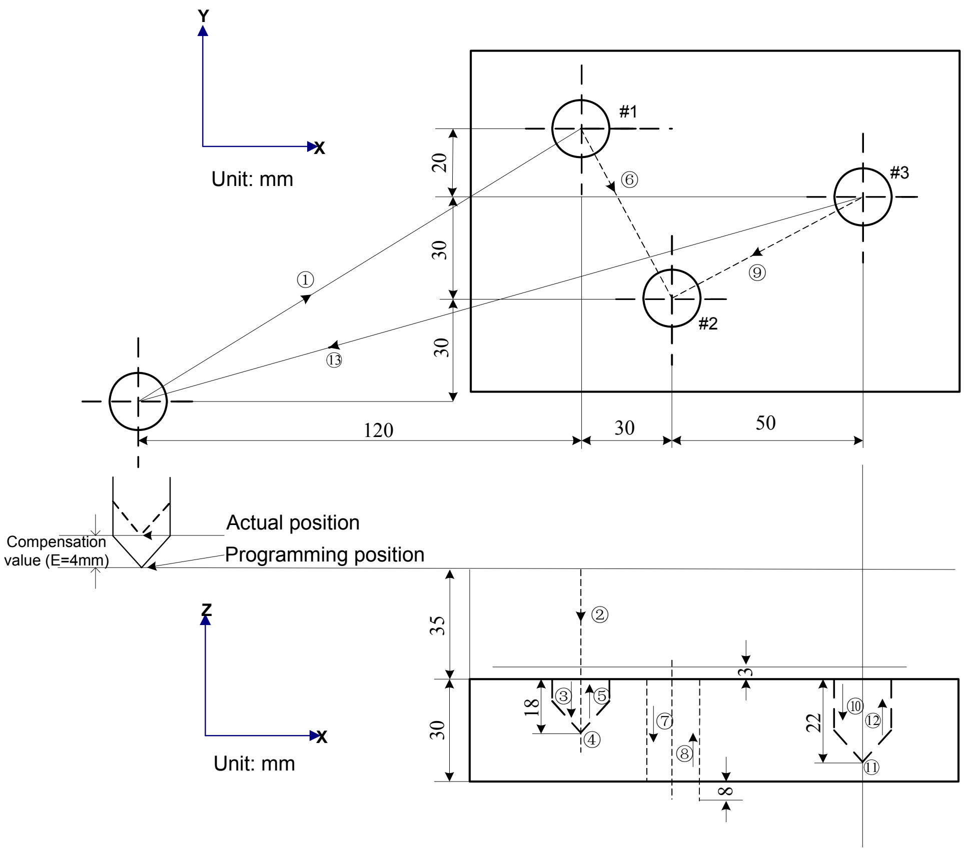


Fig. 4-7-1-1

N1 G91 G00 X120 Y80 ； …….. (1)

N2 G43 Z-32 H01 ； ....……… (2)

N3 G01 Z-21 F200 ； .... …(3)

N4 G04 P2000 ； .... (4)

N5 G00 Z21 ； ....…(5)

N6 X30 Y-50 ； ....…(6)

N7 G01 Z-41 F200 ； ....….(7)

N8 G00 Z41 ； ....….(8)

N9 X50 Y30 ； .... ….... (9)

N10 G01 Z-25 F100 ； ....….(10)

N11 G04 P2000 ； ....….(11)

N12 G00 Z57 H00 ； ....(12)

N13 X-200 Y-60 ； ....(13)

N14 M30 ；

4.7.2 Cutter Compensation G40/G41/G42

Code format:

G41 D_ X_Y_;

G42 D_ X_Y_;

G40 X_Y_;

Function:

G41 specifies the left compensation of tool movement direction.

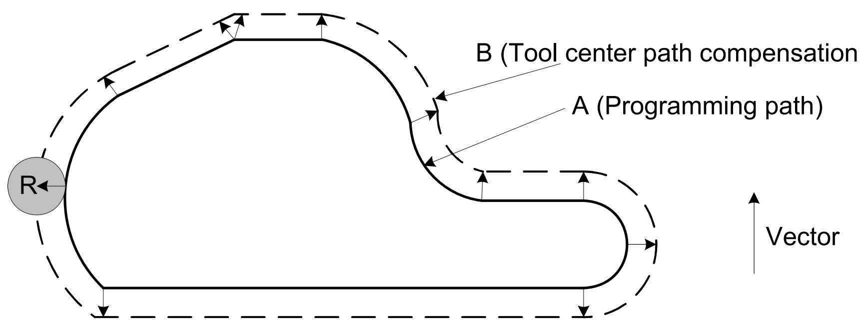
G42 specifies the right compensation of tool movement direction.

G40 Cutter compensation cancellation.

Explanations:

1. Cutter compensation function

The following figure shows that the tool with radius R cuts the workpiece A, tool center path is shown as B in the figure; the distance from path B to A is R. The distance of tool offset workpiece A radius is called compensation.



Compensation and vector

Fig. 4-7-2-1

Porgrammer uses the cutter compensation mode to compile the machining program, during machining, measure the tool diameter and record the register of CNC, and then the tool path becomes the compensation path B.

2. Compensation value (D value)

The radius offset number is specified by D code, which is the cooresponding offset value is added or deducted to the movement code value in program, so that the new movement code forms. The offset numbers can be specified D00～D255 based upon the requirements. Whether the radius compensation value is set by diameter or radius by **bit 7 of parameter No.: 40**.

The compensation value corresponding offset can be set the offset register in advance by using LCD/MDI panel.

The setting range of compensation value is shown below:

Table 4-7-2-1

	Range
Compensation value D (mm)	-999.999ｍｍ～+999.999ｍｍ
Compensation D (inch)	-39.3700 inch～+39.3700 inch

Notice: The system regards the compensation value of D00 is 0 by default, user can neither set nor modify.

The compensation plane should be performed after cancelling the compensation mode. The system will alarm if the compensation plane changes without cancelling compensation mode.

3. Plane selection and vector

The compensation calculation is performed within the plane selected by G17, G18 and G19. This plane is called compensation plane. For example, when the XY plane is selected, the compensation and vector calculations can be executed by (X, Y) in program. The coordinate value of the axis without the compensation plane is regardless of the compensation.

Simultaneously, 3 axes are controlled, only the tool path shadowed on the compensaton plane is compensated.

The alteration of compensation plane should be performed after cancelling the compensation mode.

Table 4-7-2-2

Code	Plane compensation
G17	X – Y plane
G18	Z - X plane
G19	Y - Z plane

4. G40, G41 and G42

The cancellation and performance of cutter compensation is commanded by G40, G41 and G42 of which they are defined a modal to affirm the value or the direction of compensation vector combined with the G00 or G01 and codes.

Table 4-7-2-3

G code	Function
G40	Cutter compensation cancellation
G41	Left-compensation of tool radius
G42	Right-compensation of tool radius

5. G53, G28 and G30 codes in cutter compensation method

When specifying G53, G28 or G30 code in cutter compensation mode, the offset vector of tool radius offset axis is cancelled (Wherein, it is cancelled when G53 moves to the specified position and when G28, G30 moves to the reference position.) when moving to the specified position; the axie other than the tool radius offset axis does not cancel. When G53 is shared a same block with the G41//G42, the radius compensation is cancelled when the overall axes are moved to the command position. When G28 or G30 is shared a block with G41//G42, the overall axes are cancelled the radius compensation when moving to the reference position. The vector cancelled cutter compensation will recover by the buffered next compensation plane block.

Note: In the compensation mode, the bit 2 of parameter No.:42 can be selected that when G28 or G30 code moves at the intermediate point, and then confirm whether the compensation will dwell temporarily.

Cutter compensation cancellation (G40)

In the G00, G01 state, the following code is used, G40 X__ Y__;

The linear operation from the old vector of start to the end is performed. In the G00 mode, each axis moves to the end at the rapid traverse rate. The system enters the tool compensation cancellation state from the tool compensation state when using this code.

Tool will never operate when X__ Y__ does not specify but the G40.

Cutter compensation Left (G41)

1) G00, G01

G41 X__ Y__ D__; code locates at the end of block and forms a new vector vertical with the (X, Y) directin, tool moves to the point of new vector from the old one at the start.

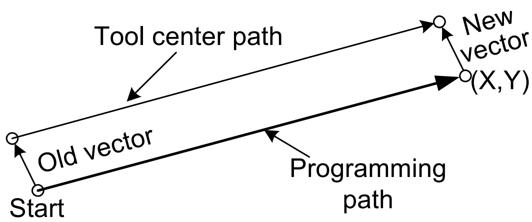


Fig. 4-7-2-2

Using this code when old vector is regarded as 0, so that the tool enters to the cutter compensation state from the tool offset cancellation state. In this case, the offset value is specified by D code.

2) G02, G03

G41……;

……

……

G02 /G03 X__ Y__ R__ ;

New vectors can be carried out by the above-mentioned programs, which are located at the line between arc center and end; viewing from the forward direction of the arc, point at the left (or right); tool center moves along arc from the old vector point of arc to the new one. However, the premise is that the old vector has been performed correctly.

The offset vector points to the arc center or departs from the center from the start or end.

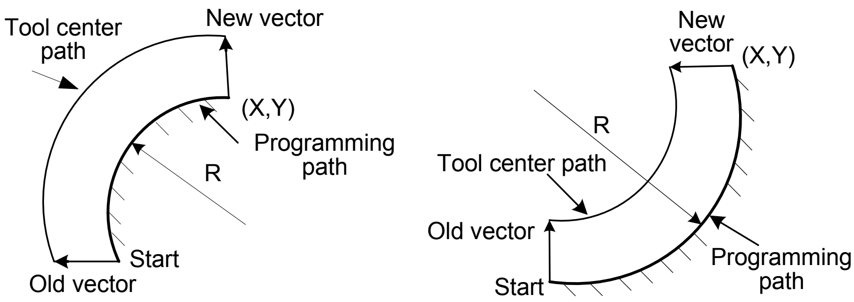


Fig. 4-7-2-3

Cutter compensation Right (G42)

G42 is opposite to G41, which moves forward along the tool, and the tool offsets at the right of workpiece. That is, the vector direction from G42 is opposited to the one of G41; however the offset method is absolute identical with the G41 other than the vector direction.

1) G00, G01

G42 X__ Y__ D__ ;

G42 X__ Y__ ;

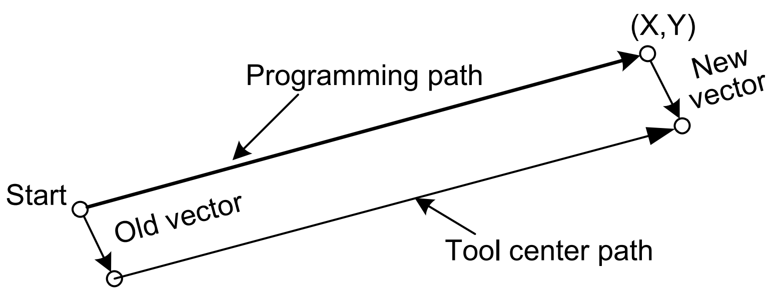


Fig. 4-7-2-4

2) G02, G03

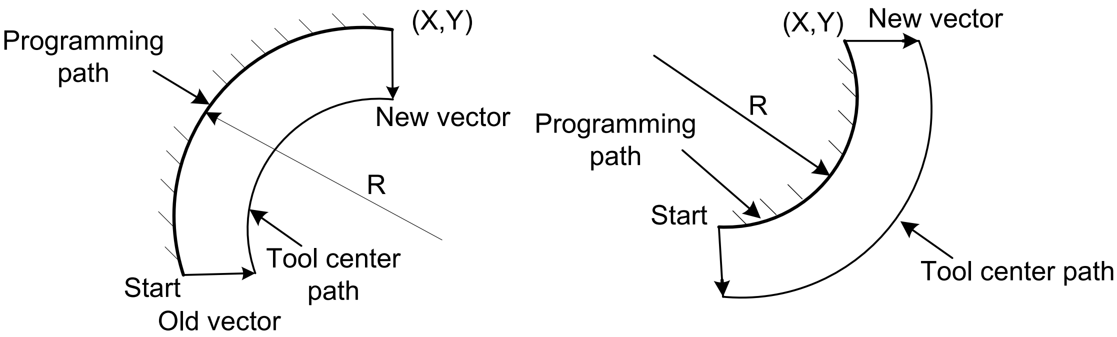


Fig. 4-7-2-5

6. The common precautions for the offset:

(A) The specification of offset

G41, G42 and G42 are modal codes of which the offset numbers are specified by G code. Any position can be specified before the cutter compensation state from offset state cancellation.

(B) Enter the cutter compensaton state from the offset state cancellation

The movement codes from offset state cancellation to the cutter compensation state should be performed the positioning (G00) or linear interpolation (G01) instead of using the arc interpolation (G02, G03).

(C) The shifting between the cutter compensation left and right

Generally, whenever the offset direction performs from left to right or on the contrary, which goes throught the offset cancellation state. However, the positoning (G00) or linear interpolation (G01) can be directly shifted instead of passing the offset cancellation state. In this case, the tool path is as follows:

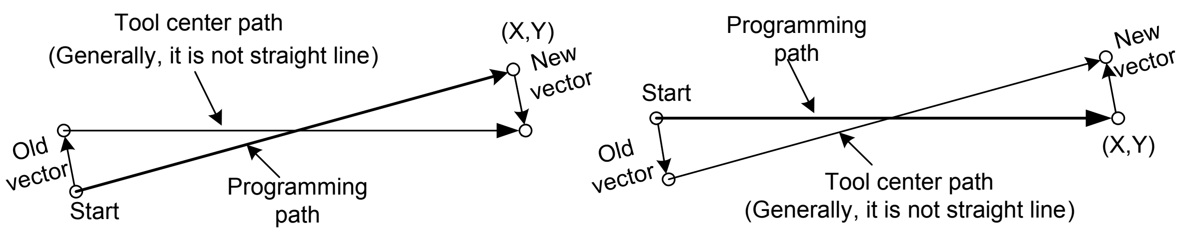


Fig. 4-7-2-6

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

(D) The alteration of offset value

The alteration of offset value, generally, is in the offset cancellation state, when the tool-change performs, the position (G00) and linear interpolation also can be performed in the offset state, its description is shown below:

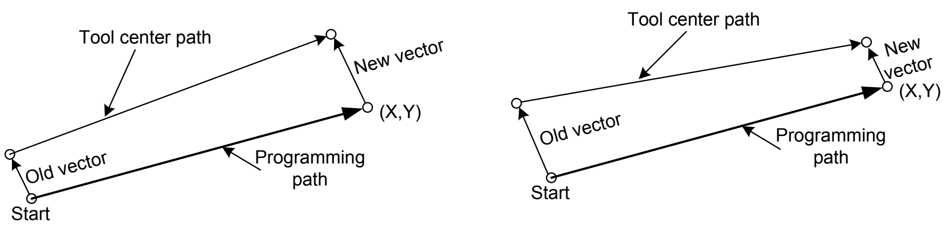


Fig. 4-7-2-7 (The alteration of offset vector)

(E) The +/- of offset value and tool center path

If the offset value is set as negative value, the machined workpiece equals to that the G41 and G42 on the program list are totally changed. And therefore, the cutting along the external workpiece becomes to the internal cutting, and the original internal machinig along workpiece turns into the external machining.

It is supposed that the offset value is positive value in the common program; refer to the following figure:

When the offset value of the tool path programming (Fig. A) is set as negative value, so that the tool movement path is as the Fig. B; similarly, if the offset value of the tool path programming (Fig. B) is set as negative value, and then the tool movment path is as the Fig. (A).

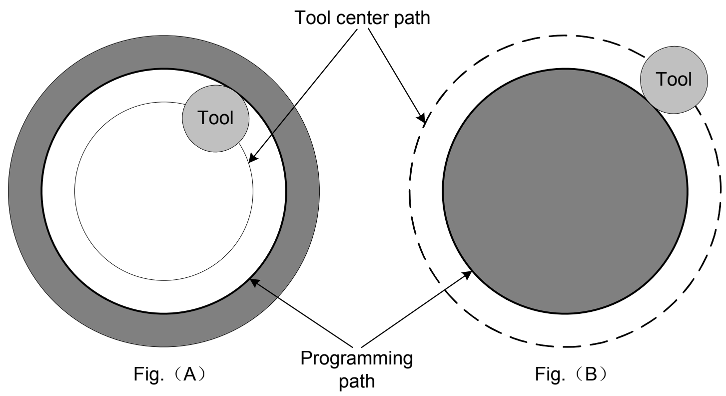


Fig. 4-7-2-8

Generally, the figure with the pointed angle is the most common. However, after the offset value is set to negative value, the inner round of machining component can not be performed. When an inner pointed angle of one angle is cut, inser the sutiable radius arc at this position, and then the cutting can be performed after the acr transition.

Whether it is the left compensation or the right is confirmed that the tool movement is left or right roleated to the workpiece (it is regarded as the workpiece unmoved). The system enters the compensatin mode by G41 or G42 and cancels it by G40.

The example of compensation program shows below:

The block (1) calls starting, the compensation cancellation mode in the block becomes compensation mode by G41. At the end of this block, tool center is vertical to the compensation direction of the next program path (From P1 to P2) by tool radius. The tool compensation value is specified by D07, that is, the compensation number is set to 7, and the G41 means the tool path path compensation left.

The tool path compensation is automatically performed after the compensation starts and when the workpiece shape is compiled such as P1→P2……P9→P10→P11.

The program example of tool path compensation:

G92 X0 Y0 Z0;

(1) N1 G90 G17 G0 G41 D7 X250 Y550 ; (The compensation value should be set by its compensation number in advance.)

(2) N2 G1 Y900 F150 ;

(3) N3 X450 ;

(4) N4 G3 X500 Y1150 R650 ;

(5) N5 G2 X900 R-250 ;

(6) N6 G3 X950 Y900 R650 ;

(7) N7 G1 X1150 ;

(8) N8 Y550 ;

(9) N9 X700 Y650 ;

(10) N10 X250 Y550 ;

(11) N11 G0 G40 X0 Y0 ;

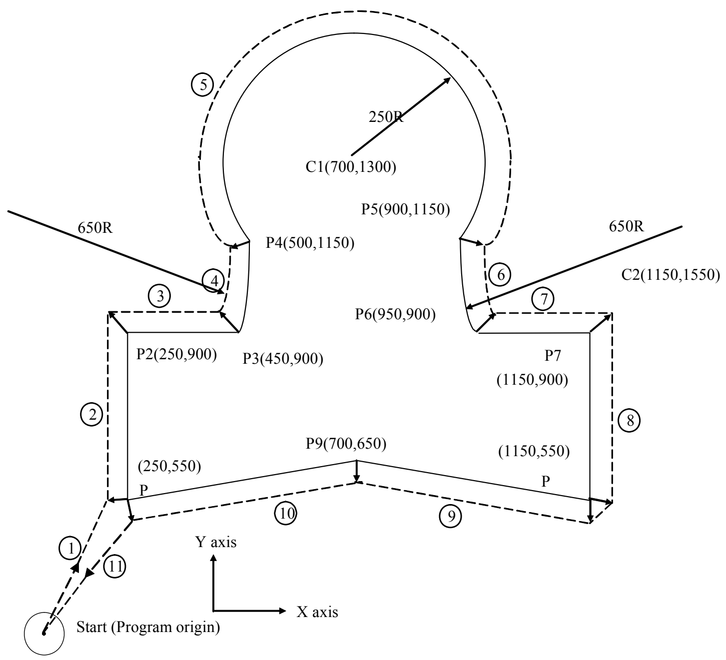


Fig. 4-7-2-9

4.7.3 Details of Cutter Compensation

Concept: Inner side and outer side: When an angle of intersection created by tool paths specified with move command for two blocks in over 180°, it is referred to as “inner side.” When the angle is between 0° and 180°, it is referred to as “outer side.”

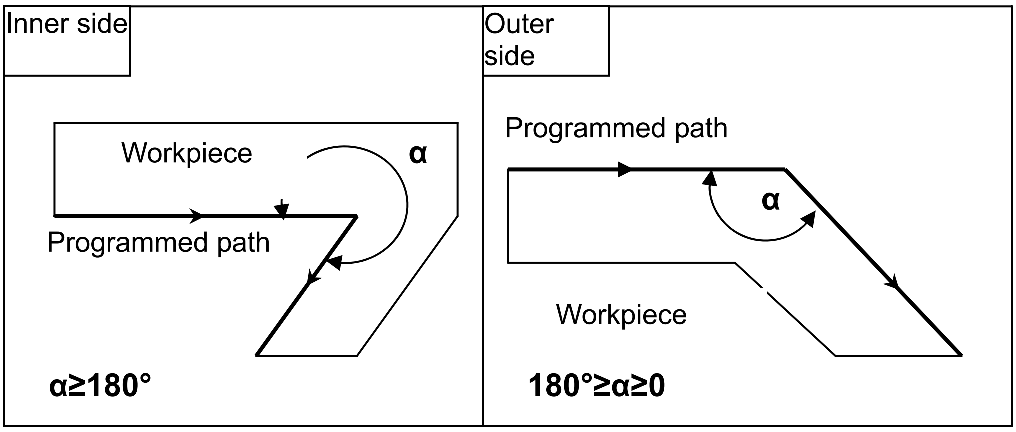


Fig. 4-7-3-1

The meaning of symbols:

The following symbols are used in sebsequent figures:

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

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

—— SSS indicates a position at which a single block is executed thrice.

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

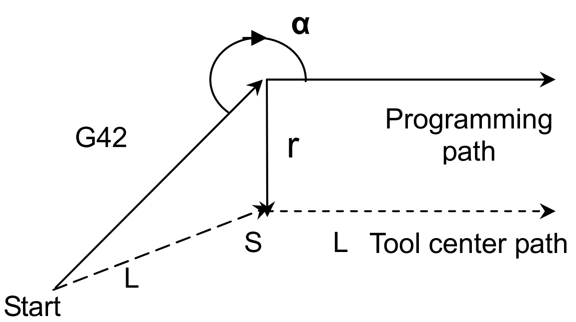
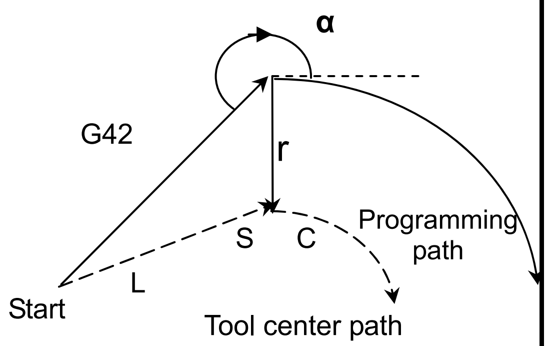
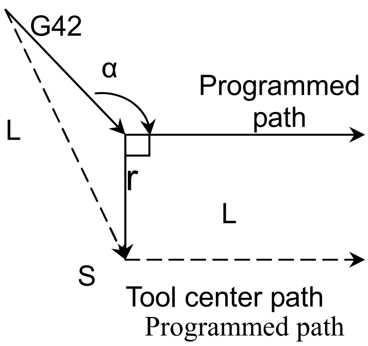
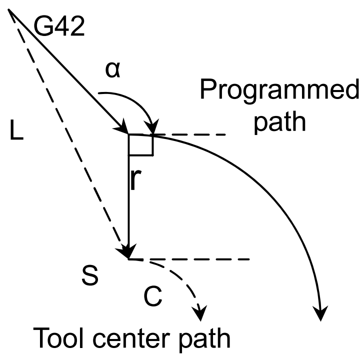
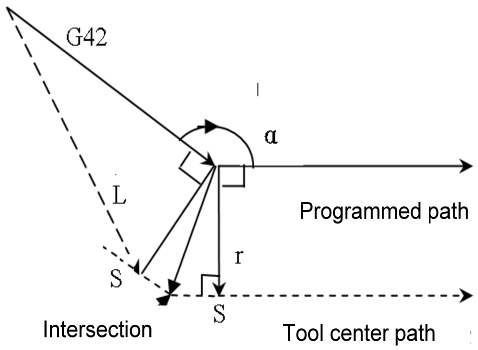
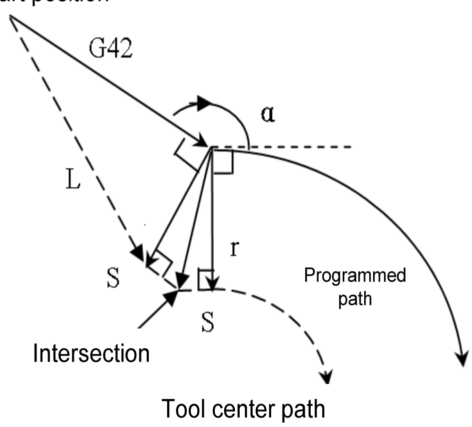
—— C indicates that the tool moves along an arc.

—— r indicates the cutter compensation value

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

—— O indicates the center of the tool

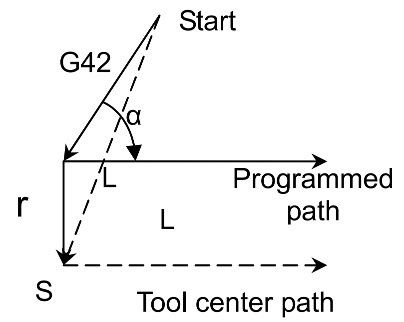
1. Tool movement in start-up When the offset cancellation mode is changed to offset mode, the tool moves as illustrated below (Start-up):

<table>
<tr><td colspan="3">(a) Tool movement around an inner side of a corner (α≥180°)</td></tr>
<tr><td colspan="2">Linear - Linear
</td><td>Linear - Arc
</td></tr>
<tr><td colspan="3">(b) Tool movement around the outside of a corner at an obtuse angle (180°＞α≥90°)
Tool path in the beginning of compensation has two types A and B, which are selected by bit 0 of parameter No.: 40:</td></tr>
<tr><td>A</td><td>Linear - Linear
Start position
</td><td>Linear - Arc
Start position
</td></tr>
<tr><td>B</td><td>Start position Linear-Linear
Note: Intersection is the position where offset paths of two successive blocks intersect.</td><td>Start position Linear-Circular
</td></tr>
<tr><td colspan="3">(C) Tool movement around the outside of an acute (α＜90°)
Tool path in the beginning of compensation has two types A and B, which are selected by bit 0</td></tr>
</table>

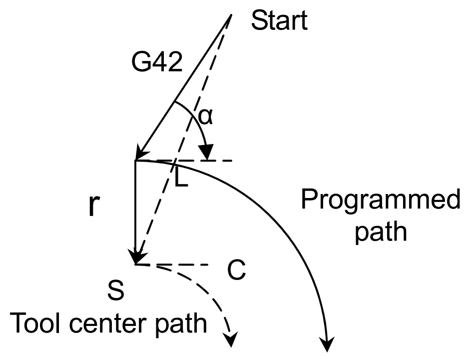
of parameter No.: 40:

A

Linear—Linear

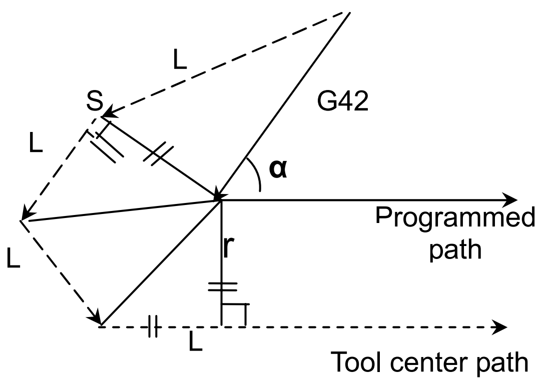


Linear—arc

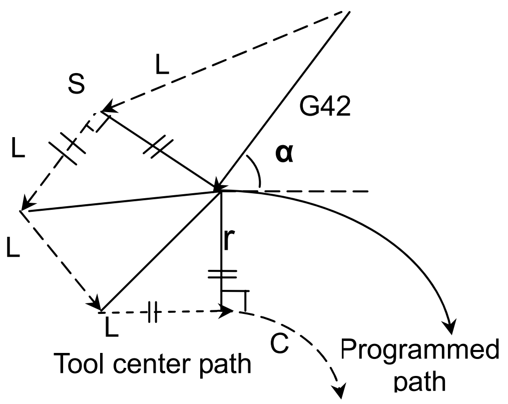


B

Linear—Linear



Linear—Arc



(d) Tool movement around the outside linear→linear at an acute angle. (α＜1°)

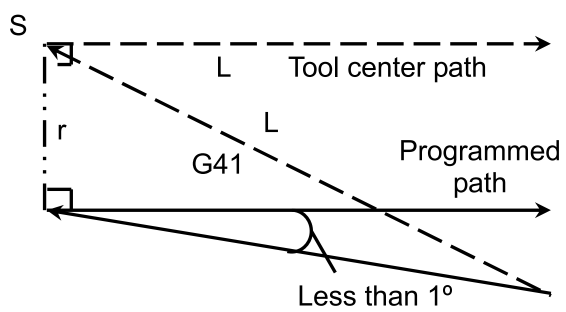


Fig. 4-7-3-2

2. Tool movement in offset mode

The invariable compensation plane is performed in the compensation mode; otherwise, the alarm may issue, and the tool stops at the same time. In the offset mode, the movement of tool is as follows:

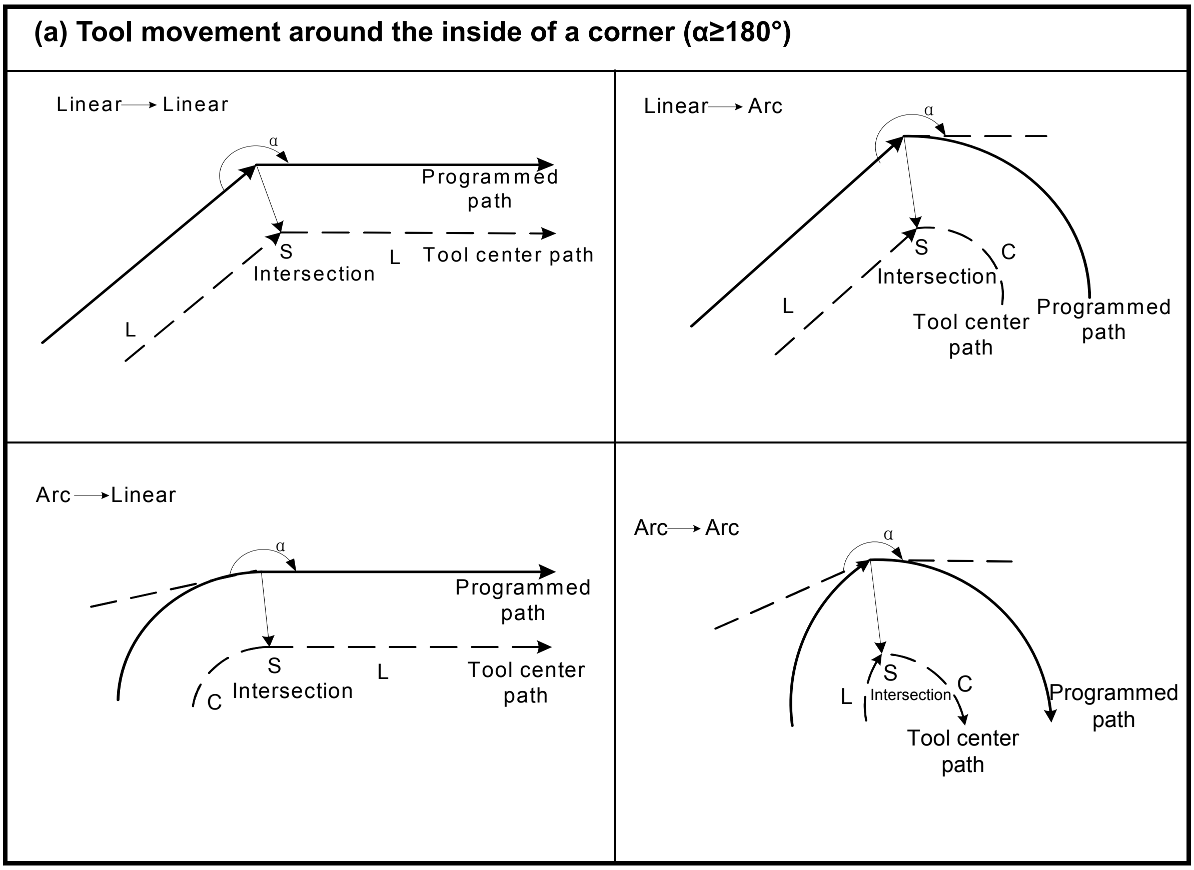


Fig. 4-7-3-3

3. When it is exceptional

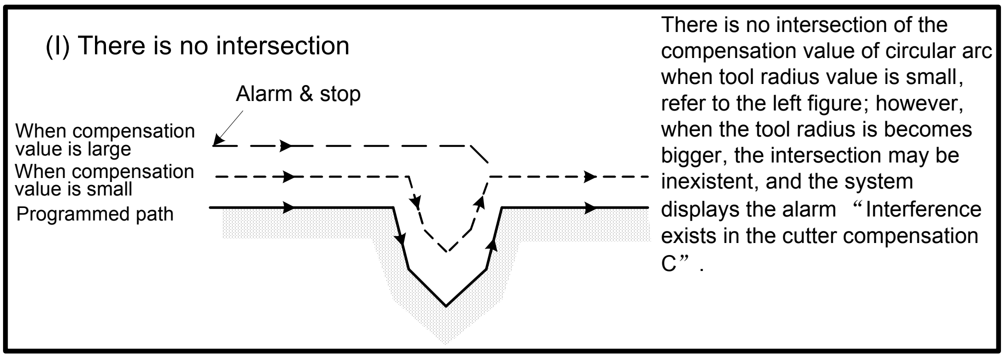


Fig. 4-7-3-4

4. The tool movement in offset cancellation mode

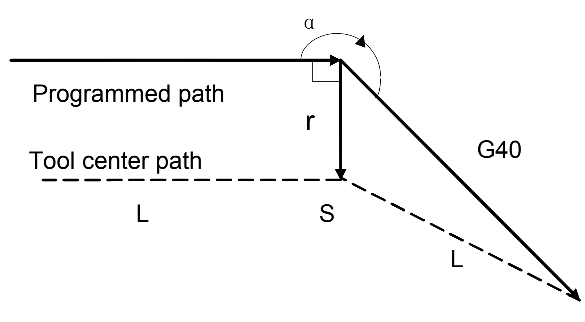
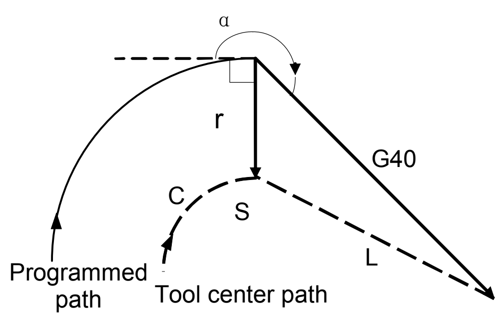
In the compensation mode, when any of the following items of the block is performed; the system then enters the compensation cancellation mode, and the movement of the block is called Compensation cancellation.

a) Code G40

b) Cutter compensation number is 0.

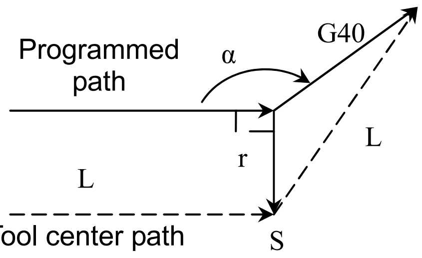
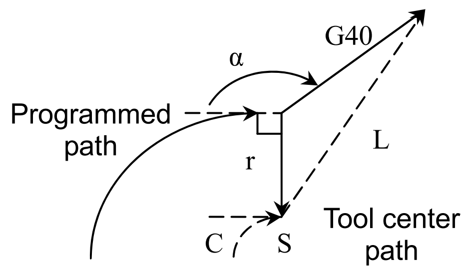
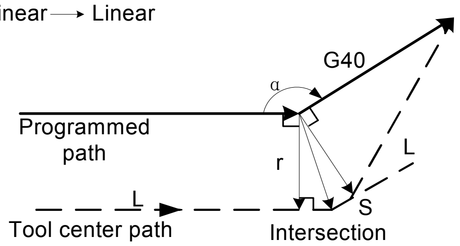
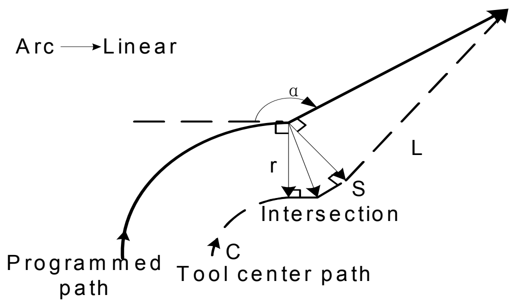
Fail to use the arc code (G03 and G02) to cancel when the compensation cancellation is performed, if the commanded arc may alarm and tool may stop.

(a) Tool movement around the inside of a corner (α≥180°)

Linear→Linear	Arc→Linear
	

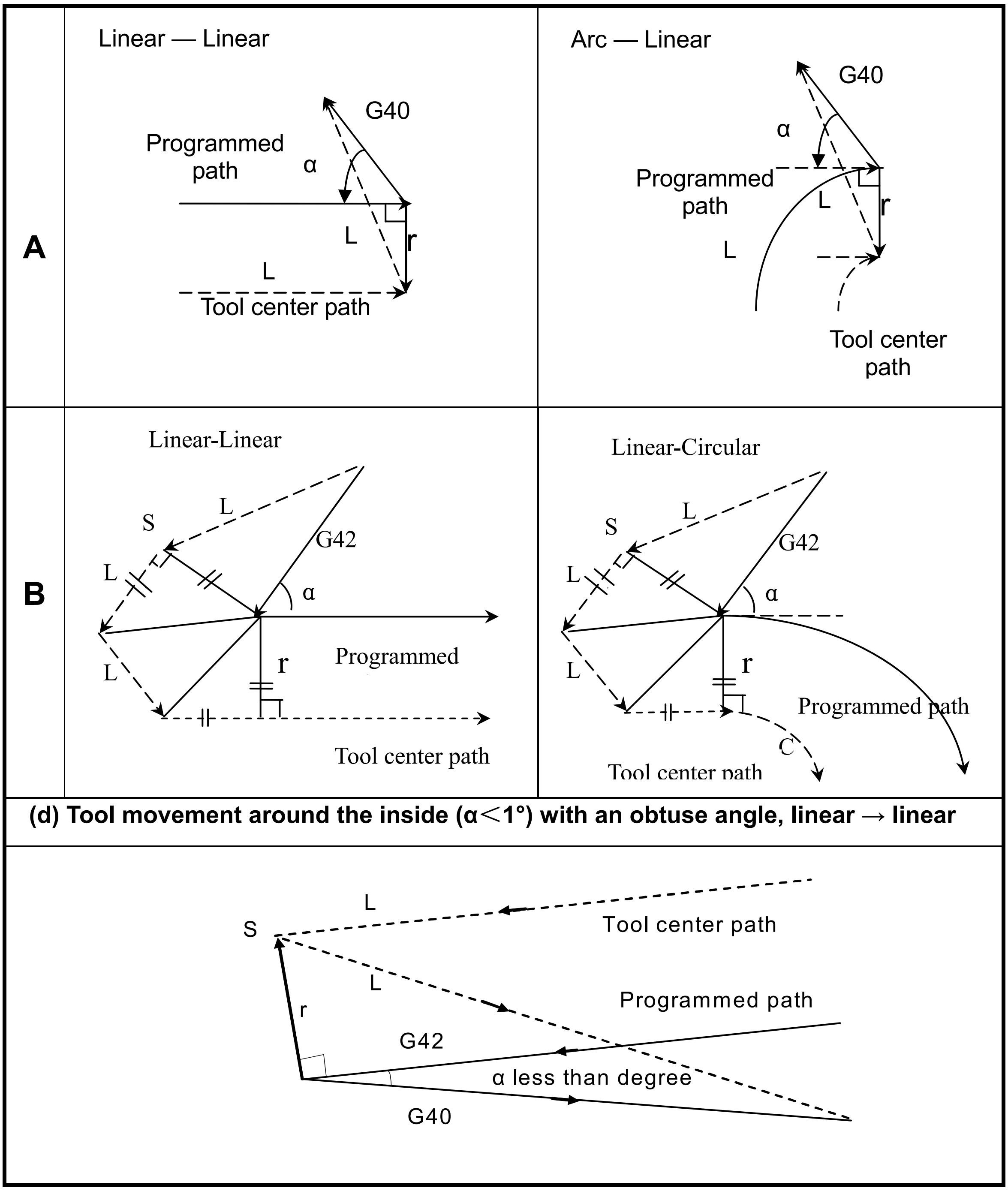
(b) Tool movement around the inside with a corner (90°≤α<180°)

Tool path in the beginning of compensation has two types A and B, which are selected by bit 0 of parameter No.: 40:

A	Linear—Linear 	Arc—Linear 
B	Linear→Linear 	

(c) Tool movement around the outside with an acute angle (α< 90°)

Tool path in the beginning of compensation has two types A and B, which are selected by bit 0 of parameter No.: 40:



(d) Tool movement around the inside (α<1°) with an obtuse angle, linear → linear

Fig. 4-7-3-5

5. Alter the compensation direction in the compensation mode

The compensation direction is determined by cutter compensation G code (G41 and G42), the symbol of compensation value is shown below:

Table 4-7-3-1

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

In the special occasion, the compensation direction can be changed in the compensation mode instead of altering at the beginning of stat and its subsequent blocks. When the compensation direction changes, there is no inner side and outer side. It is supposed to positive for the following compensation value.

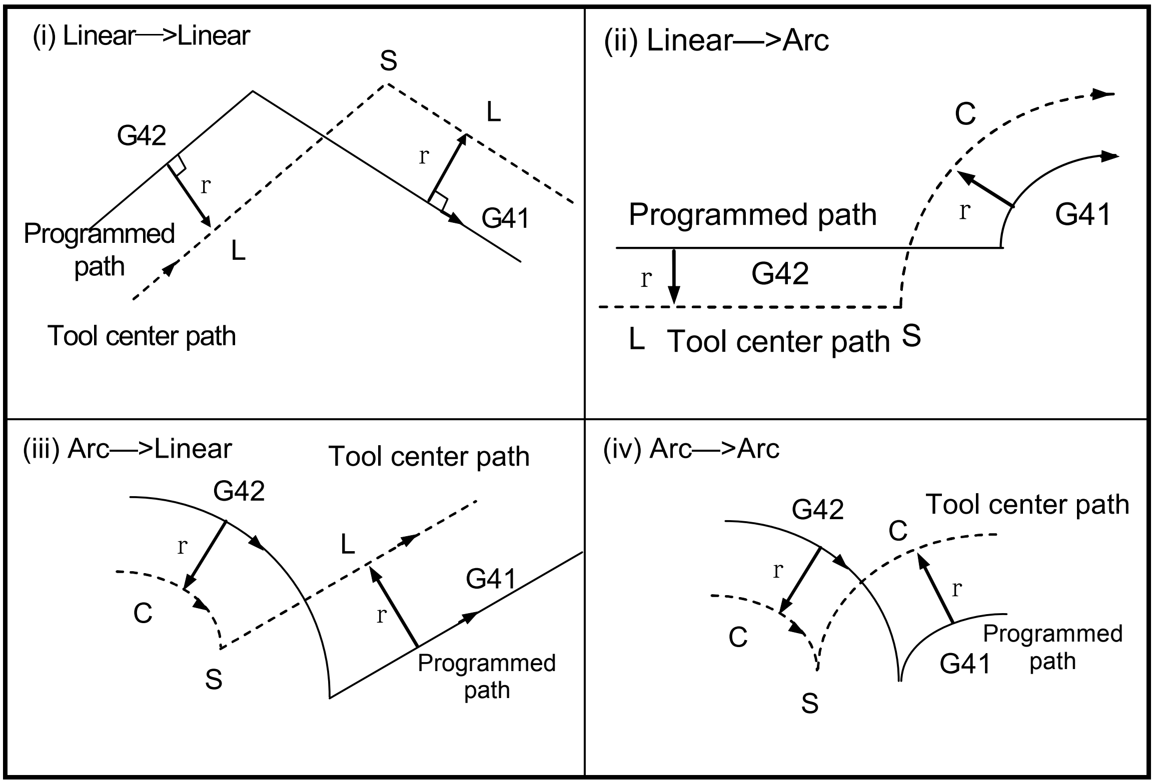


Fig. 4-7-3-6

(V) If the compensation is normally performed but there is no intersection

When the offset direction blocks A and B are changed by G41 and G42, if there is no need any intersection of compensation path, the start at the block B is vertical to the vector of the block B.

(1) Linear-----Linear

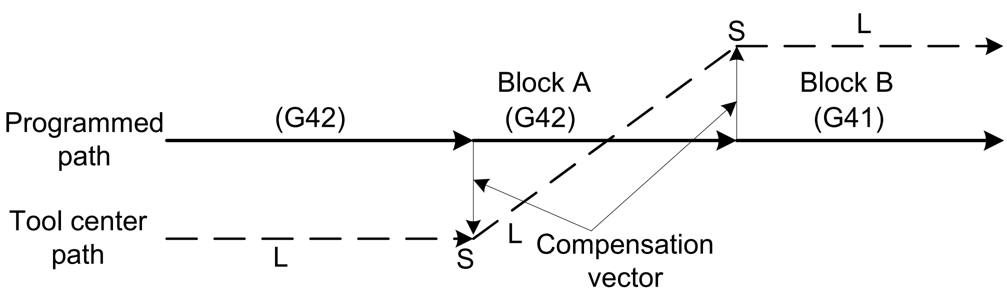


Fig. 4-7-3-7

(2) Linear-----Arc

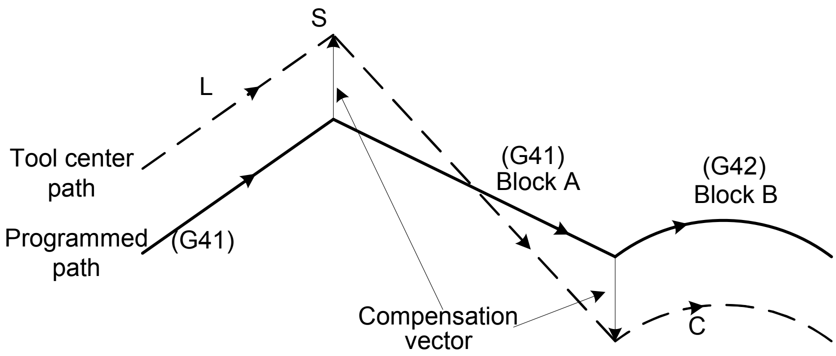


Fig. 4-7-3-8

(3) Arc-----Arc

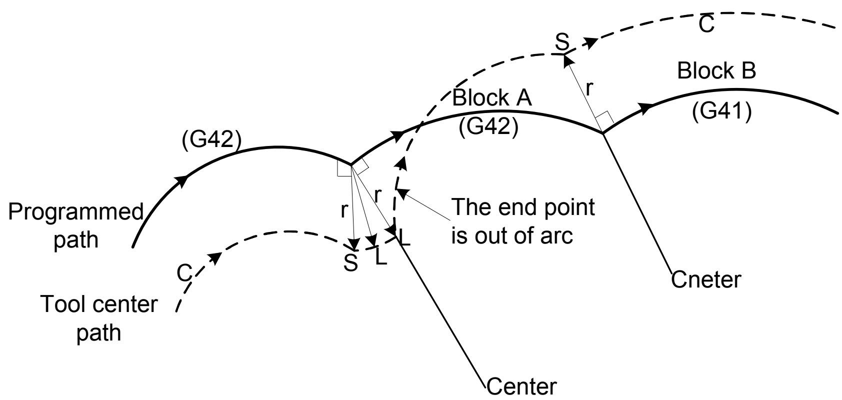


Fig. 4-7-3-9

(vi) When the cutter compensation

(vi) Generally, the following decription will not occur when cutter compensation makes the tool center path length above one circle. However, the following conditions may be generated when G41 and G42 are charge.

Arc—Arc (Linear—Arc) The system will alarm when cutter compensation changes its direction; the alarm prompts “Arc code can not cancel the tool compensation!” when tool number is regarded as D0.

Linear – Linear Tool compensation direction can be altered.

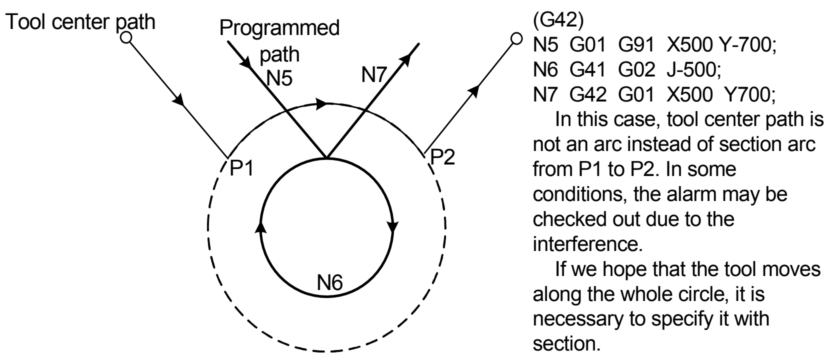


Fig. 4-7-3-10

6. Compensation cancellation in dwell

In the compensation mode, when the G28, G30 code can be specified by bit 2 of parameter No.: 40, whether the compensation is cancelled temporarily at the intermediate point.

a) G28 automatic reference position return

In the compensation mode, if the G28 is commanded, and the compensation will be cancelled at the intermediate point; the compensation mode is automatically recovered after the reference position is turned.

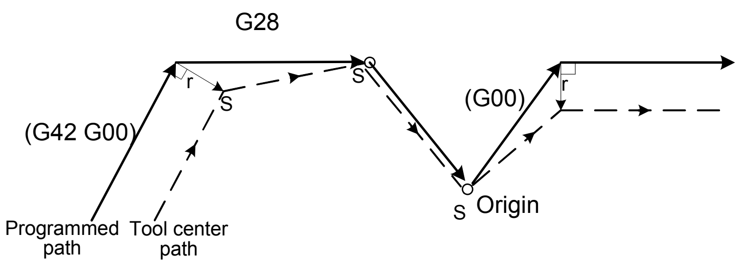


Fig. 4-7-3-11

b) G29 automatically returns from reference origin

In the compensation mode, if the G29 is specified, and the compensation will be cancelled at the intermediate point; the compensation mode is automatically recovered when returning the specified point by G29.

Immediately specify it followed with G28

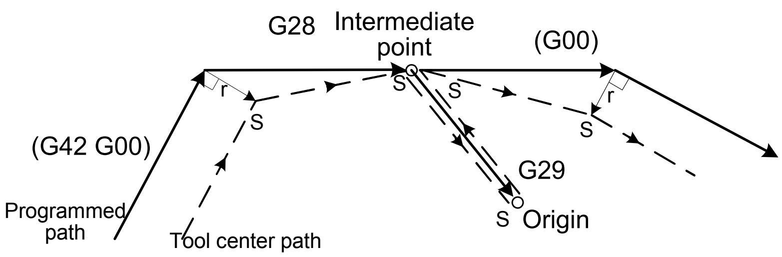


Fig. 4-7-3-12

It does not immediately specify it followed with G28:

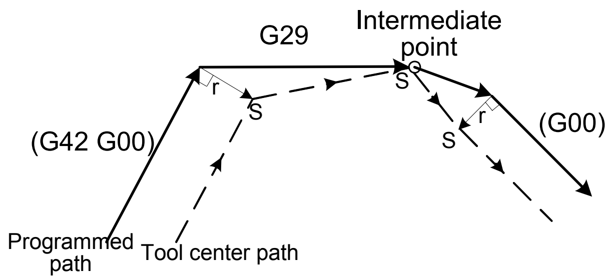


Fig. 4-7-3-13

7. Cutter compensation G code in compensation mode

When specifying the cutter compensation G codes (G41, G42) in the compensation mode, it becomes a rectangular vector with the previous block related to the movement direction and it regardless of the machining inner side and outer side. However, if this G codes are specified in arc code, the correct arc can not be gained accordingly.

When the cutter compenatin G codes (G41, G42) are changed the compensation direction, refer to the (5).

Linear-----Linear

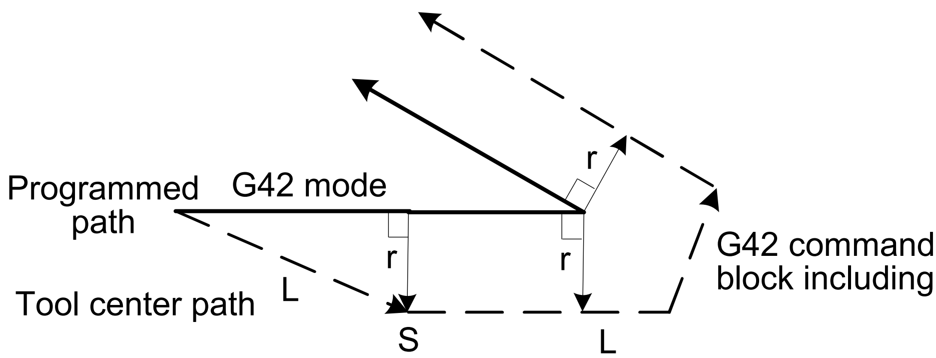


Fig. 4-7-3-14

Arc-----Linear

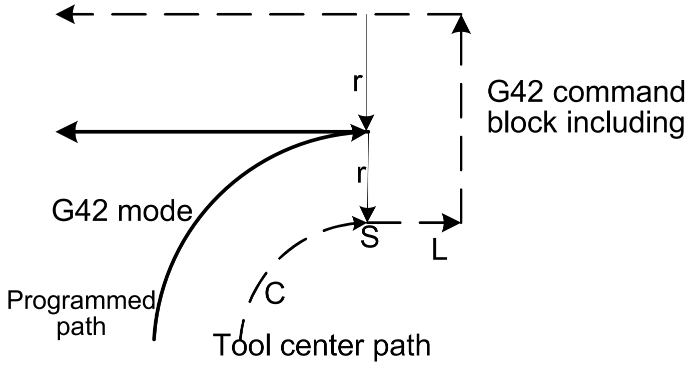


Fig. 4-7-3-15

8. A block without tool movement

There is no tool movement in the following blocks. In the blocks, it will not move even if the cutter compensation mode is enabled.

(1) M05 ； …………………… M code output

(2) S21 ； …………………… S code output

(3) G04 X10； …………… Dwell

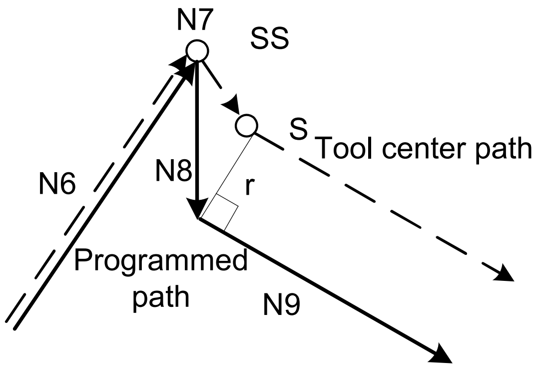
(4) (G17) Z100 ； ……………… Without movement code inside the compensation plane

(5) G90 ； ……………………… G code only

(6) G01 G91 X0； …………… Movement value is 0

a) The code at the beginning of compensation

The system will generate start-up operation at the next movement code if the start-up block does not move.



G40 G91……

·

·

N6 X100 Y100;
N7 G41 X0;
N8 Y-100;
N9 X100 Y-100;

Fig. 4-7-3-16

b) In the compensation mode command

When only one block without tool movement is specified in compensation mode, the vector and the tool center path is same as that when the command does not specify. (Refer to the item

(3) compensation mode). In this case, the block without tool movement is performed at the single block stop position.

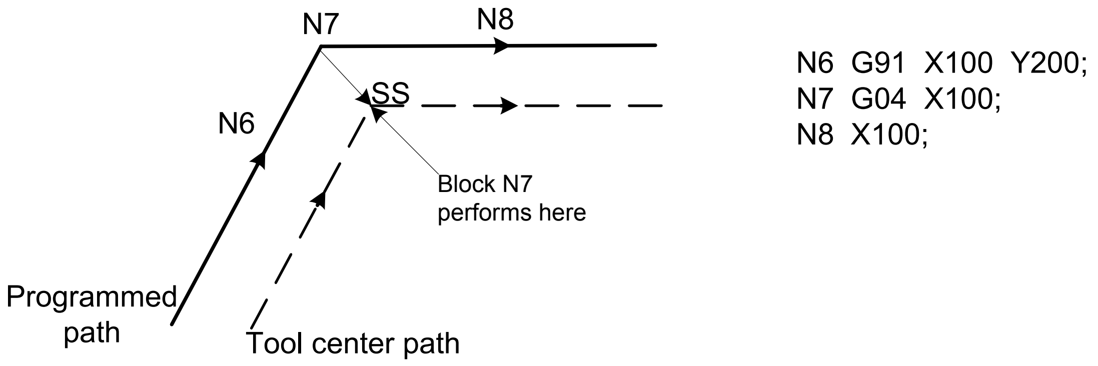


Fig. 4-7-3-17

However, when the movement value is zero, tool motion becomes the same as that when two or more blocks of without tool movement are commanded even if only one block is specified.

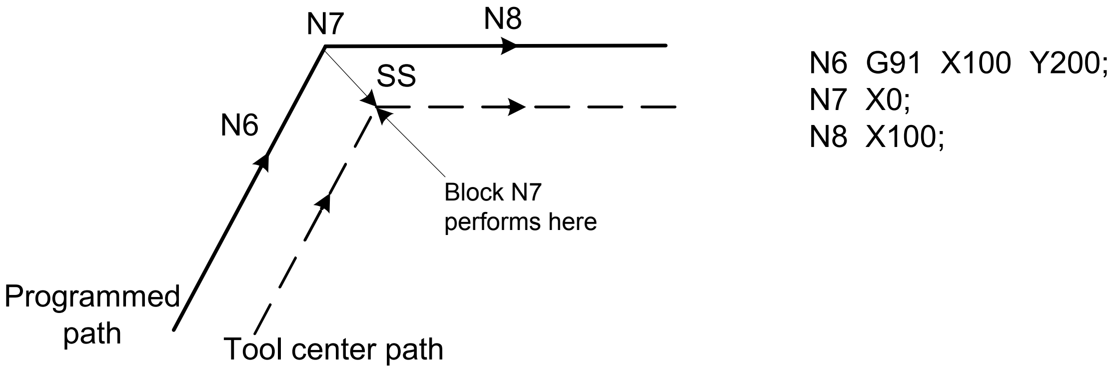


Fig. 4-7-3-18

Note: The above-mentioned block is operated based upon the G1, G42, the path is inconsistent with the figure in G0.

c) It is specified with the compensation cancellation together

When there is no tool movement of a block specified with the compensation cancellation together, it may form the length compensation, and its direction is vertical to the movement direction vector of the previous block, and this vector will be cancelled at the next command.

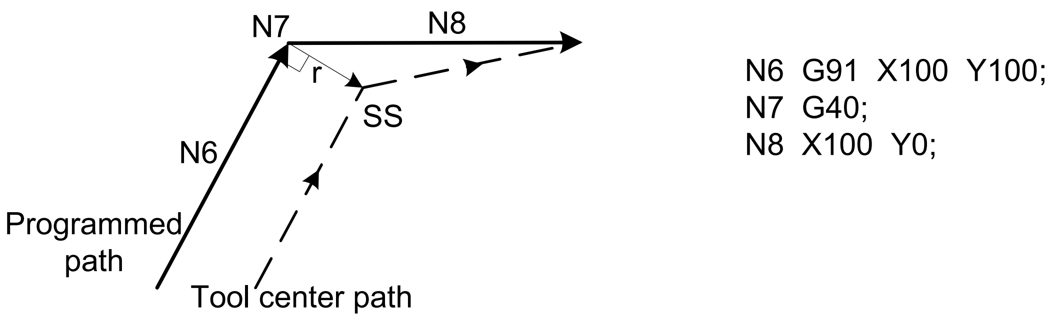


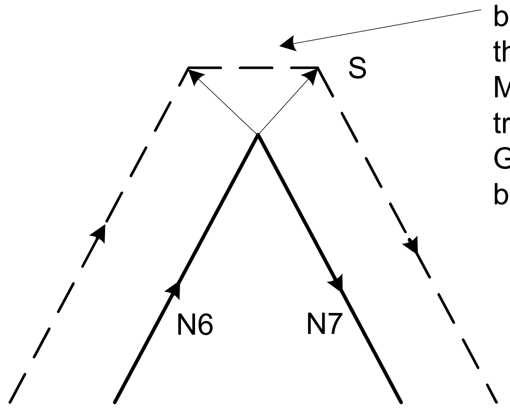
Fig. 4-7-3-19

9. Corner movement

When more than two vectors are produced at the end of a block, the tool moves linearly from one vector to another. This movement is called the corner movement.

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

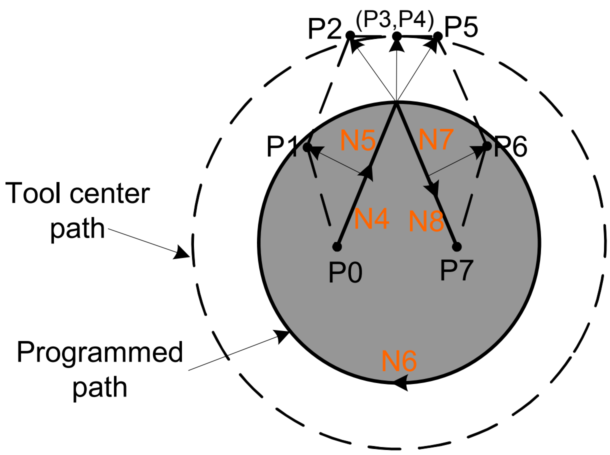
If these vectors do not coincide, a movement is generated to turn around the corner, which belongs to the previous block.



These motions are belonged to the block N6, thus, the feedrate equals to that in block N6. If the block N6 is G00 Mode, the tool is moved at the rapid traverse rate; if the block N6 is G01, G02 and G03 modes, the tool is moved based upon the cutting feedrate.

Fig. 4-7-3-20

However, if the path of the next block is semicular or more, the above-mentioned function is not performed; and its reason is as follows:



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

Fig. 4-7-3-21

If the vector is not ignored, the tool path is as follows:

P0 →P1 →P2 →P3 (circular arc) →P4 →P5 →P6 →P7

But if the distance between P2 and P3 is negligible, the point P3 is then omitted. Therefore, the tool path is as follows:

P0 →P1 →P2 →P4 →P6 →P7 The arc cutting of block N6 is ignored.

10. Interference check

Tool overcutting is called interference. The interference can check the tool overcutting in advance. If the interference is checked in the syntax inspection after loading the program, the system will then alarm. Whether the interference check is performed when the bit 6 of parameter No.: 41 is set the radius compensation.

The basis conditions of interference:

(1) The movement distance of the block for establishing the cutter compensation is less than tool radius.

(2) The tool path direction defers from the programmed path direction. (The angle of path between the 90° and 270°).

(3) When the arc machining is performed, the angle of its start and end of tool center path is great different to the one of the programmed path (more than 180°).

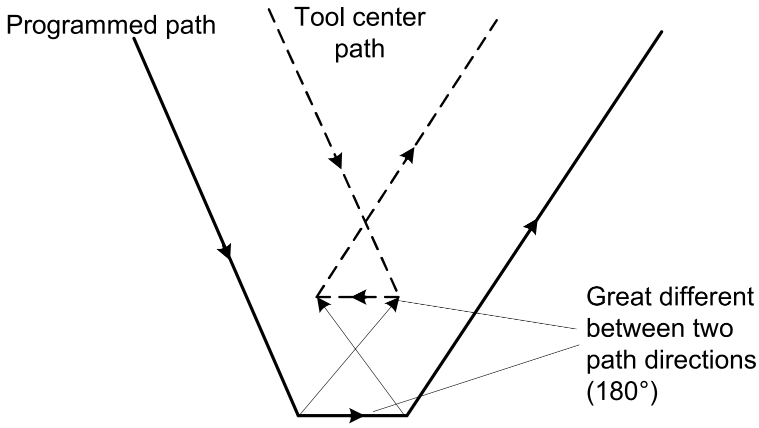


Fig. 4-7-3-22

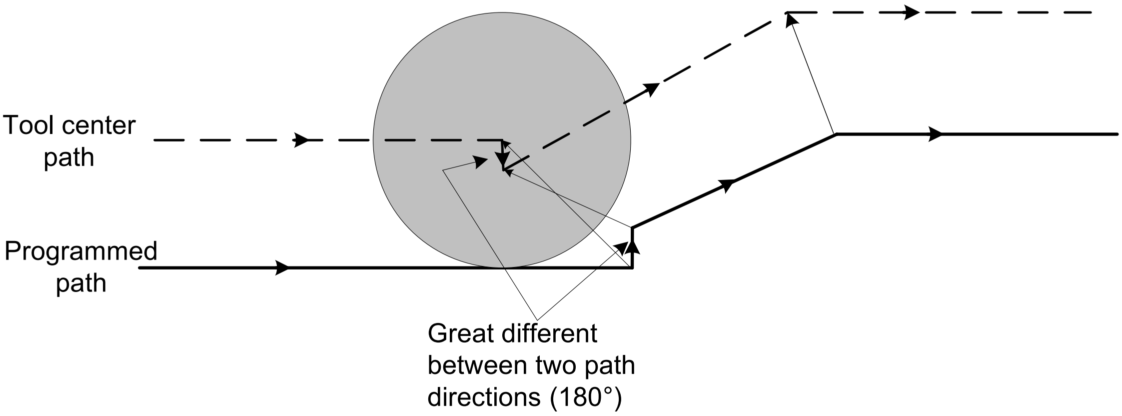


Fig. 4-7-3-23

11. Manual operation

The manual operation in tool point radius compensation, refer to the manual operation in the

Volume Operation.

12. The general precautions for compensation

a) Command compensation value

The compensation value is specified the compensation value number by D code. The D code is always enabled till another one is specified or cancelled the compensation once it is specified. D code, also, is not only used for specifying the compensation value to the cutter compensation but also for the offset value of tool offset.

b) Compensation value alteration

Usually, when the tool-change is performed, the compensation value should be altered in the cancellation mode. If the compensation value is changed in the compensation mode, a new compensation value will be calculated at then end of the block.

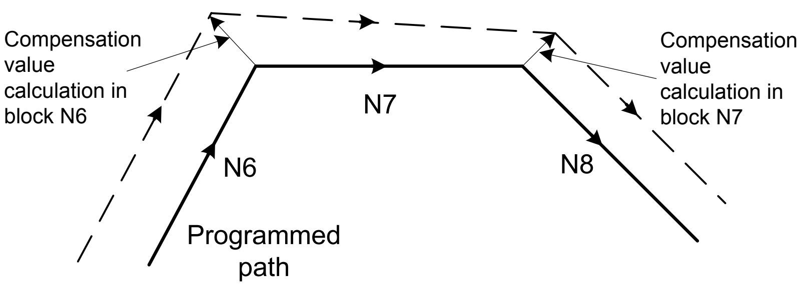


Fig. 4-7-3-24

c) The +/- of compensation and tool center path

The G41 and G42 in the program will be exchanged if the compensation value is negative (-). If the tool center moves along the workpiece outer side, the compensation value may move along the inner side, vice versa.

The following figure shows that the compensation value is (+) when the program is performed. When the tool path (Fig. a) is programmed, if the compensation value is negative (-), and the tool center moves as (Fig. b), vice versa. Therefore, the male form or female form can be cut at the same program, and its interval can be adjusted by the comensation value.

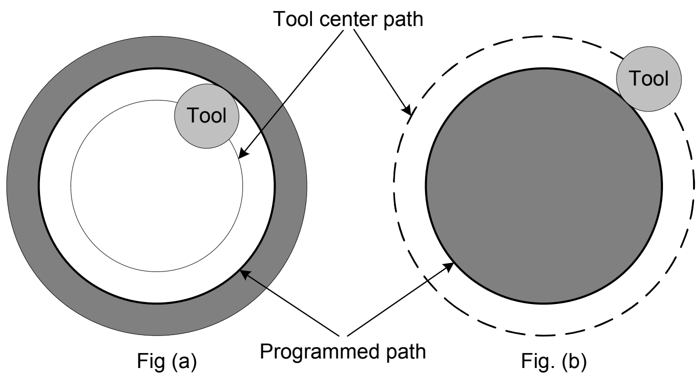


Fig. 4-7-3-25

d) Overcutting by cutter compensation

(1) When the machining is performed by the arc less than the tool radius

When the corner radius is less than the cutter radius, the internal compensation of the cutter will result in overcutting, therefore, the interference may alarm and stop before performing.

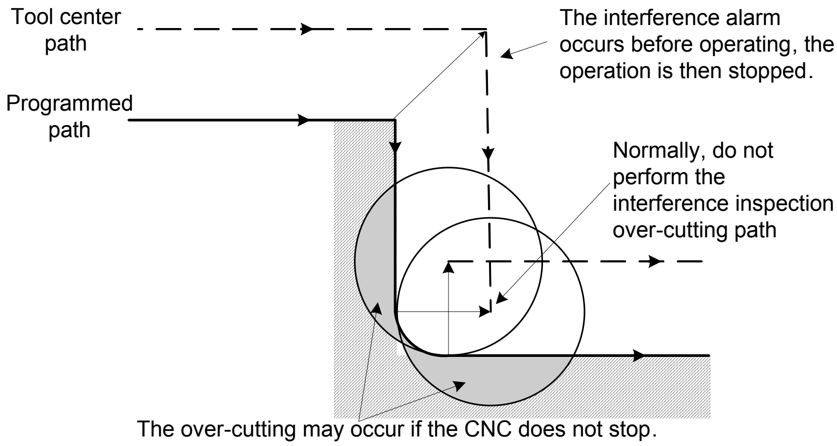


Fig. 4-7-3-26

(2) Machining groove smaller than the tool radius

When the machining is performed by the cutting-groove less than the tool radius, the cutter compensation forces the tool center path that moves along the negative direction of programmed path; the overcutting then may occur.

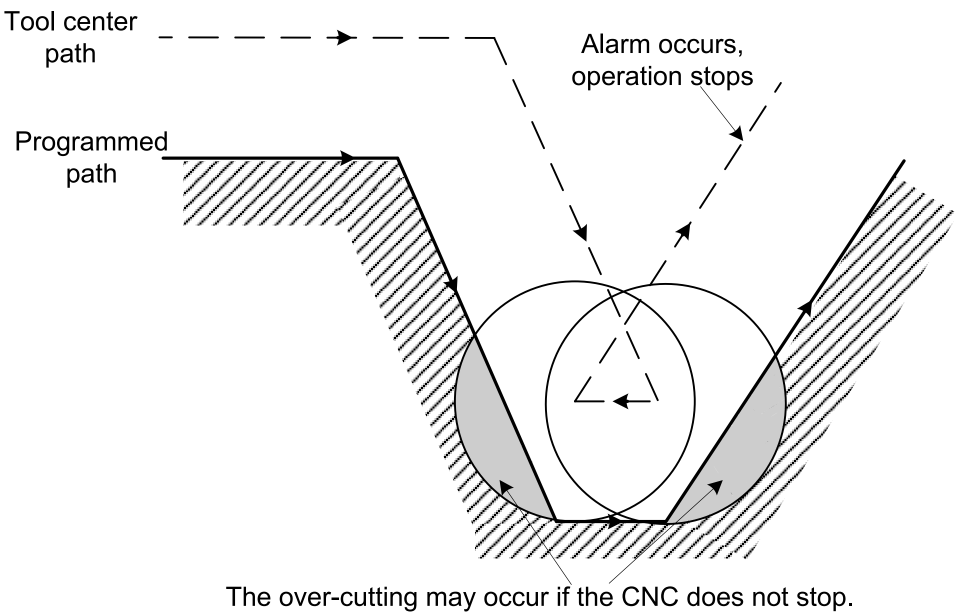


Fig. 4-7-3-27

(3) Machining section-difference smaller than the tool radius

If the section-difference smaller than the tool radius performs in program, and when the machining of this section-difference is specified by arc machining, the normal tool center path of compensation then becomes opposite to the program direction. In this case, the initial vector is omitted, and the tool moves to the 2nd vector linearly. Single block stops here. The automatic operation may continue if the machining is performed regardless of the single mode. If the section-difference is straight line, the alarm will not issue instead of cutting correctly, but the end of cutting section may be remained.

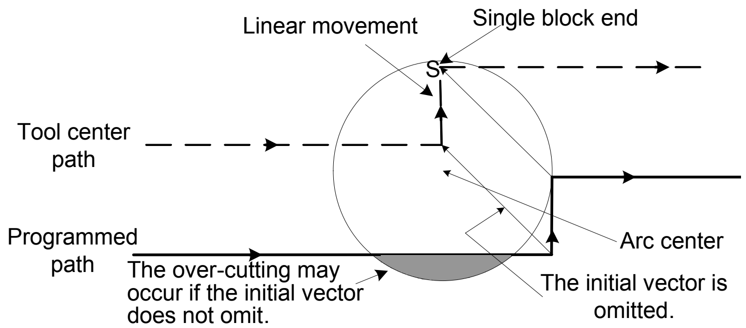


Fig. 4-7-3-28

Cutter compensation start and movement along the Z axis

Generally, when the machining is started and after the cutter compensation is enabled, tool moves a section of distance from the workpiece along with Z axis. If the movement along the Z axis is divided into rapid feed and cutting feed based upon the above-mentioned conditions, refer to the following programs:

If it is the block N3 (movement code along Z axis)

It is divided into as the follows:

N1 G91 G00 G41 X500 Y500 D01;

N3 Z-250;

N5 G01 Z-50 F1;

N6 Y100 F2;

N1 G91 G0 G41 X500 Y500 D1;
N3 G01 Z-300 F1;
N6 Y100 F2;
N6, also, enter to the buffering area when performing N3. The correction compensation between N3 and N6 is as right figure shown.

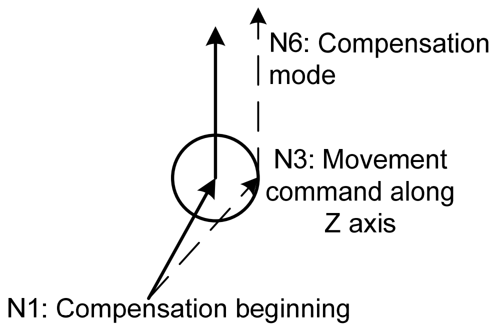


Fig. 4-7-3-29

4.7.4 Corner Offset Arc Interpolation (G39)

Format: G39

Function: By specifying G39 in offset mode during cutter compensation, corner offset circular arc compensation can be performed. The radius of corner compensation equals to the compensation value. Whether the corner arc in the radius compensation is enabled by setting the bit 5 of parameter No.: 41.

Explanation:

1. When the G39 code is specified, corner arc interpolation in which the radius equals to the compensation value can be performed.

2. G41 or G42 preceding the code determines whether the arc is CW or CCW, G39 is one-shot G code.

3. When G39, is programmed, the arc at the corner is formed so that the vector at the end point of the arc is perpendicular to the start point of the next block. Refer to the following figure:

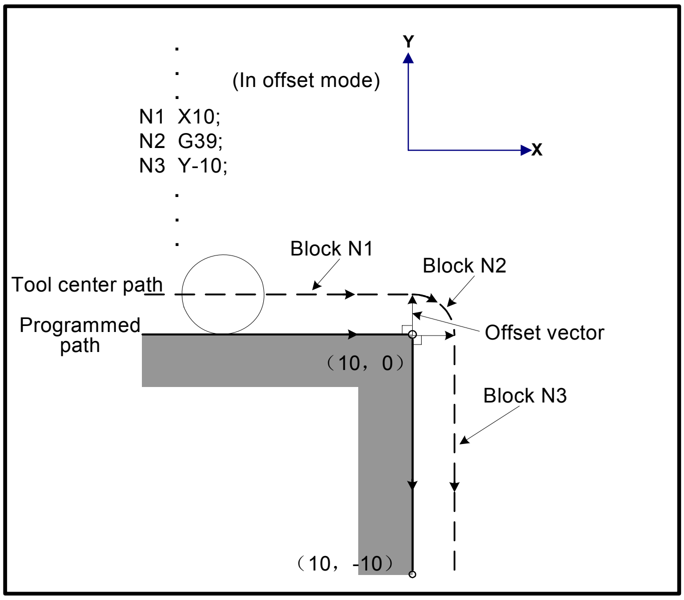


Fig. 4-7-4-1 G39

4.7.5 Tool Compensation Value, Entering Compensation Number From Program (G10)

Format: **G10 L10 P_ R_ ;** Geometric compensation value of H code
G10 L12 P_ R_ ; Geometric compensation value of D code
G10 L11 P_ R_ ; Wear compensation value of H code
G10 L13 P_ R_ ; Wear compensation value of D code

P : Tool compensation number.

R : Tool compensation value in the absolute value code (G90).

Tool compensation value in the absolute value code (G91) adds the value of the specified tool compensation number (The sum is tool compensation value)

Explanation: Available input range of tool compensation value:

Geometric compensation: metric input -999.999mm~+999.999mm; Inch input -39.3700inch~+39.3700inch.

Wear compensation: metric input -400.000mm~+400.000mm (Evaluate the No.267 data parameter setting); Inch input -39.3700inch~+39.3700inch (Evaluate 1/25.4 of the No.267 data parameter setting).

Note 1: The maximum value of the wear compensation is restricted by data parameter P267.

4.8 Feed G Code

4.8.1 Feed Mode G64/G61/G63

Format:

Exact stop G61

Tapping mode G63

Cutting mode G64

Functions:

Exact stop method G61: This function is always enabled once specifying before G62, G63 or G64 is specified. Tool decelerates and performs in-position detection at the end of the block, and then perform the next block.

Tapping method G63: This function is always enabled once specifying before G61, G62 or G64 is specified. Tool does not decelerate instead of executing next block at the end of the block. When specifying G63, both the feedrate override and feed hold are disabled.

Cutting method G64: This function is always enabled once specifying before G61, G62 or G63 is specified. Tool does not decelerate instead of executing next block at the end of the block.

Explanations:

1. Without parameter format.

2. G64 is the default feed mode of the system; the end of the block does not decelerate instead of executing next block.

3. The destination of the in-position detection in exact mode is check whether the servo motor is reached within the specified position range.

4. In the exact stop mode, the tool paths are different between the cutting and tapping methds.

Refer to the following Fig. 4-8-1-1

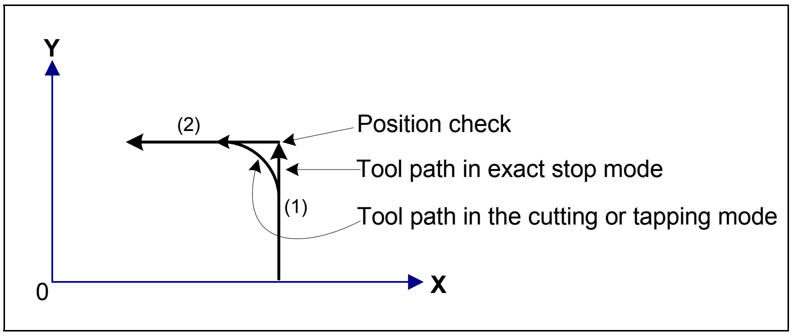


Fig. 4-8-1-1 Tool path from block 1 to block 2

4.8.2 Automatic Corner Override (G62)

Format: G62

Function: Once the Auto corner override method G62 is specified, this function is always enabled before G61, G62 or G63 is specified. When the tool moves along the inner corner during the cutter compensation, the cutting feed override is performed to restricting the cutting value within the unit time; in this case, the fine surface accuracy can be machinined accordingly.

Explanatins:

1. When cutter compensation is performed, the movement of the tool is automatically decelerated and reduced the load on the cutter at an inner corner and internal circular arc area, so that it produces a smoothly machined surface.

2. Whether the automatic corner override function is available is set by bit 7 of parameter No.: 16. Control the automatic corner deceleration function (0: Angle control, 1: Speed difference control) by bit 2 of parameter No.: 15.

3. When specifying G62 and machining inner corner with cutter compensation function, automatically adjust the feedrate both ends of corner. There are four inner corners, refero to Fig. 4-8-2-1; wherein, 2°≤θ≤θp≤178°; θp is set by data parameter **P144**.

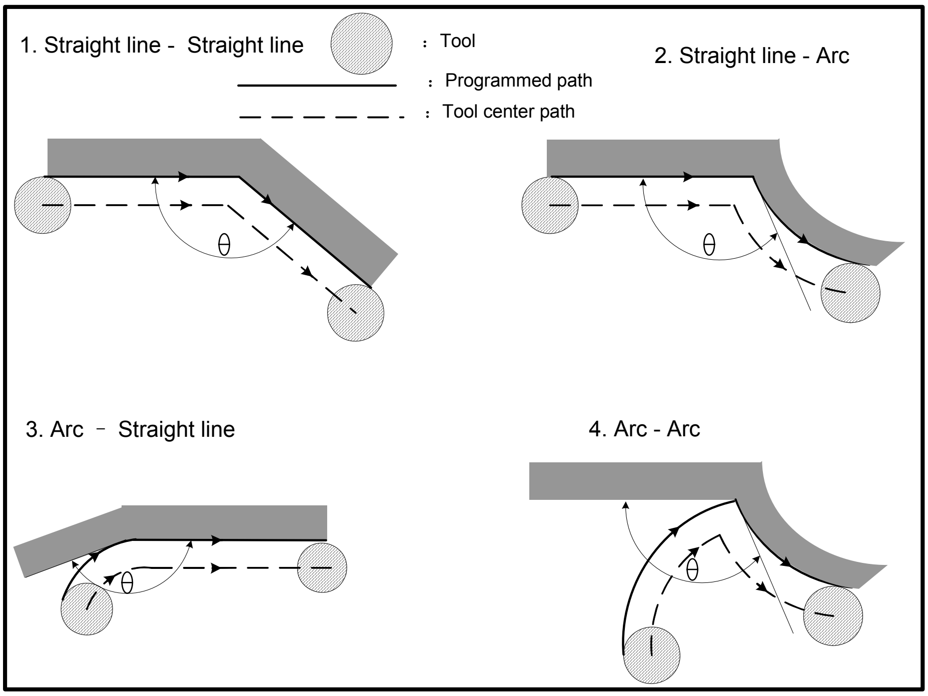


Fig. 4-8-2-1

4. 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 distances from the points on the cutter center path to the corner. Refer to 4-8-2-2, wherein, Ls+Le≤2mm.

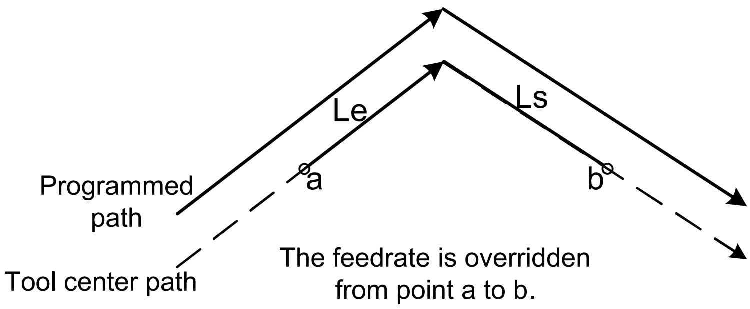


Fig. 4-8-2-2 Straight line to straingt line

5. When a programmed path consists of two arcs, the feedrate is overridden if the start and end points are in the same quadrant or in adjacent quadrants. The lowest feedrate is decelerated by controlling automatic corner using data parameter P145, refer to the Fig. 4-8-2-3.

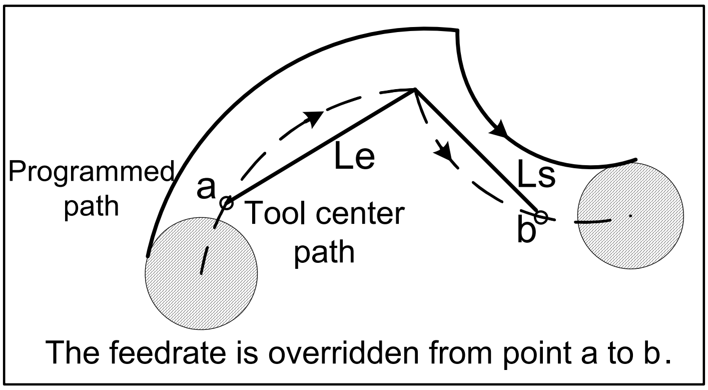


Fig. 4-8-2-3　Arc to arc

6. Regarding a program is both from straight line to arc and from arc to straight line; the feedrate is overridden from point a to b and from c to d. Refer to the Fig. 4-8-2-4.

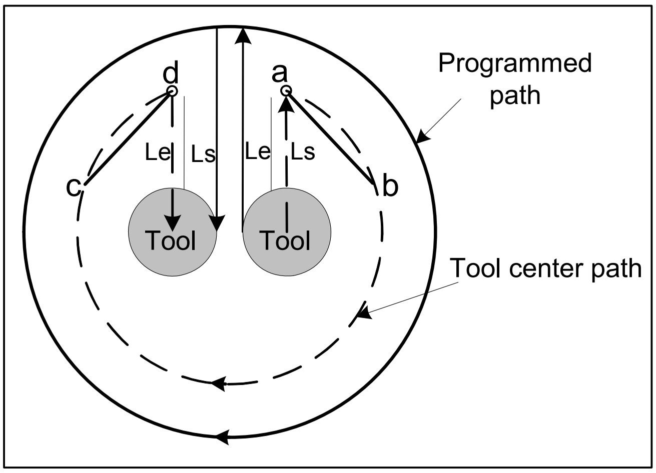


Fig. 4-8-2-4 Straight line to arc, arc to straight line

Limitations:

1. Override for inner corner is disabled during acceleration/deceleration before interpolation.
2. Override for inner corner is disbled if the corner is preceded by a start-up block or followed by a blocking including G41 or G42.
3. Inner cornder is not performed if the offset is zero.

4.9　Macro Function G Code

4.9.1　User Macro Program

One function carried out by one group code is registered to memory in advance, like subprogram. This functions can be indicated by one code, which can be performed only write the represented code in program. The group codes are called user macro program body, and the represented codes are regarded as "User Macro Code". The user macro program body, sometimes, is abbreviated as Marco Program, and the user macro code is also treated as Macro program call code.

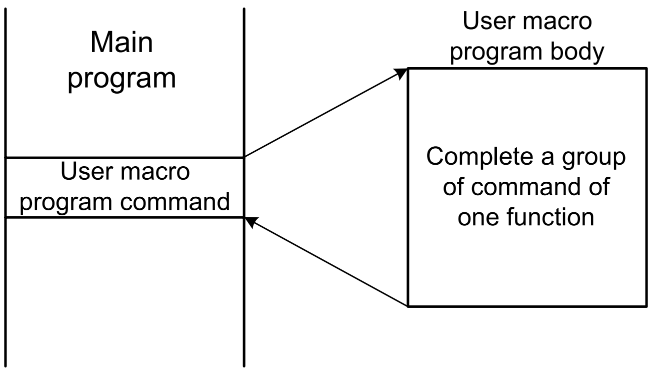


Fig. 4-9-1-1

The variable can be used in user macro program body. The calculation can be performed among variables, and the variable also can be assigned by macro code.

4.9.2 Macro Variable

Both the common CNC command and the variable, as well the calculation and code shifting can be used in user marco program.

User macro program ends by M99 from the beginning of program number.

```
O0066;                          Program number
G65  H01  ……;                   Calculation command
G90  G00  X#101  ……;  CNC command used with variable
……
……
……
G65  H82  ……;                   Command shifting
……
……
M99;                  User macro program body end
```

Fig. 4-9-2-1 The composition of user macro program body

1. Usage of variable

The parameter value in user macro program body can be specified by variable. The variable value can be assigned by main program or set by LCD/MDI, as well, evaluate the value to be calculated when performing the user macro program body.

Multiple variables can be used, which can be distinguished from variable number.

(1) The indication of variable

The variable can be indicated by the consecutive variable number followed with #, the format shows below:

#i (i = 1, 2, 3, 4 ……)

(For example) #5， #109， #1005

(2) The quotation of variable

The numerical value followed with the parameter can be replaced by variable.

(For example) F#103 When #103 = 15, it is identical with the F15 command.

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

Notice: 1. Parameter word O and N (Program number and Seqence number) can not be quoted the variable. And it can not be programmed by O#100, N#120.

2. It can not be used if exceeds the top code value specified by parameter value. When #30 = 12, M#30 exceeds the Max. code value.

3. The display and setting of variable value: The variable value can be displayed on LCD and set by MDI mode.

2. Types of variable

Variable can be divided into null, local, common and system variables, which are different in the purpose and character.

(1) Null variable #0: (This variable is always null, no value can be assigned to this variable)

(2) Local variable #1～#50: Local variables can only be registered the data in macro program; bit 7 of parameter No.: 52 can be reset or whether it is eliminated after ESP. When a macro program is called, arguments are assigned to local variables.

(3) Common variables #100～#199, #500～#999: Bit 6 of parameter No.:52 can be reset or whether it is eliminated the common variables #100～#199 after ESP.

Common variables are public one in main program and each user macro program called by main program; that is, the used variable #i in one user macro program is identical with the one of other macro programs. Therefore, the common variable #i in one macro program calculation result can be used in other macro programs.

The purpose of common variable does not specify in system, user can be used it freely.

Table 4-9-2-1

Variable number	Variable type	Function
# 100～ # 199	Common variable	The variable numbers are eliminated when the power is turned off and reset to “Null” when the power is turned on.

＃500～＃999		The data are registered in file, and they will not absent even if the power is turned off.

(4) System variables: System variables are used to read and write a variety data when CNC is performed, which are separately shown below:

1) Interface input signal #1000 --- #1015 (Read PLC based upon the bit and input the signal to system, that is G signal)
#1032 (Read PLC based upon byte and input the signal to system, that is G signal)

2) Interface output signal #1100 --- #1115 (Write system output based upon bit to the signal of PLC, that is F signal)
#1132 (Write system output based upon byte to the signal of PLC, that is F signal)

3) Tool length compensation value #1500 --- #1755 (Read-write)

4) Length wear compensation value #1800 --- #2055 (Read-write)

5) Cutter compensation value #2100 --- #2355 (Read-write)

6) Radius wear compensation value #2400 --- #2655 (Read-write)

7) Alarm #3000

8) User data table #3500 --- #3755 (Read only)

9) Modal information #4000 --- #4030 (Read only)

10) Position information #5001 --- #5030 (Read only)

11) Workpiece zero offset value 5201 --- #5235 (Read-write)

12) Additional workpiece coordinate system #7001 --- #7250 (Read-write)

3. The details of system variable

1) Modal information

Table 4-9-2-2

Variable No.	Function	Group No.
#4000	G10,G11	Group 00
#4001	G00,G01,G02,G03	Group 01
#4002	G17,G18,G19	Group 02
#4003	G90,G91	Group 03
#4004	G94,G95	Group 04
#4005	G54,G55,G56,G57,G58,G59	Group 05
#4006	G20,G21	Group 06
#4007	G40,G41,G42	Group 07
#4008	G43,G44,G49	Group 08

#4009	G22,G23,G24,G25,G26 G32,G33,G34,G35,G36,G37,G38 G73,G74,G76,G80,G81,G82,G83,G84,G85,G86,G87,G88,G89	Group 09
#4010	G98,G99	Group 10
#4011	G15,G16	Group 11
#4012	G50,G51	Group 12
#4013	G68,G69	Group 13
#4014	G61,G62,G63,G64	Group 14
#4015	G96,G97	Group 15
#4016	To be extended	Group 16
#4017	To be extended	Group 17
#4018	To be extended	Group 18
#4019	To be extended	Group 19
#4020	To be extended	Group 20
#4021	To be extended	Group 21
#4022	D	
#4023	H	
#4024	F	
#4025	M	
#4026	S	
#4027	T	
#4028	N	
#4029	O	
#4030	P (The additional workpiece coordinate system selection at present)	

Note 1: The P code is the selected additional workpiece coordinate system at present.

Note 2: When performing G#4002, the value gained from #4002 is 17, 18 or 19.

Note 3: Modal information can not be wrote, which is read only.

2) Current position information

Table 4-9-2-3

Variable No.	Position Information	Relevant Coordinate System	Read Operation in Moving	Tool Compensation Value
#5001	Block end position along X axis (ABSIO)	Workpiece coordinate system	Can	Take no consider of the tool point position
#5002	Block end position along Y axis (ABSIO)			

#5003	Block end position along Z axis (ABSIO)			(program specified position)
#5004	Block end position along 4th axis (ABSIO)			
#5006	Block end position along X axis (ABSMT)	Machine tool coordinate system	Can not	Conside of the tool reference point position (Machine coordinate)
#5007	Block end position along Y axis (ABSMT)			
#5008	Block end position along Z axis (ABSMT)			
#5009	Block end position along 4th axis (ABSMT)			
#5011	Block end position along X axis (ABSOT)	Workpiece coordinate system		
#5012	Block end position along Y axis (ABSOT)			
#5013	Block end position along Z axis (ABSOT)			
#5014	Block end position along 4th axis (ABSOT)			
#5016	Block end position along X axis (ABSKP)		Can	
#5017	Block end position along Y axis (ABSKP)			
#5018	Block end position along Z axis (ABSKP)			
#5019	Block end position along 4th axis (ABSKP)			
#5021	Tool length compensation value along X axis		Can not	
#5022	Tool length compensation value along Y axis			
#5023	Tool length compensation value along Z axis			
#5024	Tool length compensation value along 4th axis			
#5026	Servo position compensation along X axis			

#5027	Servo position compensation along Y axis			
#5028	Servo position compensation along Z axis			
#5029	Servo position compensation along 4th axis			

Note 1: ABSIO: The end coordinate system of the end of previous block in workpiece coordinate system.
Note 2: ABSMT: The current machine tool coordinate system position in its system.
Note 3: ABSOT: The current coordinate position in workpiece coordinate system.
Note 4: ABSKP: The skip signal in G31 block is with the enabled position in workpiece coordinate system.

3) Zero offset values of workpiece and addition:

Table 4-9-2-4

Variable No.	Function
#5201 … #5204	External workpiece zero point offset value along the 1st axis … External workpiece zero point offset value along the 4th axis
#5206 … #5209	G54 workpiece zero offset value along the 1st axis … G54 workpiece zero offset value along the 4th axis
#5211 … #5214	G55 workpiece zero offset value along the 1st axis … G55 workpiece zero offset value along the 4th axis
#5216 … #5219	G56 workpiece zero offset value along the 1st axis … G56 workpiece zero offset value along the 4th axis
#5221 … #5224	G57 workpiece zero offset value along the 1st axis … G57 workpiece zero offset value along the 4th axis
#5226 … #5229	G58 workpiece zero offset value along the 1st axis … G58 workpiece zero offset value along the 4th axis
#5231 … #5234	G59 workpiece zero offset value along the 1st axis … G59 workpiece zero offset value along the 4th axis
#7001 …	G54 P1 workpiece zero offset value along the 1st axis …

VOLUME I PROGRAMMING

#7004	G54 P1 workpiece zero offset value along the 4th axis
#7006 … #7009	G54 P2 workpiece zero offset value along the 1st axis … G54 P2 workpiece zero offset value along the 4th axis
#7246 … #7249	G54 P50 workpiece zero offset value along the 1st axis … G54 P50 workpiece zero offset value along the 4th axis

4. Local variable

The corresponding relationships between addresses and local variables are shown below:

Table 4-9-2-5

Argument Add.	Local variable No.	Argument Add.	Local variable No.
A	#1	Q	#17
B	#2	R	#18
C	#3	S	#19
I	#4	T	#20
J	#5	U	#21
K	#6	V	#22
D	#7	W	#23
E	#8	X	#24
F	#9	Y	#25
M	#13	Z	#26

Note 1: The assignment is performed by that the English letter adds the numerical value, the rest of 20 letters can be assigned to argument other than the G, L, O, N, H and P. Each letter only can be assigned once, from A-B-C-D… to X—Y—Z, and the assignment can be performed regardless of the alphabet sequence, and the addresses without assignment can be omitted.

Note 2: There is not alternative other than to specify the G65 before using any argument.

5. The precautions about the user macro program body

1) Input by keyboard

Press the # followed with the parameter words G, X, Y, Z, R, I, J, K, F, H, M, S, T, P and Q, and then the # is inputted.

2) The calculation can be commaned and the code can be shifted in MDI state.

3) The H, P, Q and R of the calculation and shifting codes are regarded as the parameter specified by G65 whenever before or after G65.

H02 G65 P#100 Q#101 R#102 ;   Correct

N100 G65 H01 P#100 Q10 ;   Correct

VOLUME I PROGRAMMING

4) The input range of variable can not be exceeded the available 15-digit; its calculation results should be less than 9 integer digits; the manual input range of variable are the enabled 8-digit.
5) The calculation result of variable can be performed by decimal, and its accuracy is 0.0001. The rest of calculations will not discard the decimal other than the H11 (OR calculation), H12 (AND calculation), H13 (NON calculation) and H23 (Remainder calculation) may omit the decimal in variable during calculating.
For example:

#100 ＝ 35, #101 ＝ 10, #102 ＝ 5

#110 ＝ #100÷#101 (＝3.5)

#111 ＝ #110×#102 (＝17.5)

#120 ＝ #100×#102 (＝175)

#121 ＝ #120÷#101 (＝17.5)

6) The performance time of calculation and shifting codes are variable according to the conditions; generally, the average value can be considered as 10ms.
7) When the variable does not define, the variable may become “Null”. Variable #0 is always null, which can be wrote only instead of reading.

a. Quotation

When an undefined variable is quoted, the address, itself, is also ignored.

For example:

When the variable #1 is 0, and the variable #2 is null, the performance result of G00X#1 Y#2 is G00X0;

b. Operation

<Vacant> is the same as 0 except when evaluating by <vacant>.

Table 4-9-2-6

When #1=<vacant>	When #1=0
#2=#1 ↓ #2=<vacant>	#2=#1 ↓ #2=0
#2=#1*5 ↓ #2=0	#2=#1*5 ↓ #2=0
#2=#1+#1 ↓ #2=0	#2=#1+#1 ↓ #2=0

c. Conditional expressions

<Vacant> deffers from 0 only for EQ and NE.

Table 4-9-2-7

When #1=<vacant>	When #1=0
#1 EQ #0 ↓ Established	#1 EQ #0 ↓ Not established
#1 NE #0 ↓ Not established	#1 NE #0 ↓ Established
#1 GE #0 ↓ Established	#1 GE #0 ↓ Established
#1 GT #0 ↓ Not established	#1 GT #0 ↓ Not established

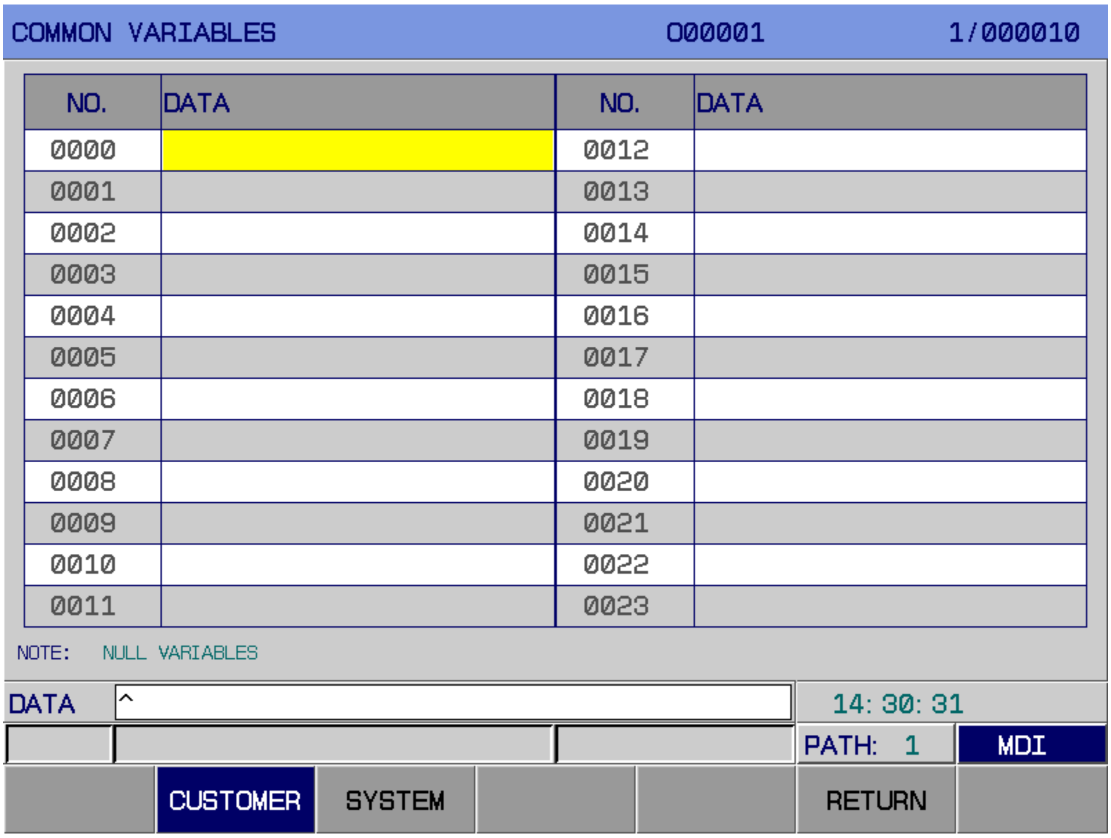


Fig. 4-9-2-2

When the value of a variable is blank, the variable is null.

4.9.3 User Macro Program Call

When G65 is specified, the user macro program specified at address P is called, and the data can be passed to the custom macro program by argument.

Format:

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

Numbers of calling (L□□□□)

The program number of called macro program body (P□□□□□)

The program number of user macro program can be specified by address P followed with G65. L specifies the call times of macro program; and the argument passes the data to macro program.

Specify the repeated times from 1 to 9999 after address L when a number of repititions are required. 1 is assumed when L is omitted.

By using argument specification, values are assigned to corresponding local variables.

Note 1: When the subprogram number specified by address P can not be indexed, an alarm (PS 078) occurs.

Note 2: The subprograms from No. 90000~9999 is the reserved program by system, when user calls this subprogram, the system can execute the conent of the subprogram; however, the cursor may always stop at the G65 code section, and the main program content will always show on the program interface. (The content of subprogram can be displayed by modifying the bit 4 of parameter No.: 27.

Note 3: Never attempt to call the macro program in DNC mode.

Note 4: Up to 5-layer can be nested for the macro program calling.

4.9.4 User Macro Program Function A

1. Common form:

G65 Hm P#i Q#j R#k ;

m: 01～99 means the function of operation code or the shifting code.

#i: The variable name for registering the operation result

#j: The variable name 1 for operating. It also can be regarded as constant; and the constant can be expressed directly regardless of the #.

#k: The variable name 2 for operating. It also can be treated as constant.

Meaning: #i = #j ○ #k

Operation symbol, it is specified by Hm.

(For example) P#100 Q#101 R#102……#100 = #101 ○ #102 ;

P#100 Q#101 R15 ……#100 = #101 ○ 15 ;

P#100 Q-100 R#102……#100 = -100 ○ #102

The H code specified by G65 is without any effect for selecting the offset value.

G code	H code	Function	Defintion
G65	H01	Assignment	#i ＝ #j
G65	H02	Plus	#i ＝ #j + #k
G65	H03	Minus	#i ＝ #j - #k
G65	H04	Multiplication	#i ＝ #j × #k
G65	H05	Division	#i ＝ #j ÷ #k
G65	H11	Logic plus (OR)	#i ＝ #j OR #k
G65	H12	Logic multiplication (AND)	#i ＝ #j AND #k
G65	H13	Exclusive OR	#i ＝ #j XOR #k
G65	H21	Square root	#i ＝ $\sqrt{\#j}$
G65	H22	Absolute	#i ＝ $\lvert\#j\rvert$
G65	H23	Remainders	#i＝#j-trunc(#j ÷ #k)×#k
G65	H26	Complex multiplication & division operations	#i ＝ (#i × #j) ÷ #k
G65	H27	Complex square root	#i ＝ $\sqrt{\#j^2+\#k^2}$
G65	H31	Sine	#i ＝ #j×SIN(#k)
G65	H32	Cosine	#i ＝ #j×COS(#k)
G65	H33	Tangent	#i ＝ #j×TAN(#k)
G65	H34	Arc tangent	#i ＝ ATAN(#j/#k)
G65	H80	Unconditional branch	Turn to N
G65	H81	Conditional branch 1	IF #j ＝ #k， GOTO N
G65	H82	Conditional branch 2	IF #j ≠ #k， GOTO N
G65	H83	Conditional branch 3	IF #j ＞ #k， GOTO N
G65	H84	Conditional branch 4	IF #j ＜ #k， GOTO N
G65	H85	Conditional branch 5	IF #j ≥ #k， GOTO N
G65	H86	Conditional branch 6	IF #j ≤ #k， GOTO N
G65	H99	Alarm	

Fig. 4-9-4-1

2. Operation code:

1) The assignment of variable: # I = # J

G65 H01 P#I Q#J；

(For example) G65 H01 P#101 Q1005； （#101 = 1005）

G65 H01 P#101 Q#110； （#101 = #110）

G65 H01 P#101 Q-#102； （#101 = -#102）

2) Plus operation: # I = # J+# K

G65 H02 P#I Q#J R#K;

(For example) G65 H02 P#101 Q#102 R15; （#101 = #102+15）

3) Minus operation: # I = # J- #K

G65 H03 P#I Q#J R# K;

(For example) G65 H03 P#101 Q#102 R#103; （#101 = #102-#103）

4) Multiplication operation: # I = # J×# K

G65 H04 P#I Q#J R#K;

(For example) G65 H04 P#101 Q#102 R#103; （#101 = #102×#103）

5) Division operation: # I = # J÷# K

G65 H05 P#I Q#J R#K;

(For example) G65 H05 P#101 Q#102 R#103; （#101 = #102÷#103）

6) Logic plus (OR): # I = # J.OR. # K

G65 H11 P#I Q#J R#K;

(For example) G65 H11 P#101 Q#102 R#103; （#101 = #102.OR. #103）

7) Logic multiplication (AND): # I = # J.AND. # K

G65 H12 P#I Q#J R#K;

(For example) G65 H12 P# 101 Q#102 R#103; （#101 = #102.AND.#103）

8) XOR: # I = # J.XOR. # K

G65 H13 P#I Q#J R#K;

(For example) G65 H13 P#101 Q#102 R#103; （#101 = #102.XOR. #103）

9) Square root: # I = $\sqrt{\#j}$

G65 H21 P#I Q#J;

(For example) G65 H21 P#101 Q#102 ; （#101= $\sqrt{\#102}$ ）

10) Absolute: # I = | # J |

G65 H22 P#I Q#J ;

(For example) G65 H22 P#101 Q#102 ; （#101 = | #102 |）

11) Remainder: # I = # J—TRUNC(#J/#K)×# K, TRUNC: Reject the decimal section

G65 H23 P#I Q#J R#K;

(For example) G65 H23 P#101 Q#102 R#103; （#101 = #102- TRUNC (#102/#103)×#103）

12) Complex devision operation: # I = （# I×# J）÷# K

G65 H26 P#I Q#J R# k;

(For example) G65 H26 P#101 Q#102 R#103; （#101 = （#101×# 102）÷#103）

VOLUME I PROGRAMMING

13) Complex square root I = $\sqrt{\#j^2+\#k^2}$

G65 H27 P#I Q#J R#K;

(For example) G65 H27 P#101 Q#102 R#103; (#101 = $\sqrt{\#102^2+\#103^2}$)

14) Sine: # I = # J•SIN (# K) (Unit: °)

G65 H31 P#I Q#J R#K;

(For example) G65 H31 P#101 Q#102 R#103; （#101 = #102•SIN（#103））

15) Cosine: # I = # J•COS (# K) (Unit: °)

G65 H32 P#I Q#J R# K;

(For example) G65 H32 P#101 Q#102 R#103; （#101 =#102•COS（#103））

16) Tangent: # I = # J•TAN (# K) (Unit: °)

G65 H33 P#I Q#J R# K;

(For example) G65 H33 P#101 Q#102 R#103; （#101 = #102•TAN（#103））

17) Arc tangent: # I = ATAN (# K) (Unit: °)

G65 H34 P#I Q#J R# K;

(For example) G65 H34 P#101 Q#102 R#103; （#101 =ATAN（#102/#103））

Note 1: The unit of angle variable is degree.

Note 2: In each operation, when the necessary Q and R are not specified, its value regards as zero to operate.

Note 3: trunc: Rounding operation, eject the decimal section.

3. Branch command

1) Unconditional branch

G65 H80 Pn; n: Sequence number

(For example) G65 H80 P120; (Trun to N120 block)

2) Conditional branch 1 #J.EQ.# K (＝)

G65 H81 Pn Q#J R# K; n: Sequence number

(For example) G65 H81 P1000 Q#101 R#102;

When # 101 = #102, turn to N1000 block, when #101 ≠ #102, program performes with sequentially.

3) Conditional branch 2 #J.NE.# K (≠)

G65 H82 Pn Q#J R# K; n: Sequence number

(For example) G65 H82 P1000 Q#101 R#102;

When # 101 ≠ #102, turn to N1000 block, when #101 = #102, program performs with sequentially.

4) Conditional branch 3 #J.GT.# K (＞)

VOLUME I PROGRAMMING

G65 H83 Pn Q#J R# K； n：Sequence number

(For example) G65 H83 P1000 Q#101 R#102；

When #101＞#102, turn to N1000 block, when #101 ≤ #102, program performs with sequentially.

5) Conditional branch 4 #J.LT.# K (＜)

G65 H84 Pn Q#J R# K； n：Sequence number

(For example) G65 H84 P1000 Q#101 R#102；

When # 101＜#102, turn to N1000 block, when #101 ≥ #102, program performs with sequentially.

6) Conditional branch 5 #J.GE.# K (≥)

G65 H85 Pn Q#J R# K； n：Sequence number

(For example) G65 H85 P1000 Q#101 R#102；

When # 101 ≥ #102, turn to N1000 block, when #101＜#102, program performs with sequentially.

7) Conditional branch 6 #J.LE. # K (≤)

G65 H86 Pn Q#J R# K； n：Sequence number

(For example) G65 H86 P1000 Q#101 R#102；

When # 101≤ #102, turn to N1000 block, when #101＞#102, program performs with sequentially.

Note: The sequence number can be specified by variable. For example: G65 H81 P#100 Q#101 R#102; the program moves to the block specified by #100 sequence number when the condition is executed.

4. Logic AND, OR and NON codes

For example:

G65 H01 P#101 Q3；

G65 H01 P#102 Q5；

G65 H11 P#100 Q#101 Q#102；

5 means that the binary is 101, 3 means 011, the calculation result is #100=7;

G65 H12 P#100 Q#101 Q#102；

5 means that the binary is 101, 3 means 011, the calculation result is #100=1;

5. Macro variable alarm

For example:

G65 H99 P1； Macro variable 3001 alarms

G65 H99 P124； Macro variable 3124alarms

The example of user macro program

1. Bolt hole cycle

The center is regarded as reference position (X0, Y0), radius is (R), and start angle is (A); N holes can be machined accordingly.

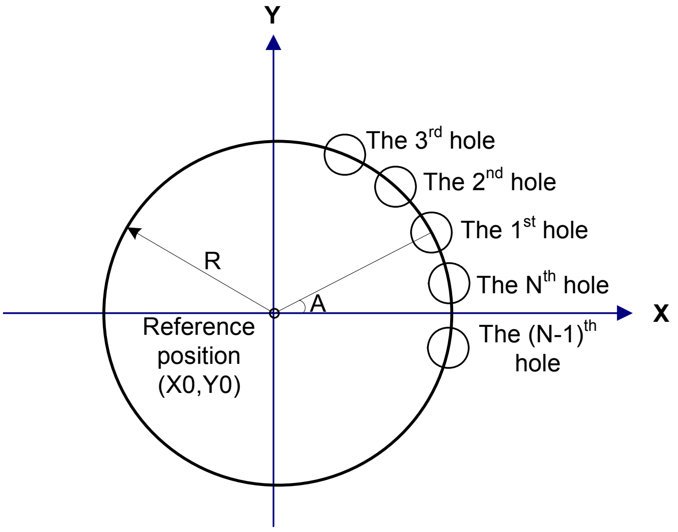


Fig. 4-9-4-2

X0, Y0 are the reference point coordinate value of the bolt hole cycle.

R: Radiu, A: Start angle, N: Numbers. The above-mentioned parameters are used the following variables.

#500: The coordinate value (X0) of reference position X

#501: The coordinate value (Y0) of reference position Y

#502: Radius (R)

#503: Start angle (A)

#504: N numbers

N＞0, CCW, numbers N.

N＜0, CW, numbers N.

The following variables are used the operation in the macro program.

#100: The 1st hole machining counting (I)

#101: Terminal value of counting (＝∣N∣) (IE)

#102: The angle (θI) of the 1st hole

#103: The X coordinate value (Xi) of the 1st hole

#104: The Ycoordinate value (Yi) of the 1st hole

The user macro program body can be written as the following forms:

O9010；

N100 G65 H01 P#100 Q0；　　　　I＝0

G65 H22 P#101 Q#504；　　　　IE＝|N|

N200 G65 H04 P#102 Q#100 R360；

VOLUME I PROGRAMMING

G65 H05 P#102 Q#102 R#504；　　θI＝A＋360°×I/N

G65 H02 P#102 Q#503 R#102；

G65 H32 P#103 Q#502 R#102；　　X I＝X I＋R·COS(θI)

G65 H02 P#103 Q#500 R#103；

G65 H31 P#104 Q#502 R#102；　　Y I＝Y I＋R·SIN(θI)

G65 H02 P#104 Q#501 R#104；

G90 G00 X#103 Y#104；　　The 1st hole positioning

G**；　　The concrete hole machining G code

G65 H02 P#100 Q#100 R1；　　I＝I＋1

G65 H84 P200 Q#100 R#101；　　When I＜IE, turn to N200 to machin the hole of IE

M99；

Call the program examples of the above-mentioned user macro program body:

O0010；

G65 H01 P#500 Q100；　　X_0=100MM

G65 H01 P#501 Q-200；　　Y_0=-200MM

G65 H01 P#502 Q100；　　R=100MM

G65 H01 P#503 Q20；　　A=20°

G65 H01 P#504 Q12；　　N=12 CCW

G92 X0 Y0 Z0；

M98 P9010；　　User macro program calling

G80；

X0 Y0；

M30；

4.9.5　User Macro Program Function B

1. Arithmetic and Logic operation

The operations listed in the following table can be performed on variables. The expression at the right of the operator can contain constants and/or variables combined by a function or operator. Variable #j and #k in an expression can be replaced with a constant. Variable at the left can also be assigned with an expression.

Table 4-9-5-1 Arithmetic and logic operation

Function	Format	Remark
Definition	#i = #j	
Sum	#i = #j + #k;	
Difference	#i = #j - #k;	
Product	#i = #j * #k;	
Quotient	#i = #j / #k;	
Sine	#i = SIN[#j];	An angle is specified in degree. 90°30’ means 90.5 degrees.
Arcsine	#i = ASIN[#j];	
Cosine	#i = COS[#j];	
Arccosine	#i = ACOS[#j];	
Tangent	#i = TAN[#j];	
Arctangent	#i = ATAN[#j] / [#k];	
Square root	#i = SQRT[#j];	
Absolute value	#i = ABS[#j];	
Rounding off	#i = ROUND[#j];	
Rounding up	#i = FUP[#j];	
Rounding down	#i = FIX [#j];	
Nature logarithm	#i = LN[#j];	
Exponential function	#i = EXP[#j];	
OR	#i = #j OR #k;	A logic operation is performed based on binary numbers bit by bit.
XOR	#i = #j XOR #k;	
AND	#i = #j AND #k;	
Conversion from BCD to BIN	#i = BIN[#j];	It used to the signal exchange for PMC.
Conversion from BIN to BCD	#i = BCD[#j];	

Explanations:

(1) Angle unit

The units of angles used with the SIN, COS, ASIN, ACOS, TAN and ATAN functions are degrees. For example, 90°30’ is represented as 90.5 degrees.

（2）**ARCSIN #i = ASIN [#j]**

(2) ARCSIN #i = ASIN [#j]

Solution ranges from -90°～90°.

When #j is beyond the range of -1 to 1, system alarm is issued.

A constant can be used instead of the #j variable.

（3）ARCCOS #i = ACOS [#j]

(3) ARCCOS #i = ACOS [#j]

The solution ranges from 180° to 0°.

When #j execeeds the range of -1 to 1, system alarm is generated.

A constant can be used instead of the #j variable.

（4）ARCTAN #i = ATAN [#j] / [#k]

(4) ARCTAN #i = ATAN [#j] / [#k]

Specify the length of two sides, separated by a slahs (/).

The solution range from 0° to 360°.

[For example] When #1 = ATAN [-1] / [-1]; is specified, #1=225°.

A constant can be replaced by # j variable.

(5) Nature logarithm #i = LN [#j]

When the antilogarithm (# j) is zero or smaller, the system alarm occurs.

A constant can be replaced by # j variable.

(6) Exponential function #i = EXP [#j]

When the result of operation exceeds 99997.453535 (#j is about 11.5129), an overflow occurs and the system alarm issues.

A constant can be replaced by # j variable.

(7) ROUND function

ROUND function rounds off at the 1st decimale position.

For example:

When #1=ROUND[#2]; is executed where #2=1.2345, the value of variable 1 is 1.0.

(8) Round down and round up

With CNC, when the absolute value of the integer produced by an operation on a number is greater than the absolute value of the original number, such an operation is referred to as rounding up to an integer. Conversely, when the absolute value of the integer produced by an operation on a number is less than the absolute value of the original number, such an operation is referred to as rounding down to an integer. Be particularly careful when handing negative numbers.

Examples:

Suppose that #1=1.2, #2=－1.2。

When #3=FUP[#1] is executed, 2.0 is assigned to #3.

When #3=FIX[#1] is executed, 1.0 is assigned to #3。

When #3=FUP[#2] is executed, －2.0 is assigned to #3。

When #3=FIX[#2] is executed, －1.0 is assigned to #3。

(9) Abbreviations of arithmetic and logic operation commands

When a function is specified in a program, the first two characters of the function name can be used to specify the function. (See the table 4-9-5-1).

Example:

ROUND→RO

FIX→FI

(10) Priority of operations

① Function

② Operations such as multiplication and devision (* / AND)

③ Operations such as addition and subtraction (+ - OR XOR)

Example) $\#1 = \underbrace{\#2 + \underbrace{\#3 * \underbrace{SIN[\#4]}_{\boxed{1}}}_{\boxed{2}}}_{\boxed{3}}$;

$\boxed{1}$, $\boxed{2}$ and $\boxed{3}$ indicate the order of operations.

(11) Restriction

[,] uses the enclosed expressions.

When specifying the divisor 0 or TAN[90] in division operation, system alarm is then issued.

2. Branch and Repetition

1) Branch and Repetition

In a program, the flow of control can be changed using the GOTO statement and IF statement. Three types of branch and repetition operations are used:

Branch and repetition
- GOTO statement (Unconditional branch)
- IF statement (Conditional branch: IF…THEN…)
- WHILE statement (Repetition while…)

2) Unconditional branch

➢ GOTO statement

Branch to the block marked with sequence number n. A sequence number can also be specified using an expression.

GOTOn; n: Sequence number (1 to 99999）

Example:

GOTO 1;

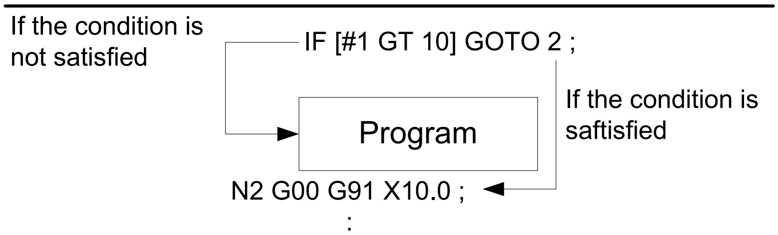
GOTO #10;

3) Conditional branch (IF statement) [<Conditional expression>]

IF[<Conditional expression >]GOTO n

If the specified conditional expression is satisfied, Branch to the block marked with sequence number n. If the specified conditional expression is not satisfied, the next block is executed.

If the variable value #1 is greater than 10, branch to the block with sequence number N2.



IF[<Conditional expression>]THEN

If the conditional expression is satisfied, a predetermined macro program statement is executed. Only a single macro statement is carried out.

If the value of #1 and #2 are the same, 0 is assigned to #3.
IF[#1 EQ #2] THEN #3=0;

Explanations:

- Conditional expression

  A conditional expression must include an operator inserted between two variables or between a variable and constant, and must be enclosed in brackets ([,]). An expression can be replaced a variable.

- Operators

  Operators consist of two letters and used to compare two values to determine whether they are equal or one value is smaller or greater than the other one.

Table 4-9-5-2 Operators

Operator	Meaning
EQ	Equal to （=）
NE	Note equal to （≠）
GT	Greater than （>）

GE	Greater than or equal to（≥）
LT	Less than（<）
LE	Less than or equal to（≤）

➢ Typical program

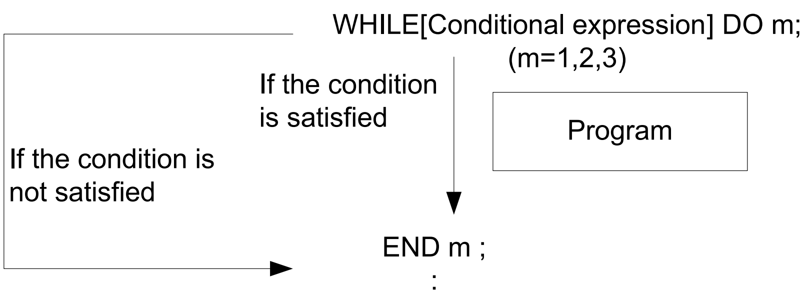
The sample program below finds the total of numbers 1 to 10.

```
O9500；
#1=0；                   Initial value of the variable to hold the sum
#2=1；                   Initial value of the variable as an addend
N1 IF[#2 GT 10]GOTO 2;   Branch to N2 when the addend is greater than 10
#1=#1+#2；               Calculation to find the sum
#2=#2+1；                Next addend
GOTO 1；                 Branch to N1
N2 M30；                 End of program
```

VOLUME I PROGRAMMING

4) Repetition (WHILE statement)

Specify a conditional expression after WHILE. While the specified consition is satisfied, the program from DO to END is executed; otherwise, turn to the block after END.



When the specified condition is satisfied, the program from DO to END after WHILE is executed; otherwise, it may perform the block after END. This kind of command format is suitable for the IF statement. The numbers followed with the DO and END are specified the labels of the program execution range. The numbers 1, 2 and 3 can be used. When a number other than 1, 2, and 3 is used, the alarm may occur.

Explanation:

➢ Nesting

The identification numbers (1 to 3) in a DO——END cycle can be used as many times as desired. Note, when a program includes the overlapping of the crossing repetition cycle DO ranges, the alarm may issue.

1. Numbers (1 to 3) can be used as many times as required.

```
WHILE[ … ] DO 1;
    Program
END 1 ;
  :
WHILE[ … ] DO 1;
    Program
END 1 ;
  :
```

2. The range of DO can not be crossed

```
WHILE[ … ] DO 1;
    Program
WHILE[ … ] DO 2;
  :
END 1 ;
    Program
END 2 ;
```

3. DO cycle can be nested to a Max. depth of 3 levels.

```
WHILE[ … ] DO 1;
  :
  WHILE[ … ] DO 2;
    :
    WHILE[ … ] DO 3;
        Program
    END 3 ;
  END 2 ;
END 1 ;
```

4. Control can be transferred to the outside of a repetition.

```
WHILE[ … ] DO 1;
IF[…] GOTO n ;
END 1 ;
Nn
```

Explanations:

➢ Infinite cycle

When DO is specified instead of specifying the WHILE statement, an infinite cycle ranging from DO to END is produced.

➢ Processing time

When the GOTO statement marked with a branch is treated, and the sequence number index is performed. For this reason, the processing in the reverse direction takes a longer time than processing in the forward direction. Using the WHILE statement for repetition reduces processing time.

➢ Undefined variable

In the condtional expression that uses EQ or NE, a <vacant> and zero have different effects. In other types of conditional expressions, a <vacant> is regarded as zero.

➢ Typical program

The sample program below counts the total of numbers 1 to 10.

```
O0001 ;
#1=0;
#2=1;
WHILE [#2 LE 10] DO 1;
#1=#1+#2;
#2=#2+1;
END 1;
M30;
```


Precautions:

- When a macro program is called by G65 and a variable is quoted by F, the system performs based upon the variable value.
- Do not use the cycle and skip command in DNC.
- GOTO statement searches downward from the current block; if it can not find the corresponding sequence number and then research again back to the beginning of the program. Do not use the identical N codes in a same program as much as possible.
- When the decimal means the variable number, the system will directly discard the decimal section instead of considering the carry-bit
- The value of the local variable will always hold before the end of main program, which is used together within each subprogram.

CHAPTER FIVE MISCELLANEOUS FUNCTION M CODE

The M codes for this machine tool can be used by user, which shows below:

Table 5-1

	M code	Function
M code for controlling the program	M30	Program ends and turns to the beginning of the program, maching number adds 1.
	M02	Program ends and turns to the beginning of the program, maching number adds 1.
	M98	Subprogram calling
	M99	Sbuprogram ends and returns/repeatedly perform
	M00	Program dwell
	M01	Program dwell selection
PLC controls the M code	M03	Spindle positive
	M04	Spindle reverse
	M05	Spindle stop
	M06	Tool-change
	M08	Cooling ON
	M09	Cooling OFF
	M10	A axis releasing
	M11	A axis clamping
	M16	Tool control releasing
	M17	Tool control clamping
	M18	Spindle orientation cancellation
	M19	Spindle orientation
	M20	Spindle neutral command
	M21	Cutter-search code in tool return
	M22	Cutter-search code in capturing a new tool
	M26	Punching water-valve ON
	M27	Punching water-valve OFF
	M28	Cancel the rigid tapping
	M29	Rigid tapping command
	M35	Start the chip-removal promotion conveyor
	M36	Close the chip-removal promotion conveyor
	M44	Start the spindle blowing

	M45	Close the spindle blowing
	M50	Start the automation tool-change
	M51	Close the automation tool-change

The movement code and the miscellaneous function are simultaneously performed when they are shared with a same block.

When the numerical value followed with the address M is specified, the code signal and strobe signal are conveyed to the machine tool to switch on or off this functions using machine. Usaully, only one M code can be specified in one block. Up to 3 M codes in one block can be specified by setting the bit 7 of parameter No.: 33. However, some M codes can not be specified at the same time due to mechanical operation restrictions. For detailed information about the mechanical operation restrictions on simultaneous specification of multiple M codes in one block, refer to the manual of each machine tool builder.

5.1 M Code Control by PLC

A M code and a movement code are simultaneously performed when the M code controlled by PLC shares the same block with a movement code.

5.1.1 Negative/Reverse Code Command (M03, M04)

Code: M03 (M04) Sx x x;

Explanation: The positive rotation is observed from negative to position along with the Z axis, spindle CCW is regarded as positive; and CW is treated as reverse. Entering the rotation direction of workpiece based upon the right helix is regarded as positive, and the leaving the rotation direction of workpiece based upon the right helix is treated as reverse.

Sx x x code, is the spindle speed, is the located gear when the gear is controlled.

Unit: rev./min/ (r/min)

Sx x x is the actual speed when the frequency conversion is controlled. For example: S1000 specifies that the spindle is revolved based upon the 1000r/min.

5.1.2 Spindle Stopping Code Command M05

Code: M05, spindle will stop rotating when performing M05 in Auto method. However, the commanded speed by S code is reserved. The deceleration method in spindle stop is determined by the machine tool manufacturer. Generally, it is the energy-consumption brake.

5.1.3 Cooling ON/OFF (M08, M09)

Code: M08, control the ON of the cooling water pump. M09, control the OFF of the cooling water pump. In the Auto mode, the water pump control code does not execute if the miscellaneous function locking is performed.

5.1.4 A Axis Releasing/Clamping (M10, M11)

Code: M10, A axis releasing. M11, A axis clamping

5.1.5 Tool Control Tool-releasing/Tool Clamping (M16, M17)

Code: M16, tool-releasing control. M17, tool-clamping control

5.1.6 Spindle Orientation/Cancellation (M18, M19)

Code: M18, spindle orientation cancels. M19, spindle orients for using the tool-change positioning.

5.1.7 Tool-searching Code Command (M21, M22)

Code: M21, the tool-searching code in cutter-return; M22, the tool-searching code in capturing a new tool.

5.1.8 Tool-magazine Return Code Command (M23, M24)

Code: M23, tool-magazine swings to the code of spindle position; M24, tool-magazine swings to code of original position.

5.1.9 Rigid Tapping (M28, M29)

Code: M28, cancel the rigid tapping. M29, rigid tapping.

5.1.10 Helical Chip-removal Conveyor ON/OFF (M35, M36)

Code: M35, control the start of the helical chip-removal conveyor. M36, control the stop of the helical chip-removal conveyor.

5.1.11 Punching Water Valve ON/OFF (M26, M27)

Code: M26 is puncing water valve ON; M27 is puncing water valve OFF.

5.1.12 Spindle Blowing ON/OFF (M44, M45)

Code: M44, Spindle blowing contron ON. M45, Spindle blowing contron OFF.

5.1.13 Automatic Tool-change Start/End (M50, M51)

Code: M50, Start the automatic tool-change control. M51, End the automatic tool-change control.

5.2 M Codes for Controlling Program

The M codes for controlling the program are divided into main program control and macro program control. Firstly perform the movement code, and then perform the M code when the M codes for controlling program are shared the same block with the movement code.

Notice: 1. M00, M01, M02, M06, M30, M98 and M99 codes can not be specified together with other M codes; otherwise, the system alarm occurs. When the above-mentioned M codes are shared the same block with other codes (except the M codes), the other codes in the same block will firstly perform and then perform these M codes.

2. These M codes include that the CNC conveys the M codes to the machine tool, simultaneously, the CNC can also performs the internal operation code; for example, the M code for the program prereading function is disabled. Additional, the CNC conveys only the M code to machine tool instead of executing the M code of the internal operation, which can be specified within the same block.

5.2.1 Program End and Return (M30, M02)

In the Auto operation, program stops the automatic state when operating to M30 (M20), the followed program will not perform and the spindle and cooling operation will stopped according; the workpiece machining number adds 1. M30 can be controlled whether it returns to the beginning of program by bit 4 of No.: 33; M02 can be controlled whether it returns to the beginning of program by bit 2 of No.: 33. If the M02 and M30 are performed at the end of the subprogram, return to program of the subprogram calling, and then continue perform the following blocks.

5.2.2 Program Dwell (M00)

In the Auto mode, the automatic state dwells when program operates to the M00. In this case, the previous modal information will registered and it continues by the repeated start button. The function equals to the feed hold button by pressing.

5.2.3 Program Optional Dwell (M01)

In the Auto mode, the optional dwell automatic operating state when program moves to the M01; if the "Optional stop" switch is turned on, the M01 shares same effect with M00; if the "Optional stop" switch is turned off, regardless of the M01 code. Refer to the Operation Manual for details.

5.2.4 Program Calls Subprogram Code Command (M98)

M98 code can be compiled in main program to call the performance of subprogram. Refer to the following format:

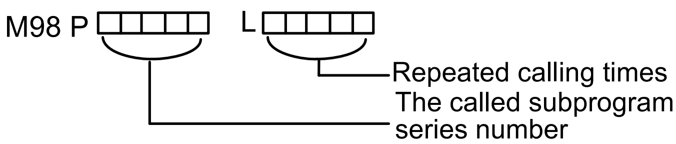


Fig. 5-2-4-1

5.2.5 Program Ends and Rturns (M99)

1. In the Auto mode, if a M99 uses at the end of the main program, which returns to the beginning of the program to automatically perform when the program operates to M99; the following programs will not execute, and the workpiece machining number does not accumulate.
2. Using M99 at the end of subprogram, the program will return to main program after operting this block, and then continue executing by calling the next program of the subprogram.
3. M30 is treated by M99 in the DNC mode, and the cursor stops at the end of the program.

CHAPTER SIX SPINDLE FUNCTION S CODE

The code signal conveys to machine tool after converting into analog signal by the S codes and its following numerical values, which is used for the spindle control of the machine tool. S is regarded as modal value.

6.1 Spindle Analog Control

When the **SPT** of the **bit 2 of parameter No.: 1** sets to **0,** the address S and its following numerical value can be controlled the spindle speed byanalog voltage; refer to the operation manual for details.

Code format: S_

Explanations:

1. One S code can be specified in a block.
2. The spindle can be directly specified by the address S and its following data, its unit is rev./min (r/min). For example: M3 S300 means that the spindle operates based upon the 300 rev./min.
3. When the movement code and S code are shared with a same block, which are performed simultaneously.
4. The spindle can be controlled by the S code and its following numerical values.

6.2 Spindle Switch Value Control

When the SPT of the **bit 2 of parameter No.: 1 sets to 1**, the spindle speed can be controlled by the address S and its following 2-digit switch value.

When the spindle speed is controlled by selecting a switch value, the system then can offers 3-level spindle mechanical gear-shifting. Refer to the manual made by the manufacturer for the corresponding relationships and the spindle shifing levels provide by machine tool of S code and spindle speed.

Code format: S01 (S1) ;

S02 (S2) ;

S03 (S3) ;

Explanations:

1. At present, there are 8 shiftings for the software, but the ladder diagram is only 3. When the above-mentioned codes other than the S codes are specified, the system then displays the **"The miscellaneous function is being performed…".**

2. If the 4-digit followed with S are specified, the later 2-digit are then enabled.

6.3 Constant Surface Cutting Speed Control (G96/G97)

Code format:

Constant surface speed control code G96 S_ Surface speed (mm/min or inch/min)

Constant surface speed control code cancellation G97 S_ Spindle speed (r/min)

Constant surface speed controlled axis code G96 P_ P1 X axis; P2 Y axis; P3 Z axis; P4 the 4th axis

Clamp of Max. spindle speed G92 S_ S specifies the Max. spindle speed (r/min)

Function: Specify the surface speed (relative speed between the tool and workpiece) following S. The spindle is rotated so that the surface speed is constant regardless of the tool position.

Explanations:

1. G96 is a modal code. After a G96 command is specified, the program enters the constant surface speed control mode, and the S value is regarded as surface speed.

2. A G96 code must specify the axis along with constant surface speed control is applied. A G97 code cancels the G96 mode.

3. It is necessary to set a workpiece coordinate system for performing the constant surface cutting feed control, so that the center coordinate of rotation axis changes into zero.

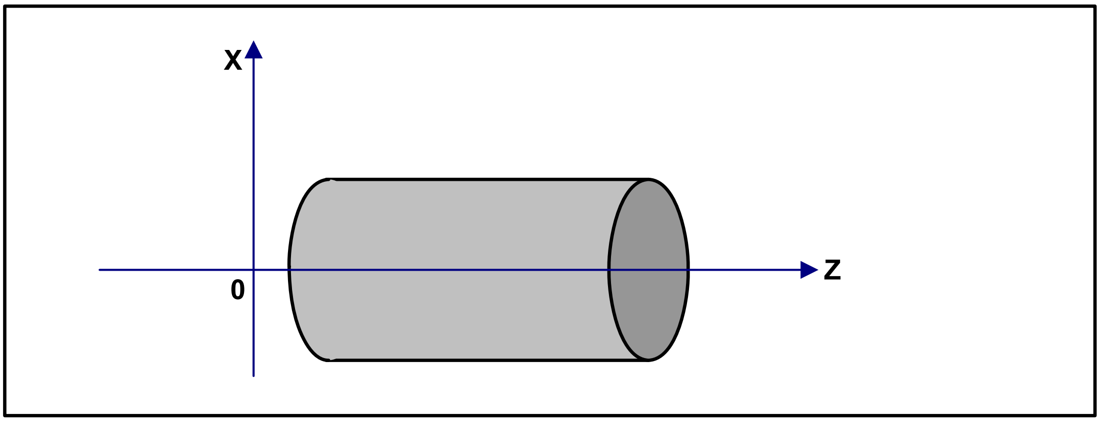


Fig. 6-3-1 The controlled workpiece coordinate system of constant surface cutting speed

4. When constant surface speed control is applied, a spindle speed higher than the value specified in G92 S_, it is convenient to clamp at the Max. spindle speed. When the power is turned on, the Max. spindle speed is not yet set. The S is regarded as S=0 in a G96 code till M3 or M4 appears in program.

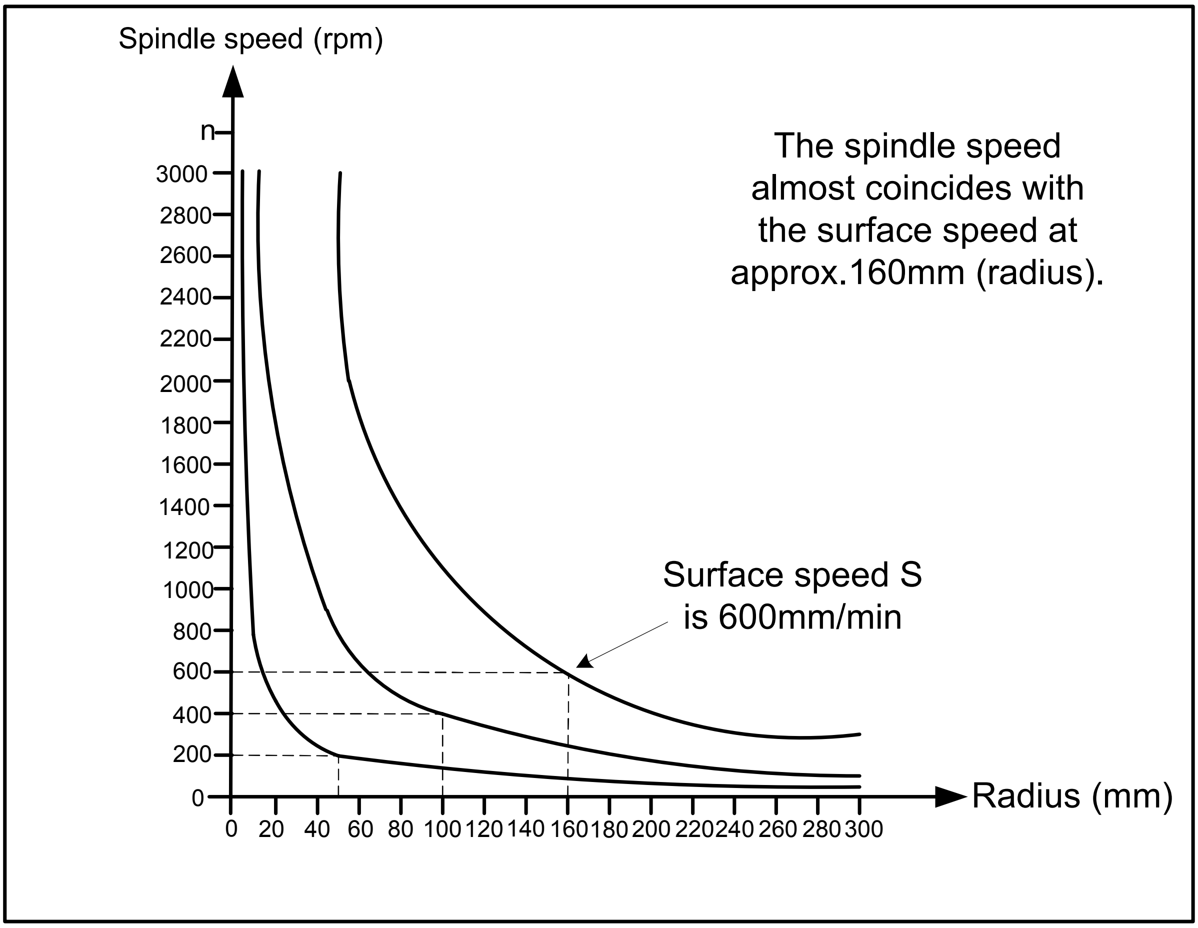


Fig. 6-3-2 Relationships among the workpiece radius, spindle speed and surface speed

5. Specify surface cutting feed in G96 mode:

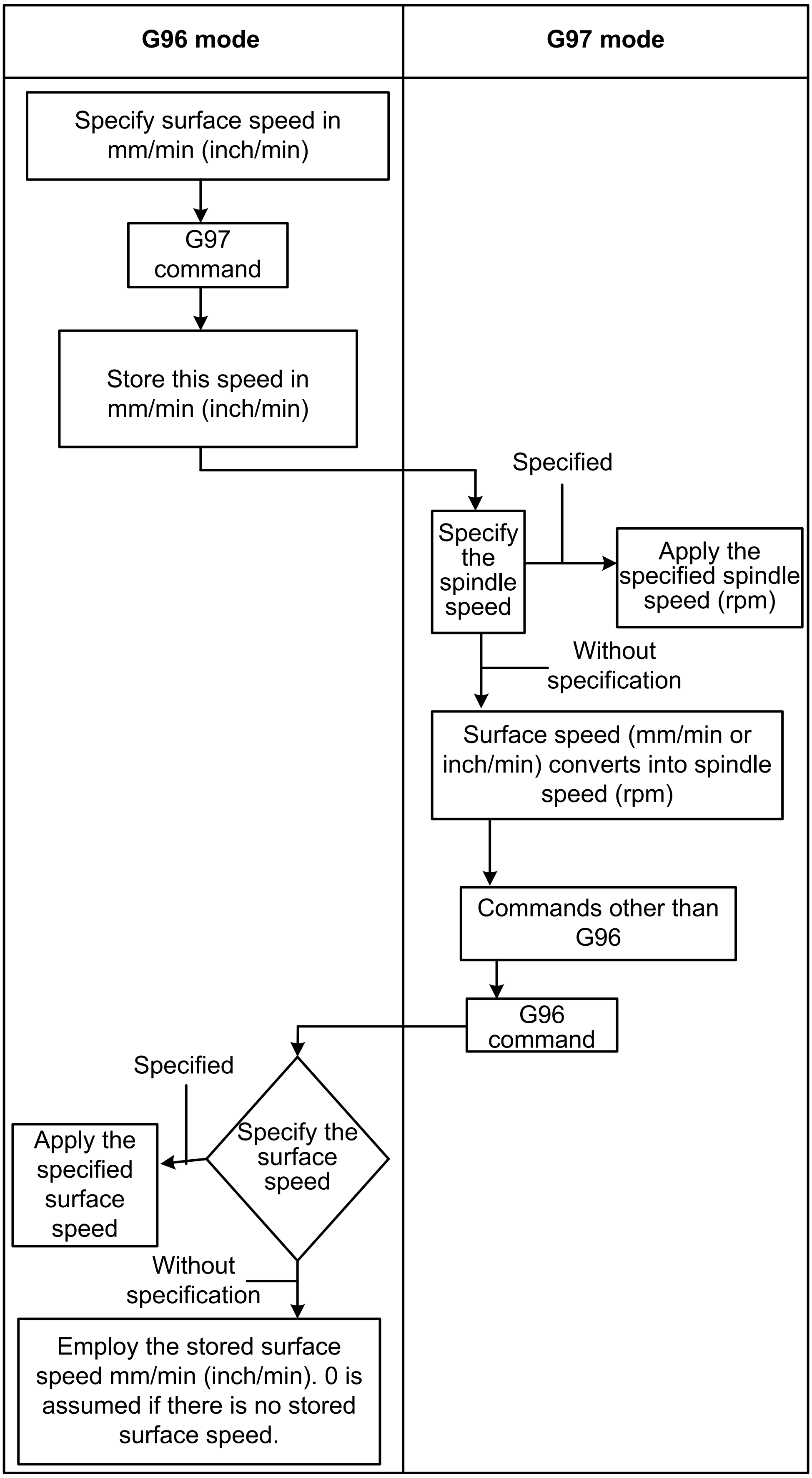

G96 mode
G97 mode
Specify surface speed in mm/min (inch/min)
G97 command
Store this speed in mm/min (inch/min)
Specified
Specify the spindle speed
Apply the specified spindle speed (rpm)
Without specification
Surface speed (mm/min or inch/min) converts into spindle speed (rpm)
Commands other than G96
G96 command
Specified
Specify the surface speed
Apply the specified surface speed
Without specification
Employ the stored surface speed mm/min (inch/min). 0 is assumed if there is no stored surface speed.

Fig. 6-3-3

6. The relative parameter setting of G96: When the bit 2 of parameter No.:37 sets the G0 at the rapid traverse rate, calculate the reference coordinate of G96 spindle speed (0: End, 1: Current point). When the bit 3 of parameter No.: 37 sets the G96 spindle speed clamping (0: Before the spindle override, 1: After the spindle override); whether the setting of bit 0 of parameter No.:61 is used the constant cycle speed control.

Restrictions:

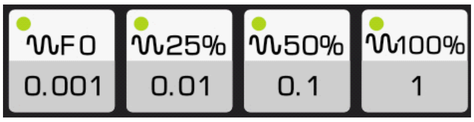
1. The response problem in the servo system does not consider when the spindle speed changes, but the constant surface cutting speed control is also enabled during thread cutting. And therefore, cancel the constant surface cutting speed by G97 before the machining of thread.
2. In a rpaid traverse block specified by G00, the constant surface speed control is not made by calculating the surface speed to a transient change of the tool position, but is made by calculating the surface speed based upon the position at the end point of the rapd traverse block, on the condition that cutting is not executed at rapid traverse.
3. When the machining, such as the flexible tapping, rigid tapping or peck tapping, etc. are performed, it is necessary to firstly cancel the constant surface cutting feed by G97; otherwise, the disorder gear or broken screw tap, etc. will occur.

CHAPTER SEVEN FEED FUNCTION F CODES

Feed function controls the feedrate of a cutter, the feed function and its control method are shown below:

7.1 Rapid Traverse

The rapid positioning is performed by code (G00). The rapid feedrate are set by data parameter **P88~P92.** The following override adjustment can be performed by its corresponding buttons on the operation panel:



218MC, 218MC-H, 218MC-V buttons

218MC-U1 button

Fig. 7-1-1 The buttons of rapid feedrate

Wherein, F0: It is set by data parameter **P93**.

The acceleration of rapid positioning (G0) can be determined by data parameter **P105~P123,** which can be set reasonably based upon the machine tool and the motor's response characters.

Note: In the G00 block, the F code is disabled even if the it is specified; the system positions at the G0 speed.

7.2 Cutting Speed

In the linear interpolation (G01) and circular arc interpolation (G02,G03), the feedrate of cutter is specified by the numerical values followed with the F code, its unit is mm/min. Tool moves along the cutting feedrate compiled in program. The cutting feedrate can be carried out by using the feedrate buttons on operation panel (the override adjustment range is 0%～200%).

Automatically perform the acceleration/deceleration to prevent the mechanical vibration at the beginning and the end of the tool movement; acceleration can be set by data parameter **P125~P128.**

The Max. cutting speed is set by data parameter **P96**; The Min. cutting feed is determined by data parameter **P97**; if the cutting feed is higher the Max restriction value, it will be restricted at the upper-limit value; if the cutting speed is lower the Min. restriction value, it will be restricted at the lower-limit value.

The cutting feedrate in the Auto mode, when the power is turned on, is determined by data parameter **P87**.

The cutting speed can be specified by the following two methods:

A) Feed/min. (G94): Specify cutter feed value per minute after F.

B) Feed/rev. (G95): Specify the cutter feed value of the spindle per revolution after F.

Code format: G94 F_

Function: Feed amount per minute. Unit: mm/min or inch/min

Explanations:

1. After specifying G94 (in the feed per minute mode), the feed amount of tool per minute is directly specified by the numerical value followed with the F.
2. G94 is a modal code. Once a G94 is specified, it is enabled until G95 is specified. It is regarded as the feed per minute by default, and the default cutting feedrate is set by data parameter P87.
3. The feed/min. can be debugged by override adjustment button or wave-band switch buttons on the operation panel, the override is from 0% to 200%.

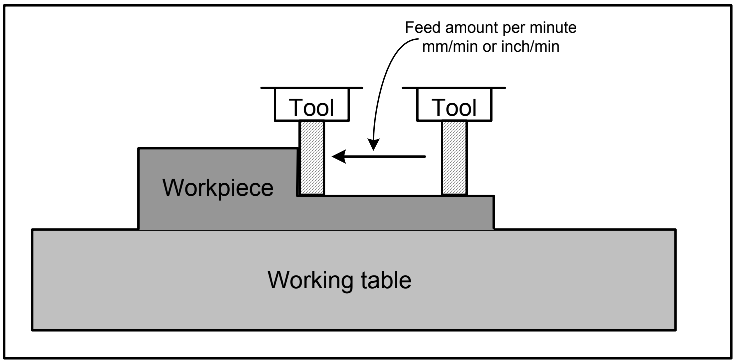


Fig. 7-2-1-1 Feed per minute

7.2.2 Feed per Revolution (G95)

Code format: G95 F_

Function: Tool feed amount per revolution. Unit: mm/r or inch/r.

Explanations:

1. The machine tool can be used this function unless installing the spindle encoder.

2. The feed amount of per revolution is directly specified by the numerical values followed with F after specifying G95 (feed/rev.) is specified.

3. G95 is modal code. Once a G95 code is specified, it is always enabled untill G94 is specified. The default feedrate per revolution is zero in the initialization.

4. The feed/min. can be debugged by override adjustment button or wave-band switch buttons on the operation panel, the override is from 0% to 240%.

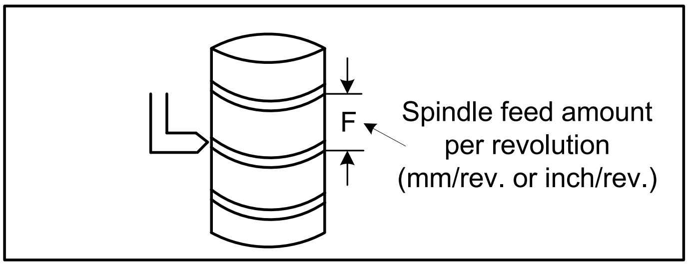


Fig. 7-2-2-1 Feed per revolution

Notice: When the speed of the spindle is low, feedrate fluctuation may occur. The slower the spindle rotates, the more frequently feedrate fluctuation occurs.

Note: In the G95 feed per revolution mode, the top speed per revolution treated by system is F500, if it exceeds F500, the alarm may issue.

7.3 Tangential Speed Control

Usually, the cutting feed is the speed for controlling contour path tangential direction, so that it can be reached to the speed value of a command.

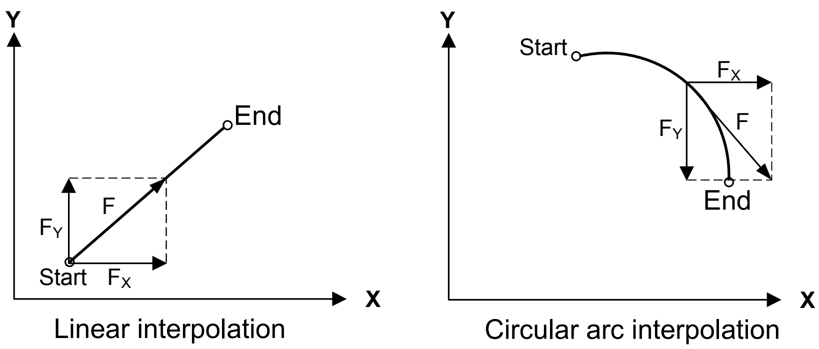


Fig. 7-3-1

F: The speed along the tangential direction $F=\sqrt{F_X^2+F_Y^2+F_Z^2}$

Fx: The speed along the X axis direction

Fy: The speed along the Y axis direction

Fz: The speed along the Z axis direction

7.4 Feedrate Override Button

The feedrate in the Manual and Auto modes can be adjusted by the override adjustment buttons on the operation panel, which can be used the override between 0～200% (10% of each gear, total 21 gears). In the Auto mode, when the override button adjusts to zero, the system will then stop the feed, and displays the cutting override 0, and then adjust the override button, the program can be continued.

7.5 Automatic Acceleration/Deceleration

The system is automatically performed the acceleration/deceleration control by moving the motor at the start and end, and therefore, it can be stably started and stopped. Also, it can be automatically accelerated or decelerated when the movement speed changes. Hernce, the acceleration or deceleration does not consider when programming.

Rapid feed: Forward acceleration/deceleration (0: Linear; 1: S type), backward acceleration/deceleration (0: Linear; 1: Exponential type)

Cutting feed: Forward acceleration/deceleration (0: Linear; 1: S type), backward acceleration/deceleration (0: Linear; 1: Exponential type)

Manual feed: Backward acceleration/deceleration (0: Linear; 1: Exponential type)

(The general-purpose time constant along each axis can be set by parameter)

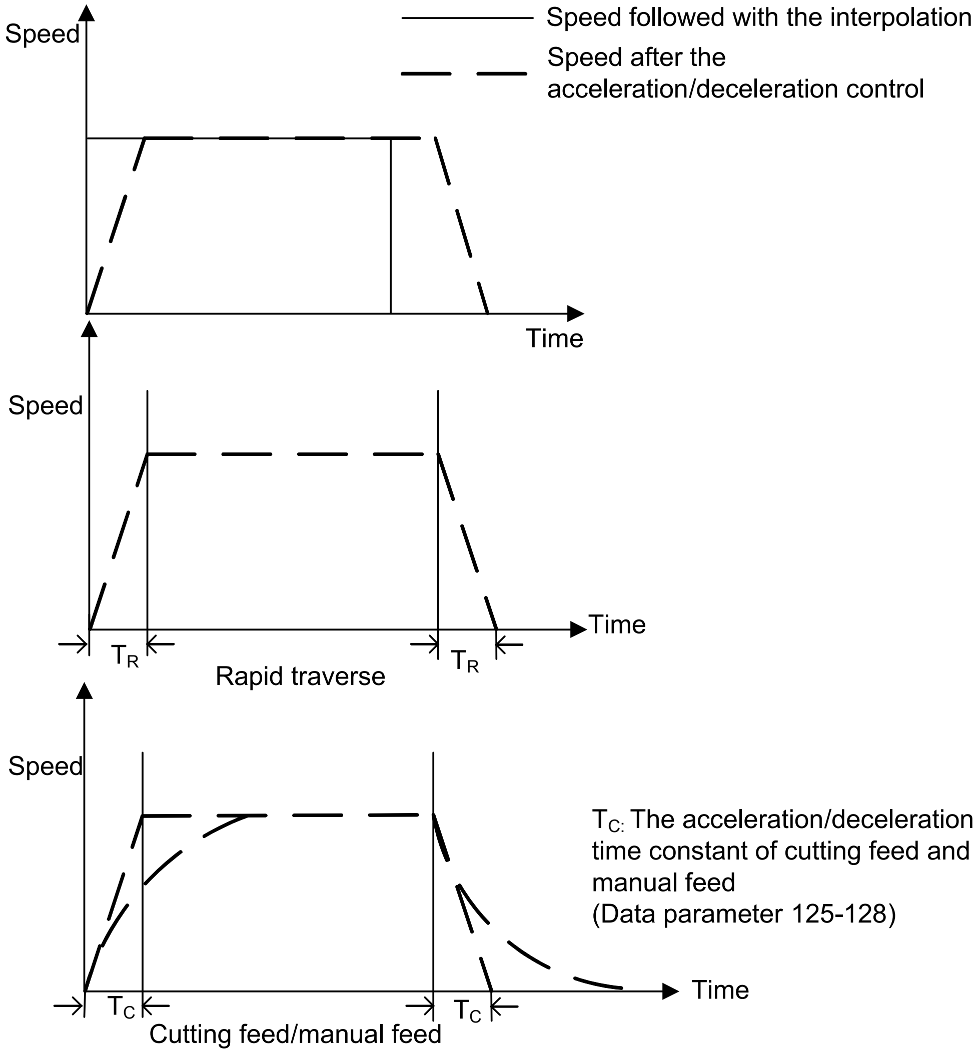


Fig. 7-5-1

7.6 Acceleration/Deceleration Treatment at Corner of Block

For example, if the previous block is only moved along Y, and the next block is only moved along X; the X line accelerates while the Y decelerates, in this case, the tool path is as follows:

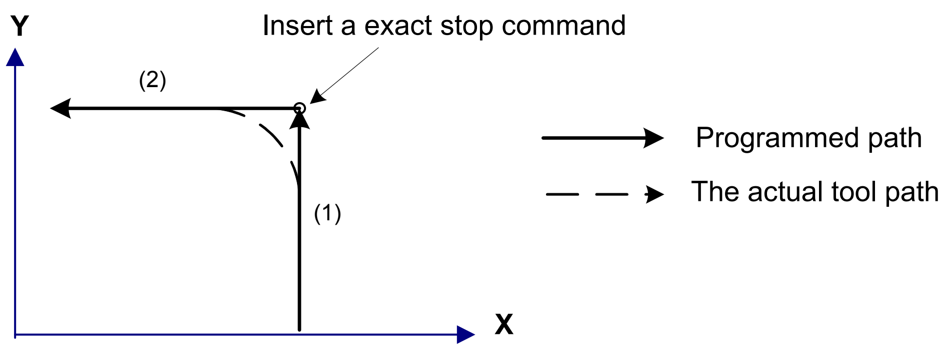


Fig. 7-6-1

If the exact stop code is input, the tool operates according to the program command as the solid line in the Fig.7-6-1. Otherwise, the bigger the cutting speed is, or the longer the acceleration/deceleration time constant is, the bigger the arc at the corner is. In the circular arc command, the arc radius of the actual tool path is smaller than the one provided by program. If you want to shorten the error at the corner, the acceleration/deceleration time constant should be set smaller as much as possible in the case of the available mechnical system.

CHAPTER EIGHT TOOL FUNCTION

8.1 Tool Function

Specify the numerical values (up to 8 digits) followed with the address T for using selecting the tool on the machine tool.

In principle, two or more T codes can not be specified at a same block. If a same codes at a same block is set without alarm, it is subject to the following T codes. The digit numbers to be specified of address T and the machine tool operation of corresponding by T code can be referred to the manual made by machine tool factory.

When the movement codes and the T codes are specified at a same block, which are performed at the same time.

When T codes and tool-change code M06 are shared with a same block, firstly perform the T code, and then the tool-change code. If the T codes and tool-change code M06 are not shared with a same block, the tool-change code detects whether the spindle tool number is consistent with the T code cutter, if does, the tool-change will not be performed.

The following programs are regarded as examples:

O00010；	
N10 T2M6；	The cutter on spindle is No.T2
N20 M6T3；	The cutter on spindle is No.T3
N30 T4；	The cutter on spindle is No.T3
N40 M6；	The cutter on spindle is No.T4
N50 T5；	The cutter on spindle is No.T4
N60 M30	
%	

The cutter on the spindle is No. T4 after the tool-change programs are performed.

VOLUME I PROGRAMMING

VOLUME TWO OPERATION

CHAPTER ONE OPERATION PANEL

1.1 Panel Classification

GSK218MC series contains of **GSK218MC, GSK218MC-H, GSK218MC-V and GSK218MC-U1; wherein, GSK218MC and GSK218MC-U1 CNC** system are used the assembly structure, **GSK218MC-H and GSK218MC-V** are separately used the horizontal structure and vertical structure. Panel divides into LCD area, Editing keyboard area, sofkey function area and machine tool control area; refer to the following figures:

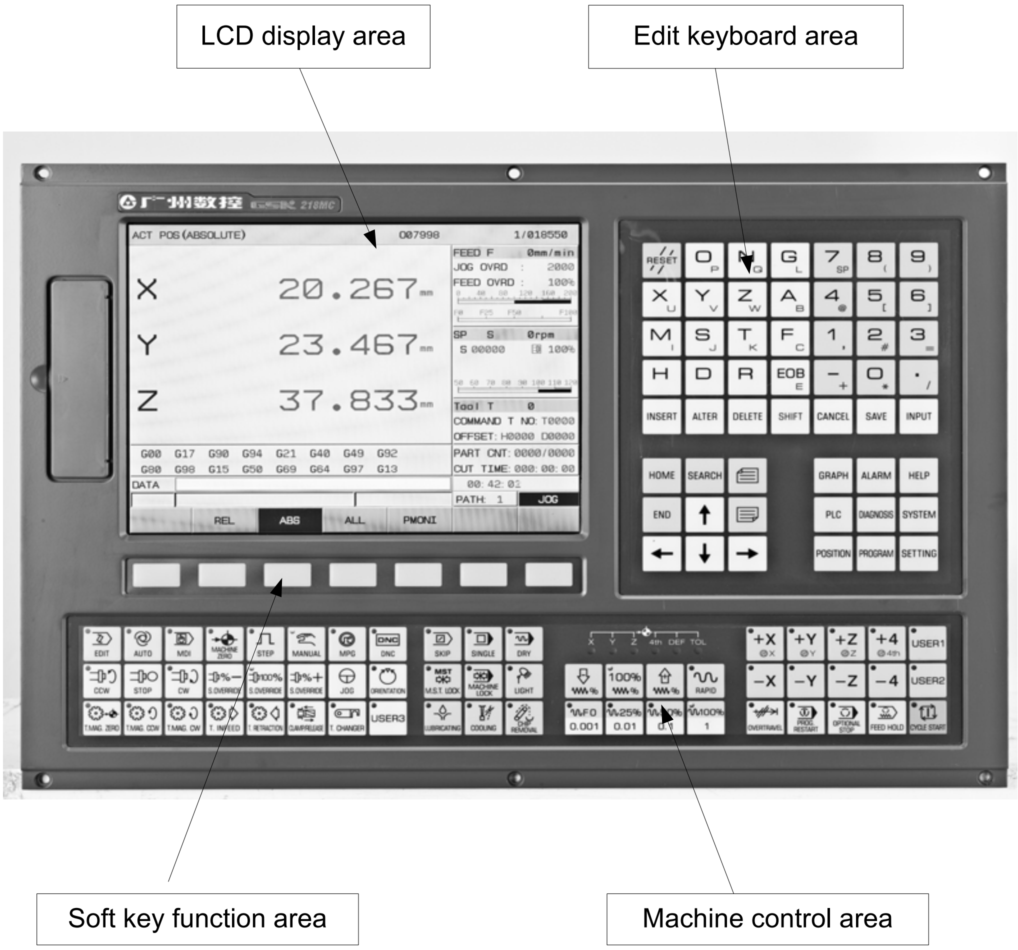


Fig. 1-1-1 GSK218MC panel

Fig. 1-1-2 GSK218MC-H panel

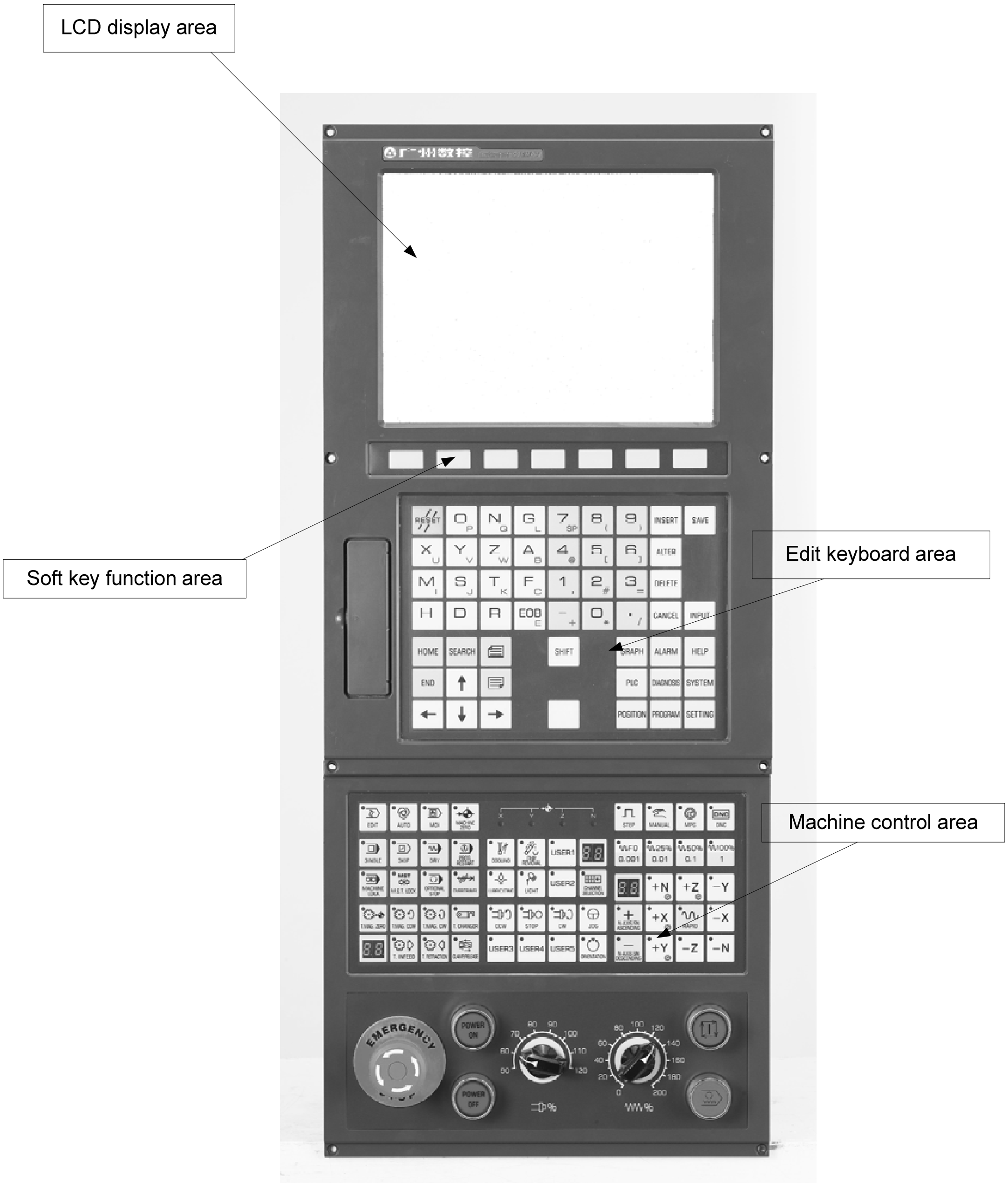


Fig. 1-1-3 GSK218MC-V panel

Fig. 1-1-4 GSK218MC-U1 panel

1.2 Panel Function Explanation

1.2.1 LCD Display Area

GSK 218MC and GSK 218MC-V systems are used the colorful 10.4 inch LCD with 800×600 resolution.

GSK 218MC-H and GSK 218MC-U1 systems are used the colorful 8.4 inch LCD with 800×600 resolution.

1.2.2 Editing Keyboard Area

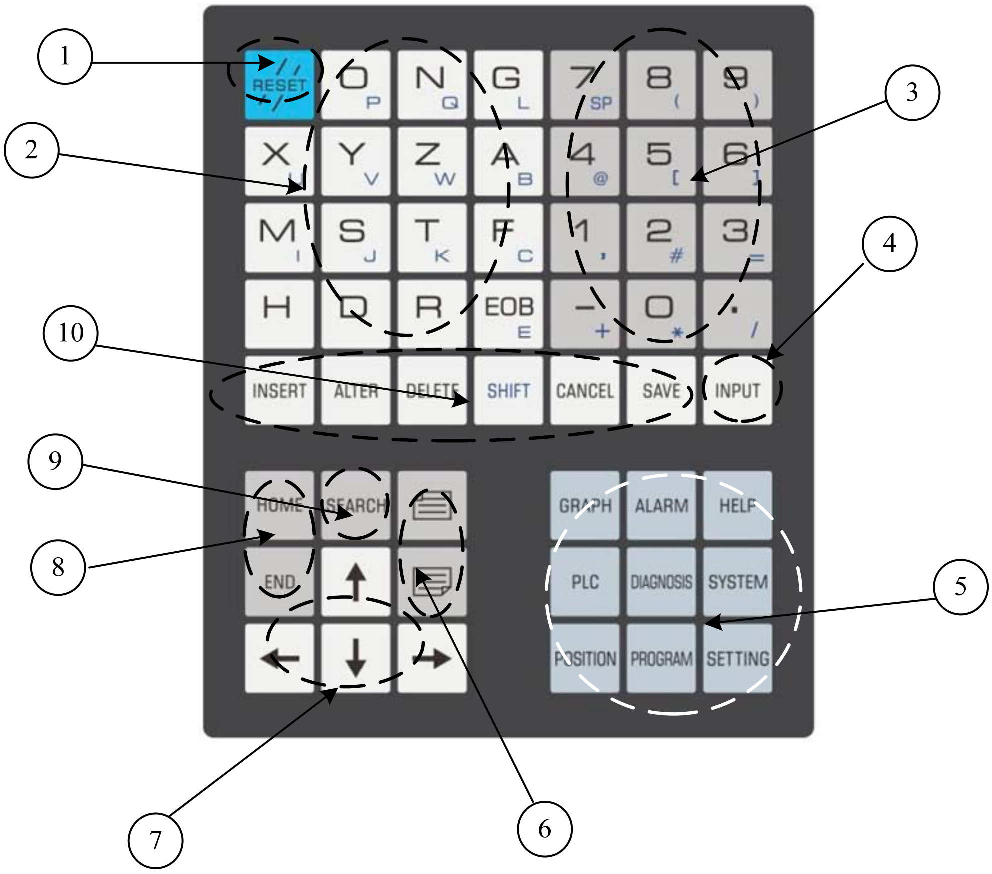


Fig. 1-2-2-1 GSK218MC及GSK218MC-H Editing keyboard area

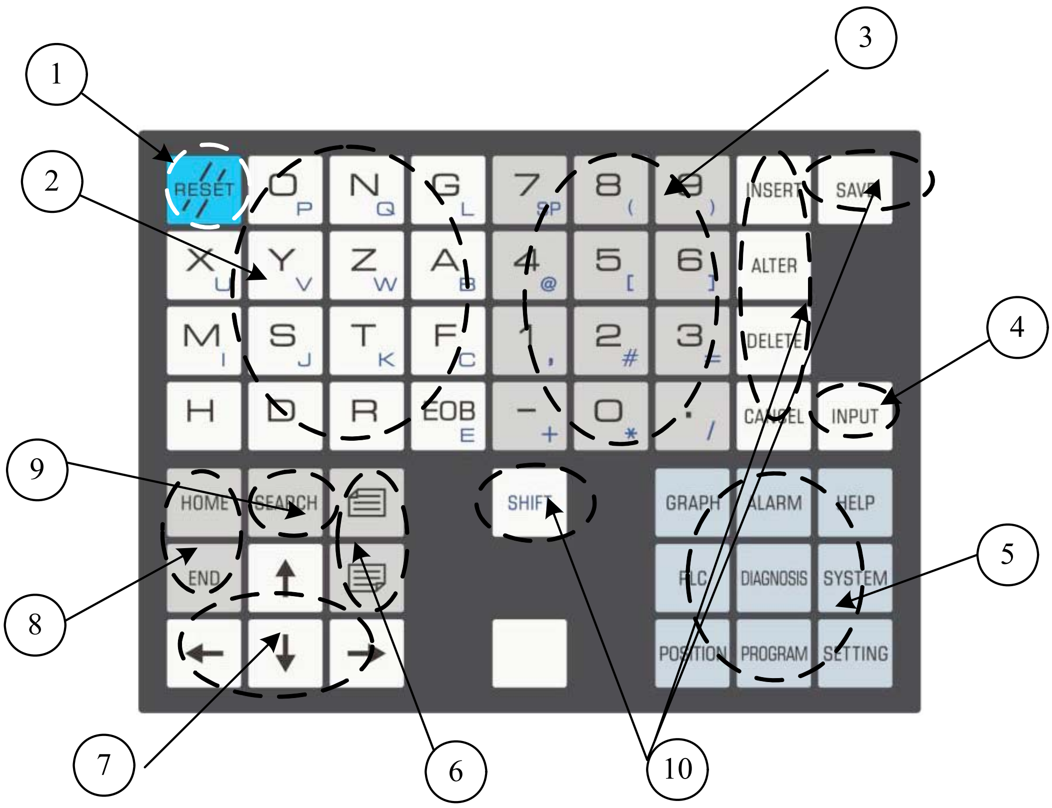


Fig. 1-2-2-2 GSK218MC-V Editing keyboard area

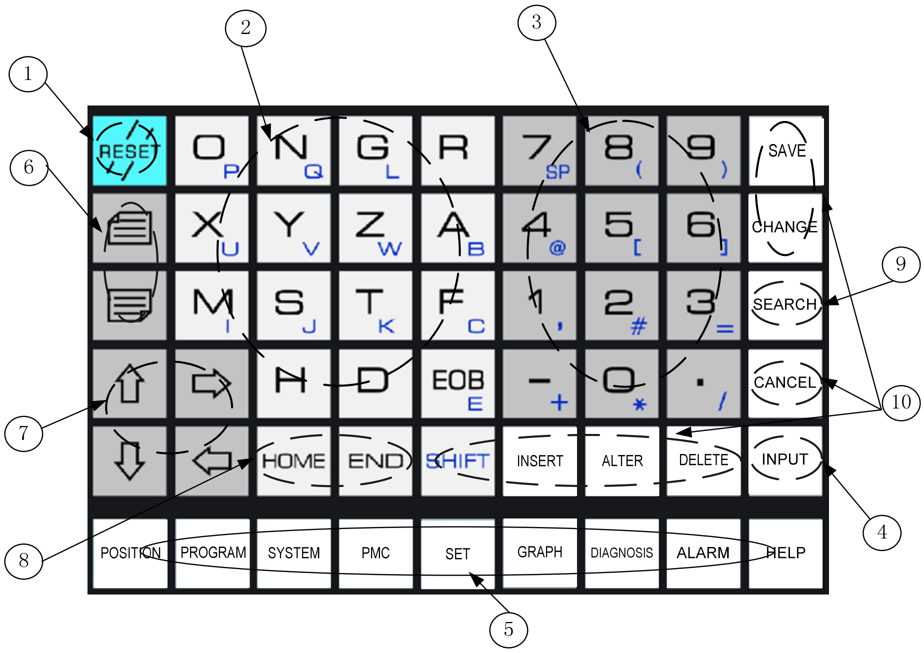


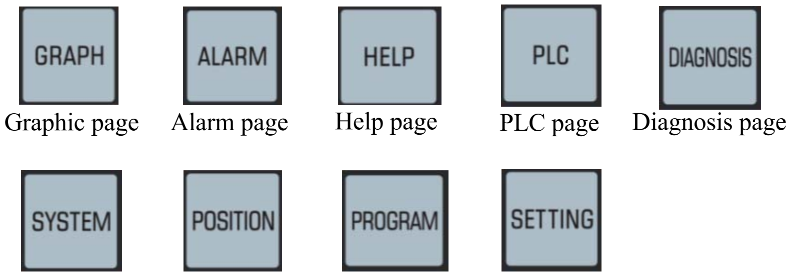
Fig. 1-2-2-3 GSK218MC-U1 Editing keyboard area

In the editing keyboard area, the function of buttons are divided into 10 areas again; the concrete usages are shown below:

Series No.	Description	Function Explanation
1	Resetting button	System resets, feed or output stops.
2	Address button	Enter the MDI addresses
3	Number button	Enter the MDI numbers
4	Input button	Input the numbers, addresses or data to the buffering area to confirm the operation result
5	Screen operation button	Control any of the buttons, enter to the corresponding interface display. (Refer to the CHAPTER THREE for details)
6	Page button	It is used for the page shifing and the page up/down of program in the same display mode.
7	Cursor movement button	It is used for moving the cursor up/down or left/right.
8	Editing button	It can be moved the cursor to the start or end of the program line and the program.
9	Searching button	It is used for searching the data, address to check or modify.
10	Editing button	It is used for operations, suchas the insertion, modification or deletion of program or word field, etc. as well the usage of complex buttons when the program is compiled.

1.2.3 Introduction of Screen Operation Buttons

There are 8 operation page display buttons and one help page display key are displayed on operation panel for this system, refer to the following figure:



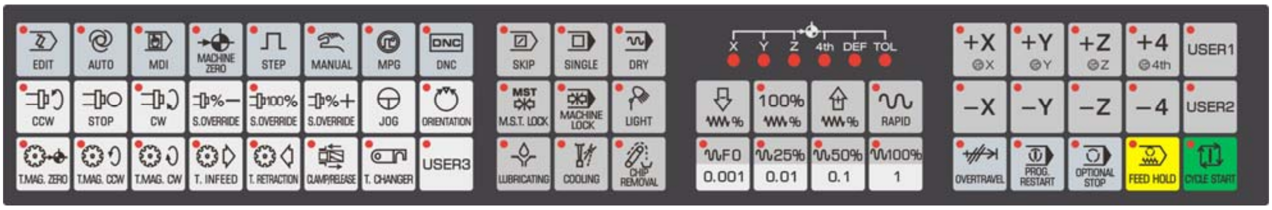
Graphic page　Alarm page　Help page　PLC page　Diagnosis page

System page　Position page　Program page　Setting page

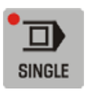
Description	Function Explanation	Remark
Graph page	Enter the graph page	Display the graph parameter and its graph display page by the corresponding softkey conversion. The graph parameter sets the graph center display, dimension and proportion.
Alarm page	Enter the alarm page	Check different alarm information pages by converting the corresponding softkey.
Help page	Enter the help page	Check different help information by converting the corresponding softkey.
Program-control page	Enter the program-control page	Check the configutration of the relevant version information and system I/O port of the PLC ladder diagram by corresponding softkey conversion; simultaneously, the PLC ladder diagram can be altered in MDI mode.
Diagnosis page	Enter the diagnosis page	Check the I/O port signal state at each side of system by corresponding softkey conversion.
System interface	Enter the system page	Display cutter offset, parameter, macro variable and screw compensation display pages by the corresponding softkey conversion.
Position page	Enter the position page	Display the relative, absolute, complex and program-monitoring display pages by the corresponding softkey conversion.
Program page	Enter the program page	Display the program, MDI, CUR/MOD, CUR/NXT, list display page; the multi-page program names can be checked by the page buttons on the list interface.
Setting page	Enter the setting page	There are 4 interfaces, such as the display setting, workpiece coordinate, data and password setting, which can be converted by corresponding softkey.

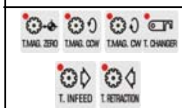
Note: The above-mentioned each softkey conversion interface can also be carried out by using the consecutive corresponding function buttons based upon the setting of bit 0~7 of parameter No.:25 and bit 6~7 of parameter No.: 26. Refer to the CHAPTER THREE of this manual for the details.

1.2.4 GSK218MC Machine Tool Control Area

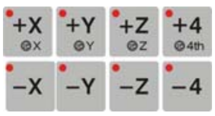
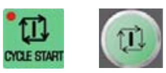


Fit. 1-2-4-1 GSK218MC Machine control area

Button	Description	Function	Remark & Operation
	Selection button in Edit mode	Enter the editing operation method	Shift to the editing method in Auto, MDI and DNC operation mode, the system decelerates to stop after operating the current block.
	Selection button in Auto mode	Enter the Auto operation method	System selects the internal memory program
	Selection button in MDI mode	Enter the MDI operation method	Shift to the MDI mode in Auto; the system decelerates to stop after operating the current program
	Selection button in mechanical zero mode	Enter the mechanical zero operation method	Shift to zero mode in Auto; the system is immediately decelerated to stop.
	Selection button in manual single step mode	Enter the manual single operation method	Shift to the single mode in Auto; the system is immediately decelerated to stop.
	Selection button in manual mode	Enter the manual operation method	Shift to manual mode in Auto; the system is immediately decelerated to stop.
	Selection button in MPG mode	Enter the MPG operation method	Shift to the manual method in Auto; the system is immediately decelerated to stop.
	Selection button in DNC	Enter the DNC operation method	Shift to DNC mode in Auto; the system decelerates to stop after performing the current block.
	Optional switch in block	Whether the "/ at the beginning of a block is skip; the indicator is ON when opening, program then skips.	Auto, MDI, DNC
	Single block switch	Program single block/consecutive operation state shifting; it is single operation when the indicator is ON.	Auto, MDI, DNC

Button	Description	Function	Remark & Operation
	Dry run switch	Indicator is ON when the dry run is enabled.	Auto, MDI, DNC
	Miscellaneous function switch	Indicator is ON when MST function is ON, M, S and T function outputs are disabled.	Auto, MDI, DNC
	Machine lock switch	Indicator is on when machine locking is ON; the axis operation output is disabled.	Auto, MDI, Mechanical zero, MPG, Single-step, Manual and DNC
	Machine working indicator switch	Machine working indicator ON/OFF	Any mode
	Lubrication switch button	Machine lubrication ON/OFF	Any mode
	Coolant switch button	Coolant ON/OFF	Any mode
	Chip-removal switch button	Chip-removal ON/OFF	Any mode
	Spindle control button	Spindle positive Spindle stop Spindle reverse	MPG mode, single-step mode and manual mode
	Spindle override button	Spindle speed adjustment (The spindle speed analog value control method is enabled)	Any mode
	Spindle JOG switch	Spindle JOG state ON/OFF	Manual mode, single-step mode and MPG mode
	Spindle exact stop button	Spindle exact stop ON/OFF	Manual mode, single-step mode and MPG mode
	Tool-magazine operation button	Tool magazine operation ON/OFF	Manual mode
	Manual releasing/clamping switch	Manual releasing/clamping switch	Manual mode

VOLUME II OPERATION

Keys	Designation	Explanation	Remarks and operation explanation
	Manual tool change key	Manual tool change	MANUAL mode
	Overtravel release key	An alarm occurs if the hard limit is reached. Press this key with its indicator lighting up to move the machine reversely till the indicator goes off.	MANUAL mode, MGP mode
	Program restart key	For exiting the running program or restoring to the last machining state before a sudden power loss	Auto mode (the distance to go is the straight-line distance from the current point to the break point)
	Optional stop ON/OFF key	Whether the operation is stopped after a block containing M01 is executed.	Auto mode, MDI mode, DNC mode
	Feedrate override key	Rapid traverse ON/OFF	Any mode
	Rapid traverse key	Rapid traverse ON/OFF	Manual mode
	Rapid, Step, and MPG override keys	For selecting rapid override, manual step override and MPG override	Auto mode, MDI mode, Machine zero return, MPG mode, Step mode, Manual mode, DNC mode
	Manual feeding key	For positive/negative movement of X, Y, Z and 4th axes in MANUAL mode and Step mode, and the axis moved in positive direction is selected by MPG	Machine zero return mode, Step mode, Manual mode, MPG mode
	Channel selection key	For machining channel switch (The function is unavailable temporarily)	Any mode
	Feed hold key	Press this key to stop Auto operation	Auto mode, MDI mode, DNC mode
	Cycle start key	Press this key to start program Auto operation	Auto mode, MDI mode, DNC mode

---

Note: When the symbol numbers at the beginning of a block is more than 1, that is, the enabling skip function does not open, the system then skips this block.

1.2.5 GSK218MC-H and GSK218MC-V Machine Tool Control Area

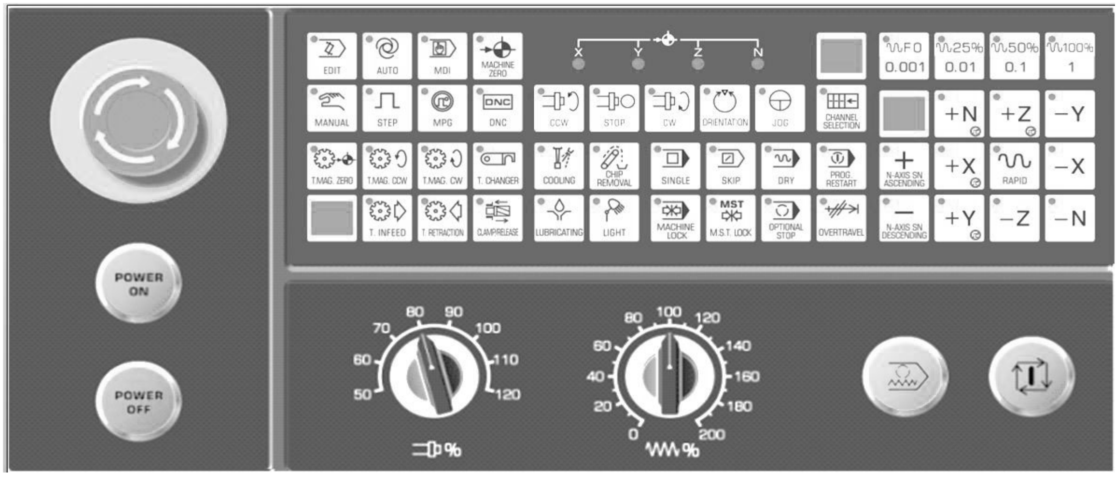


Fig. 1-2-5-1 GSK218MC-H Machine tool control area

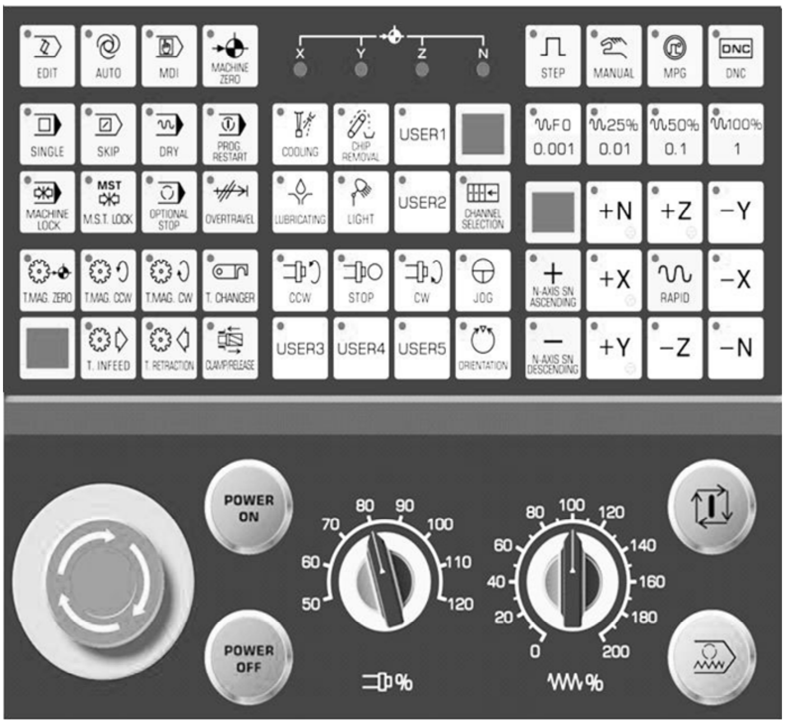
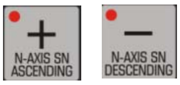


Fig. 1-2-5-2 GSK218MC-V Machine tool control area

The usages and function definitions of the basis buttons on the GSK218MC-H and GSK218MC-V machine tool control area are absolutely coincided with the GSK 218MC. Here, we just describe the new additional buttons:

Button	Description	Function	Remark & Operation
	ESP button	To make the system entering the ESP state	Any mode
	N axis selection button	Axis shifting for multi-axis	Manual mode, single mode and MPG mode
	Spindle override switch	Spindle speed adjustment (The spindle speed analog value control method is enabled)	Any mode
	Feed override switch	The adjustment of feedrate	Auto mode, MDI mode, Manual mode and DNC mode

1.2.6 GSK218MC-U1 Machine Tool Control Area

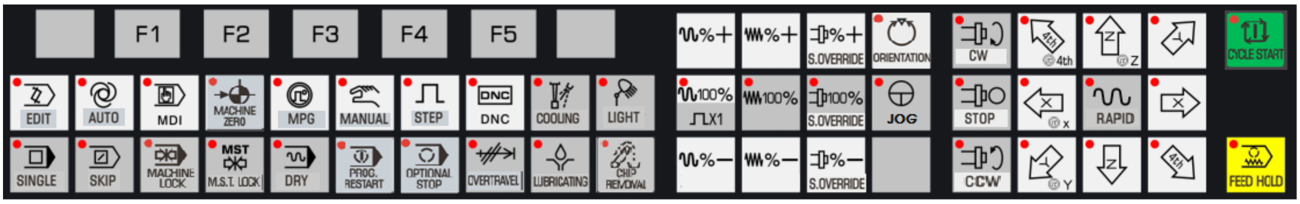
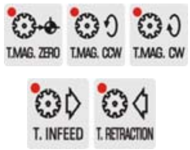
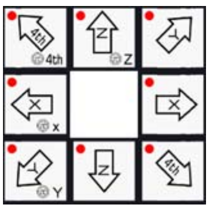


Fig. 1-2-6-1 GSK218MC-U1 Machine tool control area

The usages and function definitions of the basis buttons on the GSK218MC-U1 machine tool control area are basically coincided with the GSK 218MC. Here, we just describe the differences between GSK218MC-U1 and GSK218MC:

Button	Description	Function	Remark & Operation
	Tool-magazine operation button	Tool magazine operation ON/OFF	There is no tool-magazine relevant button from 218MC-U1.
	Manual releasing/clamping switch	Manual releasing/clamping switch	There is no tool releasing/clamping button from 218MC-U1.
	Manual tool-change	Complete the manual tool-change	There is no manual tool-change button from 218MC-U1
	Channel selection button	The shifting of machining channel (This function does not perform yet)	There is no function button for 218MC-U1
	Rapid override, manual single-step and MPG override selection buttons	Rapid override, Manual single and MPG override selection buttons	Auto, MDI, Mechanical zero, MPG, Single-step, Manual and DNC. The button function of 218MC-U1 is coincided with the 218MC, other than the button icons.
	Movement override button	Rapid traverse ON/OFF	Any mode The button function of 218MC-U1 is coincided with the 218MC, other than the button icons.
	Manual feed button	X, Y, Z and the 4th axis moves positively/reversely in manual, single-step operation methods; positive axis regards as MPG selection axis.	Mechanical zero, Single-step, Manual and MPG. The button function of 218MC-U1 is coincided with the 218MC, other than the button icons.

Note 1: The feed hold button  and cycle start button  of GSK218MC and GSK218MC-U1 are shared the same effect with the  and  of GSK218MC-H and GSK218MC-V. The following introduction of the buttons of 218MC are regarded as an example.

Note 2: In Manual mode, the manual speed override is adjusted by the feed override switch in the case of the rapid traverse buttons are not controlled.

Note 3: The buttons within the < >, in the following explanations, are regarded as panel buttons; within the 【 】 are treated as softkeys below the screen; 【 】is the corresponding interface for the current softkeys; ⊞ means that there is sub-menu within this menu.

CHAPTER TWO SYSTEM ON/OFF & SAFETY OPERATION

2.1 System ON

The following items should be confirmed before the power of the **GSK218MC** CNC system turned on.

1. The machine tool state should be normal.
2. The power voltage should be conformed with the requirements.
3. The wiring should be correct and firm.

The current position (relative coordinate) page displays after the system self-checking is normal, and the initialization is completed.

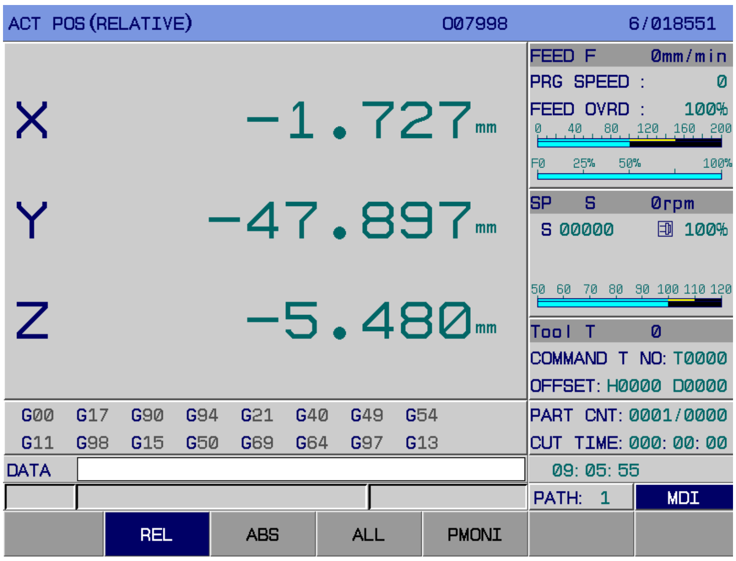


Fig. 2-1-1

2.2 Power OFF

It is necessary to confirm the following items before the power is turned off.

1. The X, Y and Z axes of the CNC are on the stop state;
2. The miscellaneous functions (such as spindle, water-pump, etc.) are closed;
3. Firstly cut off the power of CNC, then the machine tool.

The following inspections should be performed when the power is turned off:

1. The LED cycle start on the operation panel should be located at the stop state;
2. The overall movable components of the CNC machine should be on stop state;
3. Turn off the machine by POWER OFF button.

Cut off the power in an emergency

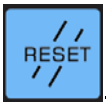
VOLUME II OPERATION

The power of machine tool should be cut off immediately in case of emergency during the operation of the machine for avoiding an accident. However, it is very important to note that the system coordinate may offset to the actual position after the power is turned off; and therefore, the operations such as the zero, tool-setting, etc. should be performed again.

Note: Refer the operation for cutting off the machine power based upon the Machine Tool User Manual manufactured by the factory.

2.3 Safety Operation

2.3.1 Resetting Operation

The system is on the resetting state after pressing 

.

1. The movement of overall axes are stopped.
2. M function stops.
3. Modify the bits 1～7 of parameter No.: 35 and bits 0～7 of parameter No.:36; set whether the G codes of each group is reserved after resetting.
4. Modify the bit 7 of parameter No.:34; set whether cleans the F, H and D codes after reseeting.
5. Modify the bit of parameter No.: 28; set whether delects the compiled program after resetting in the MDI mode.
6. Modify the bit 3 of parameter No.: 10; set whether cancels the relative coordinate system after resetting.
7. Modify the bit 7 of parameter No.: 10; set whether the resetting cursor turns to the beginning of the program in the non-compiled method.
8. Modify the bit 7 of parameter No.: 52; set whether cleans the macro program local variable #1～#50 after resetting.
9. Modify the bit 6 of parameter No.: 52; set whether cleans the macro program common variable #100～#199 after resetting.
10. It can be used in the system output abnormality and coordinate axis abnormality.

2.3.2 ESP

The system enters the ESP state after pressing the ESP button during the machine tool is being operated. In this case, the machine tool is immediately stopped. The ESP can be removed after releasing this button (Although it differs depending on the manufactures, usually, rotate this button left, it can be automatically skipped).

Note 1: It is necessary to confirm that the fault reason is eliminated or not before releasing the ESP button.

Note 2: Perform the reference point return operation again after the ESP button is released to ensure the correct of the coordinate position.

Generally, the ESP signal is NC contactor signal, when the contractor is cut off, the system then enters the ESP state, and the machine is emergently stopped accordingly. The ESP circuit connection is shown below:

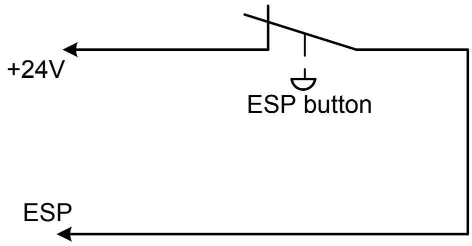


Fig. 2-3-2-1

2.3.3 Feed Hold

The operation dwell by  during the operation of the machine tool. It is essential to note that the dwells performs after the current codes are carried out in the rigid tapping or cycle code operation.

2.4 Cycle Start & Feed Hold

In the controllable panel,  and  are used for the program's start and dwell in Auto, MDI and DNC modes. Set whether uses the external start and dwell by modifying the K5.1 in the PLC address.

Note 1: The shifting is performed among the Auto, MDI and DNC modes. The cycle start is enabled before the current block is performed, and the feed hold is disabled by the <FEED HOLD> button.

Note 2: Shift to editing mode in the Auto, MDI and DNC modes. The cycle start is disabled before the current block is performed, and the feed hold is disabled by the <FEED HOLD> button.

Note 3: Shift to machine tool zero return, single step, manual and MPG modes in the Auto, MDI and DNC modes. The feed hold is disabled by the <FEED HOLD> button.

Note 4: In the cycle start mode, when shifting among the Auto, MDI and DNC modes or when the shifting is editing mode. Press the <FEED HOLD> button before performing the current block, and the feed hold function is on the disabled state.

2.5 Overtravel Defense

To avoid the machine damage due to the X axis, Y axis and Z axis are exceeded the overtravel, and the machine should be taken the overtravel defense measure.

2.5.1 Hardware Overtravel Defense

Separately install the stroke limit switch at the Max. stroke with positive or negative along the X axis, Y axis or Z axis, when the overtrvel occurs, the operation axis decelerates and stops eventually after touching the limit switch, the system then prompts the overtravel alarm information.

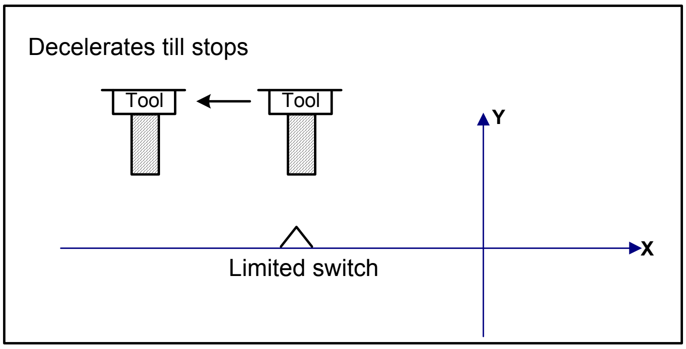


Fig. 2-5-1-1

Detailed explanations:

The overtravel in the Auto mode

In the Auto mode, when the tool touches the limit switch moves along one axis, the overall axes operations may decelerate and then stop; simultaneously, the overtravel alarm occurs, and the program stops at the overtravel block.

The overtravel in the Manual mode

In the manual operation, as long as any axis of the machine tool touches the limit switch, the operations of the overall axes will be immediately decelerated then stopped.

2.5.2 Software Overtravel Defense

The software stroke range is set by data parameter **P66~P73**, and the machine tool coordinate value is regarded as the reference value. If the movement axis exceeds the soft limit parameter setting, the overtravel alarm may occur. The bit 6 of parameter No.: 11 sets whether the stroke inspection (0: No, 1: Yes) is performed after the power is turned on till to the manual reference position return. When the bit 7 of parameter No.: 11 sets the soft limit overtravel, the overtravel (0: Before, 1: After) alarms. The overtravel moves the axis reversely in the <Manual> mode, the alarm is then released after moving out of the overtravel range.

2.5.3 Releasing of Overtravel Alarm

The method for releasing the hard limit overtravel: In the manual or MPG mode, firstly press the 

on the panel, then move out the axis (For example, if it is positive overtravel, it moves out toward to negative, vice versa) reversely.

2.6 Stroke Inspection

Two cutter forbidden areas can be specified by stored stroke detection 1 and 2.

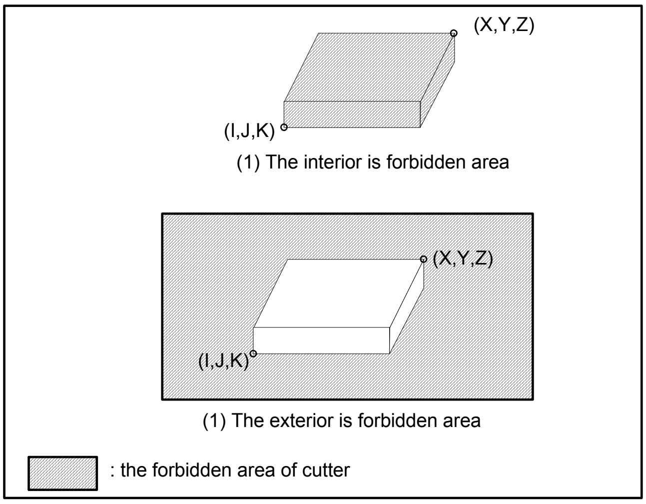


Fig. 2-6-1 Stroke inspection

When the tool exceeds the stored stroke limit, the alarm displays and the machine tool decelerates then stops.

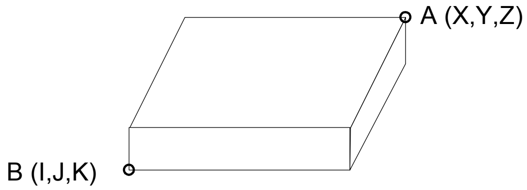
When the tool enters the forbidden area and alarm occurs, the tool can be moved along the reverse direction when it enters.

Detailded explanations:

1. Stored stroke detection 1: This boundary can be set by data parameter **P66~P73,** the exterior of this area is forbidden one. The manufacturer, usually, sets this area as the Max. stroke of the machine tool.

2. Stored stroke detection 2: Data parameter **P76~P83** or the program code can be set this boundary, the area within this boundary or out of the boundary can be set as the forbidden area, which is set by bit 0 of parameter No.: 11 (0: The forbidden area is inside; 1: The forbidden area is outside).

1) When the parameter setting forbidden area is used: the points A and B should be set in the following figure.



X>I,Y>J,Z>K

X-I> ζ (The least command incremental unit)

Y-J> ζ (The least command incremental unit)

Z-K> ζ (The least command incremental unit)

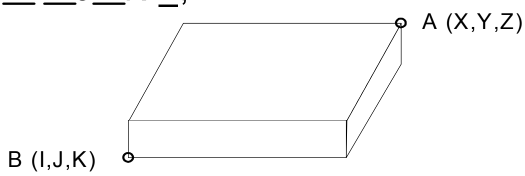
Fig. 2-6-2 Establish or change the forbidden area by parameter

When the forbidden area is set by setting the data parameter **P76~P83**, the data should be offered the distance (output increment) to the machine tool coordinate with the least command incremental unit.

2) When the program is used: G12 prohibts the cutter to enter the forbidden area; G13 allows the cutter to enter the forbidden area.

Each G12 in program should be specified by a separated block, the following commands are used for establishing or changing the forbidden area.

G12 X__Y__Z__I__J__K _;



X>I,Y>J,Z>K

X-I>ζ(The least command incremental unit)

Y-J>ζ(The least command incremental unit)

Z-K>ζ(The least command incremental unit)

Fig. 2-6-3 Establish or change the forbidden area by program

The distance (input increment) in the machine tool coordinate system is specified by the least input incremental unit based upon the setting of the G12.

The programmed data is the digit value of the least code unit converted from the least incremental unit, and this value is then set in the parameter.

Example: The forbidden area is inside (**bit 0 of parameter No. 11=0**):

N1 G12 X50 Y40 Z30 I20 J10 K15;	Set the tool forbidden area point A (5, 40, 30), B (20, 10, 15).
N2 G01 X30 Y30 Z20;	Linear interpolation to (30, 30, 20)
N3 G13;	Cancel the stored stroke detection fuction
N4 G01 X50;	

Example 2: The forbidden area is outside (**bit 0 of parameter No.: 11=1**):

N1 G12 X50 Y40 Z30 I20 J10 K15;	Set the tool forbidden area point A (50, 40, 30), B (20, 10, 15).
N2 G01 X10 Y-10 Z-10;	Linear interpolation to (10, -10, -10)
N3 G13;	Cancel the stored stroke detection fuction
N4 G01 X50;	

3) The inspection point of forbidden area: It is important to confirm the inspection point position (the top of the tool-nose or tool-setting) before programming the forbidden area. Refer to the **Fig. 2-6-4,** for example, the inspection point A (tool-nose), the distance “a” should be set as the data of the stored function inspection; the point B (tool-setting), the distance “b” should be set as the data of the stored function inspection. When the inspection point is A (tool-nose), and the tool length varies from different tools, the forbidden area should be set based upon the longest tool to guarantee the safety operation.

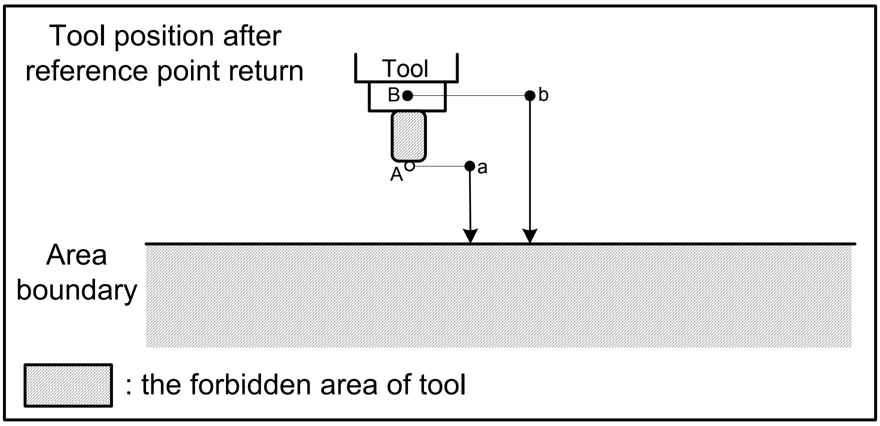


Fig. 2-6-4 Set the forbidden area

4) The overlap of tool forbidden area: the forbidden area can be set by the overlapping method. Refer to the following figure:

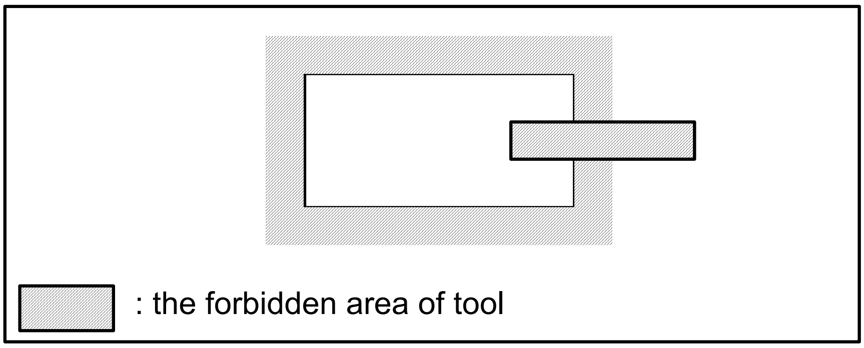


Fig. 2-6-5 Set the overlapped forbidden area

The unnecessary limit should be set out of the stroke of the machine tool.

5) When **bit 6 of parameter No.: 11=0,** the enabled time of forbidden area: after the power is turned on, the manual reference position return is performed or the automatic reference point return is performed by G28, the forbidden area bundary is then enabled inmmediatly.

When the **bit 6 of parameter No.: 11 =1,** after the power is turned on, if the reference position is within the forbidden area, it may alarm immediately, (It is only enabled in the G12 mode in the stored stroke limit 2).

6) Alarm releasing: If it enters the forbidden area and the alarm occurs, the tool is only moved toward the reverse direction. In order to eliminate the alarm, the tool moves along its negative direction, till retreating from the forbidden area, and the system is then reset. The tool can be moves forward or backward after the alarm removes. Refer to the Section 2.5.2 in this operation maual for detail.

7) The alam may immediately occur in forbidden area when G13 converts into G12.

8) The **bit 1 of parameter No.:10** sets the movement whether performs the stroke inspection. When bit 1 of parameter No.: 10 equals to 0, the stroke inspection does not perform before moving; when bit 1 of parameter No.: 10 equals to 1, the stoke inspection is performed before moving.

CHAPTER THREE INTERFACE DISPLAY & DATA MODIFICATION AND SETTING

3.1 Position Display

3.1.1 Four Methods of Position Page Display

Enter the position page display by 

. There are four position display pages: 【RELATIVE COORDINATE】, 【ABSOLUTE COORDINATE】,【COMPOSITIVE】,【PROGRAM MONITORING】, which can be viewed by the corresponding softkeys; refer to the following:

1) Relative coordinate: Display the position of the current tool at the relative coordinate system by the 【RELATIVE COORDINATE】softkey. (Refer to the Fig. **3-1-1-1**).

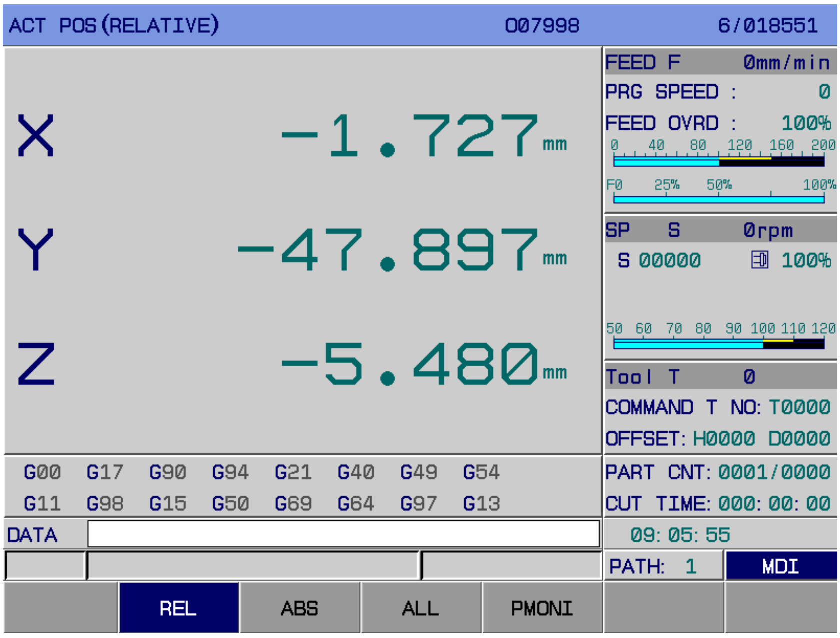


Fig. 3-1-1-1

2) Absolute coordinate: Display the position of the current tool at the absolute coordinate system by the 【ABSOLUTE COORDINATE】softkey. (Refer to the Fig. **3-1-1-2**).

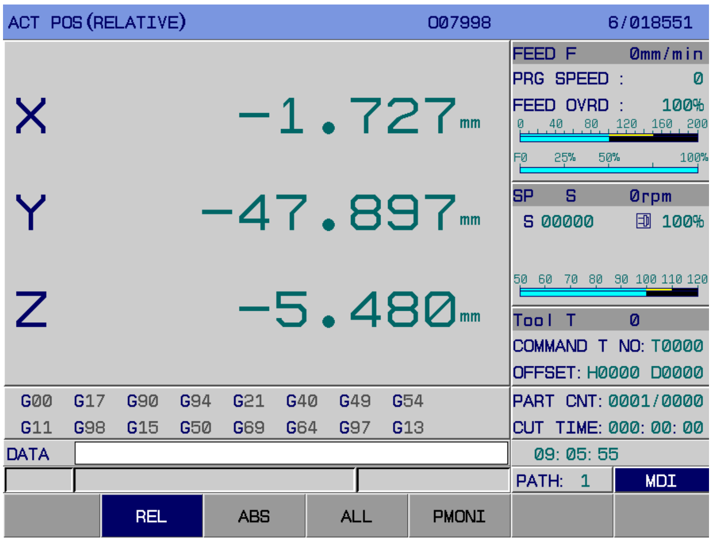


Fig. 3-1-1-2

3) Compositive: Enter the interface by 【COMPOSITIVE】softkey, they will be displayed at the same:

(A) The position at the relative coordinate system;

(B) The position at the absolute coordinate system;

(C) The position at the machine coordinate system;

(D) The offset in the MPG interruption (Shift value);

(E) Velocity component;

(F) Remainder movement value **(It displays in the Auto, MDI and DNC mode)**

The display page is as follows: (refer to the Fig. **3-1-1-3**):

Fig. 3-1-1-3

4) Program-monitoring method

Enter the 【PROGRAM-MONITORING】interface by its software, in this interface, the current position absolute coordinate, relative coordinate can be simultaneously displayed with the modal information and operation block at the current operation. (Refer to the Fig. **3-1-1-4**)

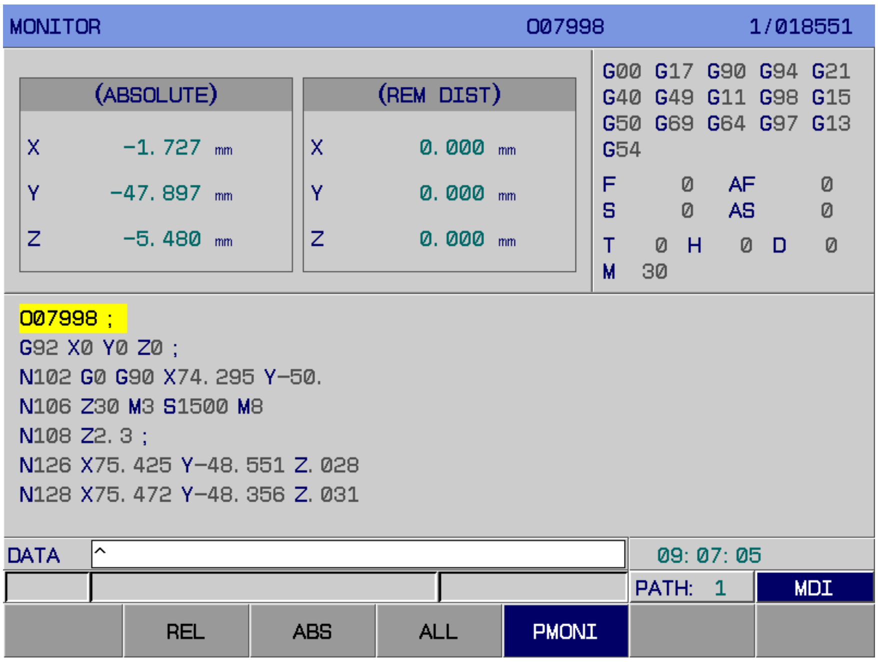


Fig. 3-1-1-4

Note 1: Whether the modal in the program-monitoring display interface can be displayed by bit 6 of parameter No.: 23. When BIT6=0, the modal code does not display on the interface instead of displaying the machine tool coordinate value at the original position.

Note 2: In the <Zero>, <Single-step>, <Manual> and <MPG> modes, the intermediate coordinate system is regarded as the relative one; in the <Auto>, <MDI> and <DNC> modes, the intermediate coordinate system is treated as the remainder movement amount.

3.1.2 Display Machining Time, Component Numbers, Programming Speed, Override and Acutal Speed, Etc. Information

In the 【ABSOLUTE】【RELATIVE】method interface displayed from <Position>, the information, such as the programming speed, actual rate, feedrate, rapid override, G function code, tool offset, machining numbers, cutting time, spindle speed override, spindle speed and machining tool, etc. can be displayed; refer to the Fig. **3-1-2-1:**

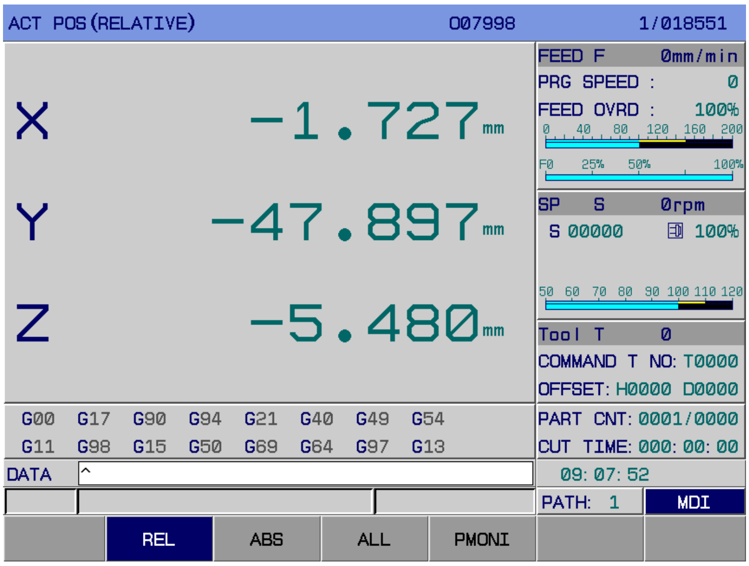


Fig. 3-1-2-1

The concrete meanings are shown below:

Rate: The actual machining rate after the override is performed during the machining.

Programming rate: The rate specified by F code code in a program.

Feedrate: The selected override by feedrate switch.

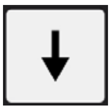
Rapid override: The override controlled by system default panel, or the selected override by rapid traverse wave band switch.

G function code: The value of the G code is being performed at present.

Tool offset: The tool length compensation of the current H0000 machining program; The radius compensation of the current H0000 machining program.

Machining numbers: When machining number of the program adds 1 and perform to M30 or M02 in the Auto or DNC mode, the other modes will not add.

Cutting time: When the Auto operation starts, the timing begins, and its unit is hour, minute and second in turn.

S00000: Command speed. Cursor positions at the "S00000" (Spindle speed) after pressing the direction button  in the relative, absolute coordinate page; in this case, the S value can be modified (the alteration range is the setting value from 0 to data parameter P258).

T0000: The tool number specified by T code in program.

Note: The machining numbers are memorized after thw power is turned off.

The zering method of machining number, cutting time:

1) Shift to the position interface, then select the MDI mode.

2) Cursor positions to the machining number column after controlling the direction button ↓, the data can be input and then press the INPUT to confirm; the machining numbers will be cleared if directly press the INPUT.

3) Shift to cutting time by up/down buttons.

4) Press the INPUT: Cutting time clears.

Note 1: There is no alternative other than to install the encoder on spindle when the actual speed of the spindle is displayed.

Note 2: Actual rate = the F value of programming rate × override. In the G00, the operation speed of each axis is set by data parameter P88～P92, which can be adjusted by rapid override; and the dry run speed can be determined by data parameter P86.

Note 3: The programming rate display of feed per revolution: It displays that the block contains each feed per revolution is being performed.

Note 4: The machined total components can be set by data parameter P356, the desired machining total components can be set by data parameter P357.

3.1.3 Relative Coordinate Clear and Middle

The relative coordinate position clears, the operation steps are shown below:

1) Enter any interface with the relative coordinate display (Refer to the Fig. **3-1-3-1**)

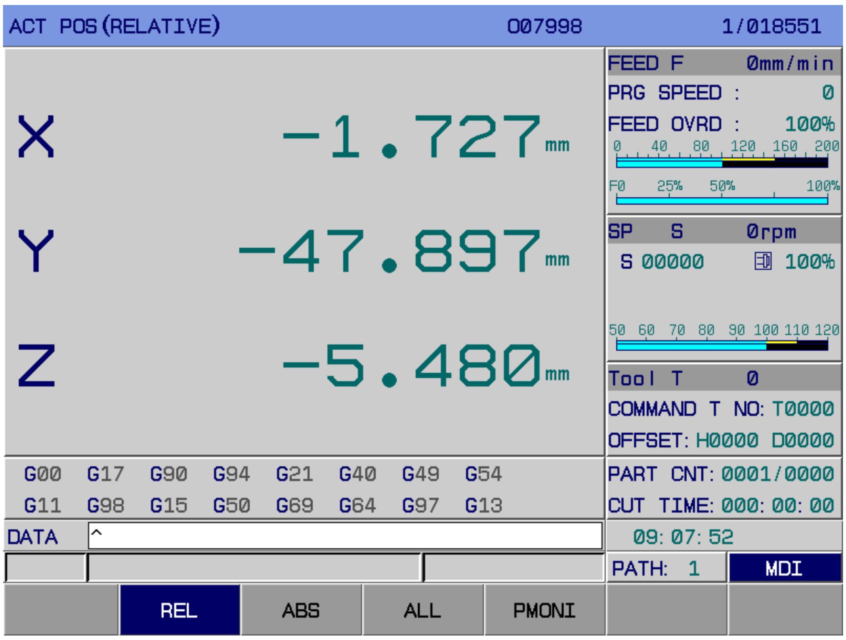


Fig. 3-1-3-1

2) Clearing operation: The X flashes by the "X" button, then press the CANCEL, in this case, the relative coordinate value towards to the X is already cleared (Refer to the Fig. **3-1-3-2);**

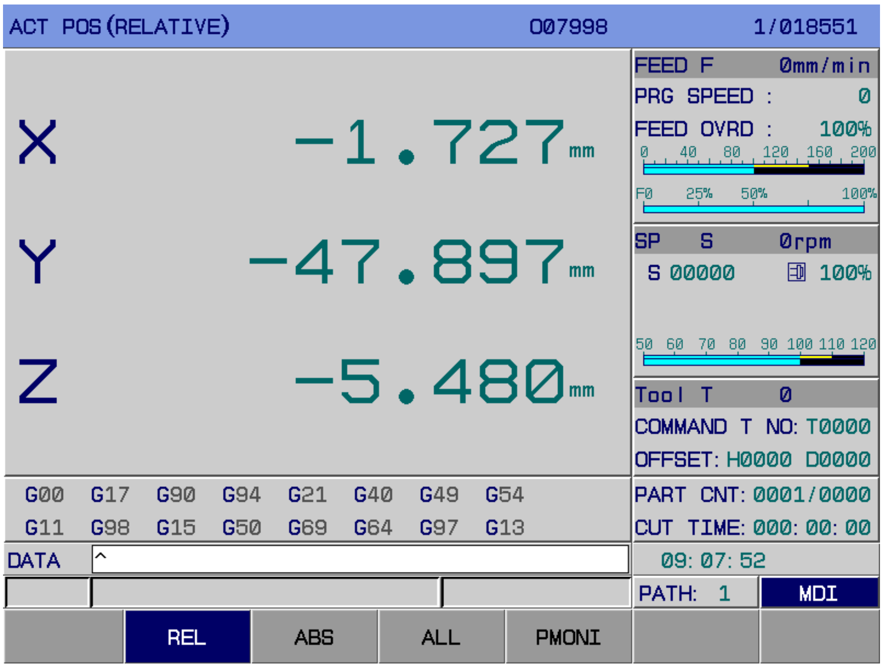


Fig. 3-1-3-2

3) Middle Operation: The X flashes by the "X" button, then press the INPUT, in this case, the relative coordinate value towards to the X is already divided at the middle (The relative coordinate value of this axis is divided by 2)**;**

4) Coordinate setting: The X flashes by the "X" button, then input the desired data confirming by INPUT, the data is inputted to the coordinate system accordingly.

5) The clearing method of the Y and Z axes are identical with the above-mentioned.

3.1.4 Bus Monitoring Position Page Display

When the system uses the Ethernet bus communication method, enter the position page display by POSITION, then the 【MONITORING】interface by its softkey; in this interface, simultaneously, the current machine coordinate, multi-coil position, encoder value, grating position, motor speed and motor loanding (% means the percentage of the rated loading) can be display. It is convenient to debug the machine and monitor the servo current operation state with real-time by this interface. (Refer to the Fig. **3-1-4**)

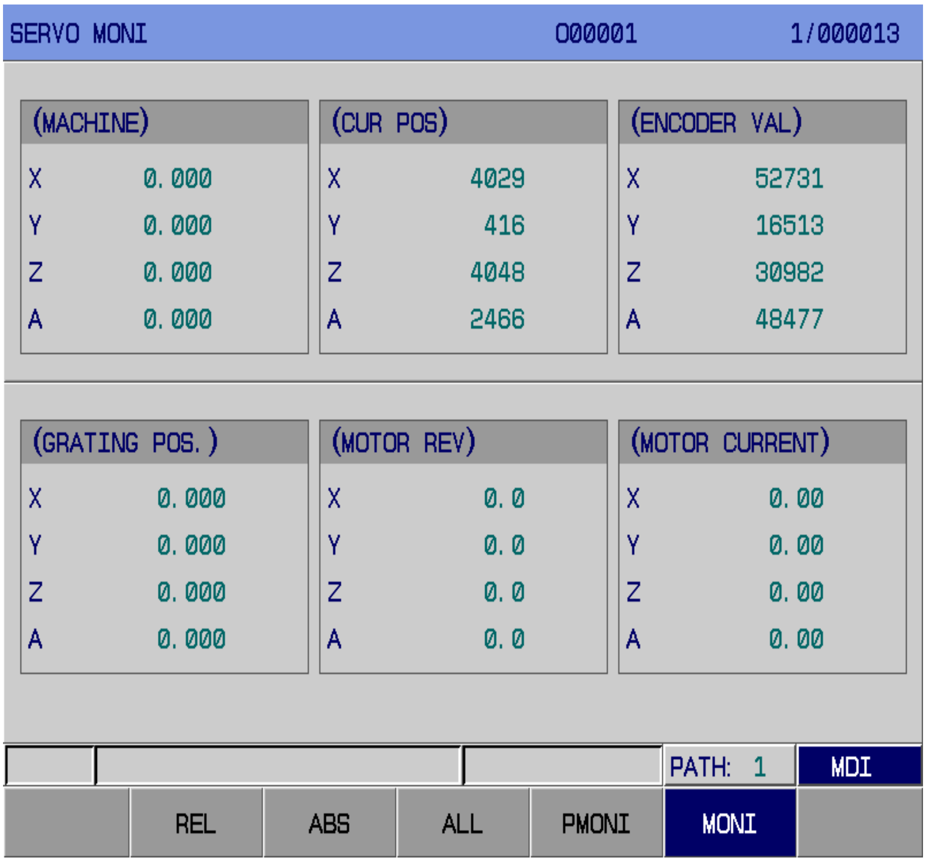


Fig. 3-1-4-1

3.2 Program Display

Enter the program page display by the 

on the panel, there are five interfaces:【⊞ PROGRAM】,【**MDI**】,【CUR/MOD】,【CUR/NXT】and 【LIST】, which can be viewed and altered by the corresponding softkeys, refer to the Fig. **3-2-1** is as follows:

1) Program display

Enter the program display page by【⊞ PROGRAM】softkey, in this page, the program at the page of the block is being performed that displays inside the memory (Refer to the Fig. **3-2-1**).

VOLUME II OPERATION

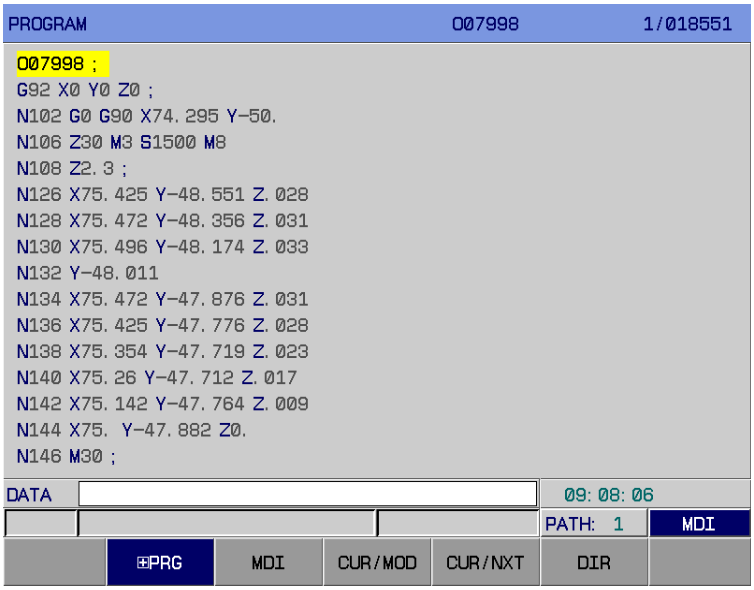


Fig. 3-2-1

Press the【⊞ PROGRAM】again, the editing and modification pages of the program are shown in this interface (Refer to the Fig. **3-2-2):**

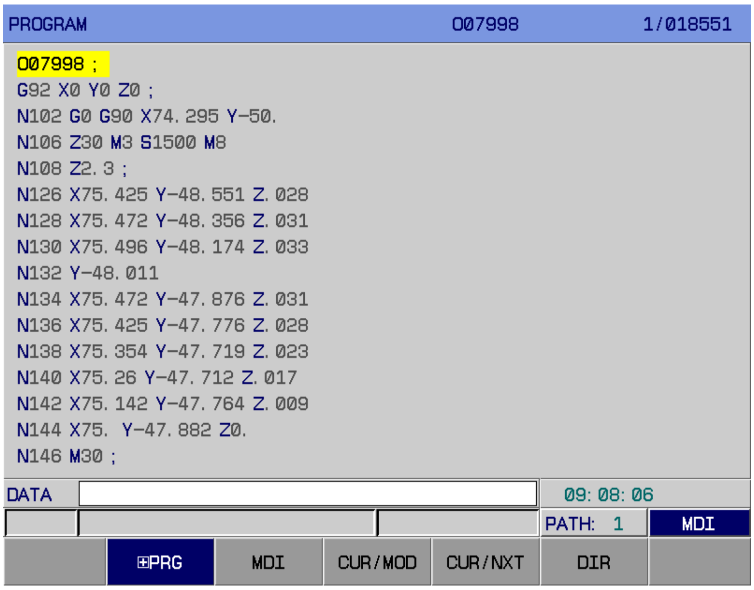


Fig. 3-2-2

Press【▶】 enter to next page

Press 【▶】enter to next page

Press【(】 return to previous page

Note: Wherein, 【CORRECTION】function can only be used in the Auto mode.

【B.EDITING】and【B.END】can only operate this function (Background editing function) in Auto and DNC modes. The editing program of each function for【B.EDITING】 is indentical in the <EDITING> mode; refer to the *Program Editing Operation* of *Chapter Ten of OPERATION.* After the editing is performed, save this editing interface by【B.END】 and retreating from it by【RETURN】.

2) MDI input display

Enter the MDI display interface by【MDI】softkey, in the MDI mode, multi-program can be compiled and performed. The program format is as same as the editing program. MDI operation is suitable for the simple testing program operation (Refer to Fig. **3-2-3).**

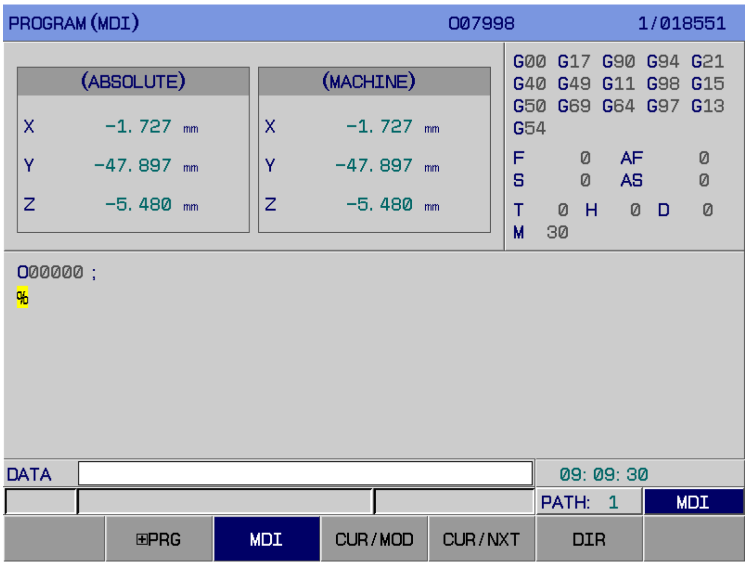


Fig. 3-2-3

4) Program (CUR/MOD) display

Enter the CUR/MOD display page by the【CUR/MOD】softkey, the code value is displayed for the being performed block and the current modal value; the MDI data input and operation can be performed in the MDI mode (Refer to the Fig. **3-2-4).**

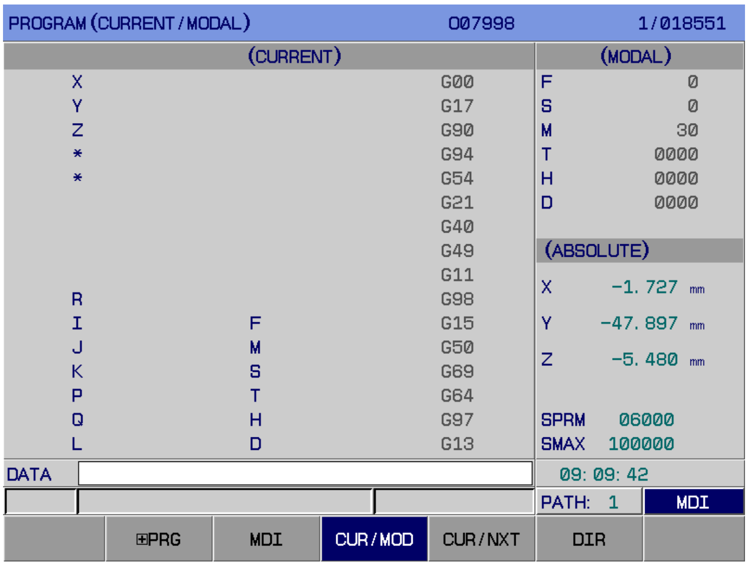


Fig. 3-2-4

4) Program (CUR/NXT) display

Enter the CUR/NXT display page by the【CUR/NXT】softkey, the code value is displayed for the being performed block and the next block to be performed program (Refer to the Fig. **3-2-5**).

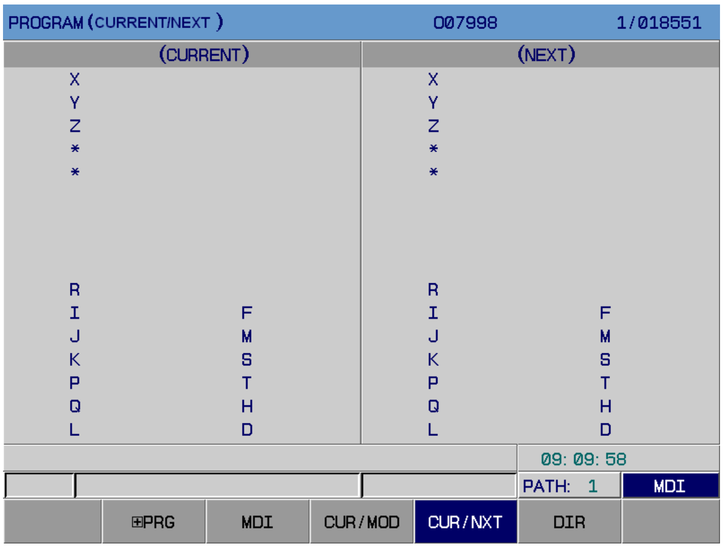


Fig. 3-2-5

5) Program list display

I. Enter the program (System list) display interface by【LIST】softkey, the display content is shown below (Refer to the Fig. **3-2-6**):

(a) Program numbers: the stored program number (include the subprogram) / the Max. program numbers to be registered.

(b) Memory capacity: The occupied memory capacity/program storage total capacity for the stored programs.

(c) Program list: Display the program number of the registered program in turn.

(d) Preview the program at the current cursor located.

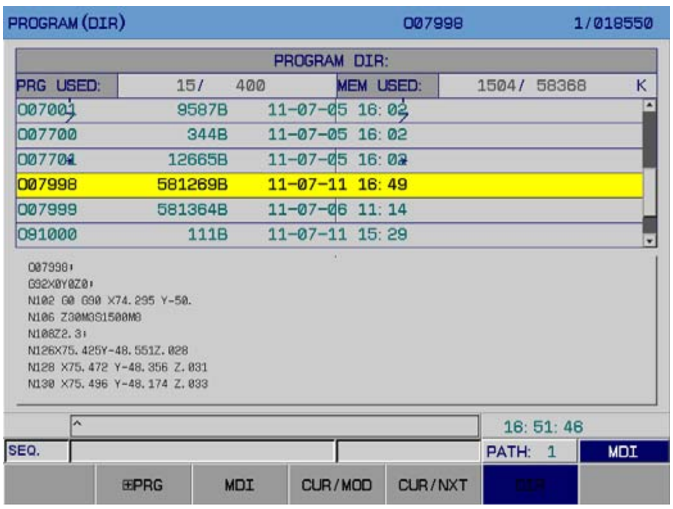


Fig. 3-2-6

II. Enter the program (USB list) display interface by【LIST】softkey, the display content is as

VOLUME II OPERATION

follows (Refer to Fig. **3-2-7**).

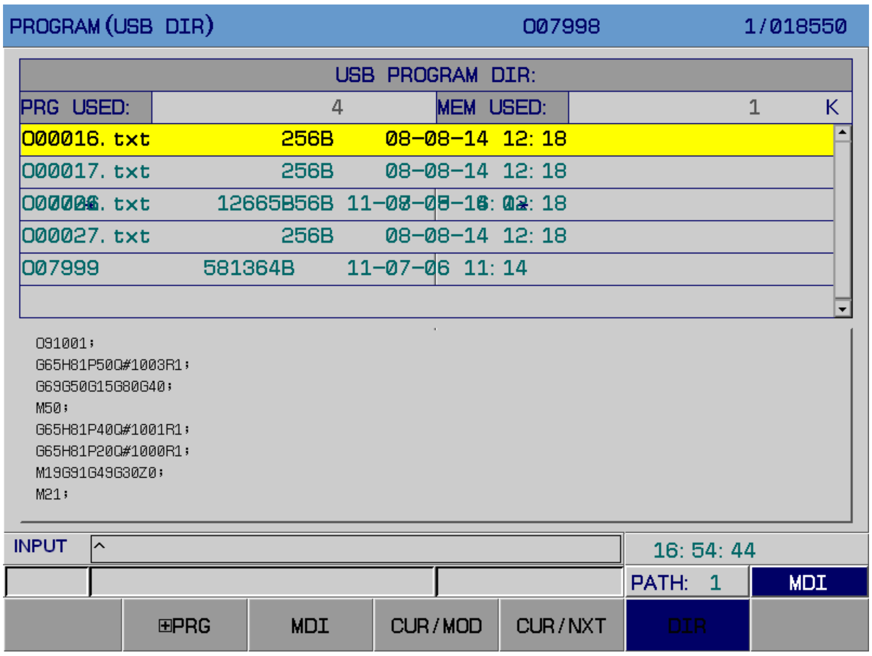


Fig. 3-2-7

Explanation: The overall program numbers within the memory is displayed by the page buttons, the program names can not be viewed if they are more than 6-digit or improper the regular.

3.3 System Display

Enter the system page by 

, there are five sub-interface for this page: 【⊞OFFSET】,【⊞PARAMETER】,【⊞ MACRO VARIABLE】,【PITCH COMPENSATION】 **and**【⊞ BUS CONFIGURATION】, which can be shifted and displayed by its corresponding software. Refer to the following figure (Fig. **3-6-1**):

3.3.1 Display, Modification and Setting of Offset

3.3.1.1 Display of Offset

Enter the offset display page by【⊞OFFSET】softkey, the offset interface is then shown below (Refer to the Fig. **3-3-1-1-1**):

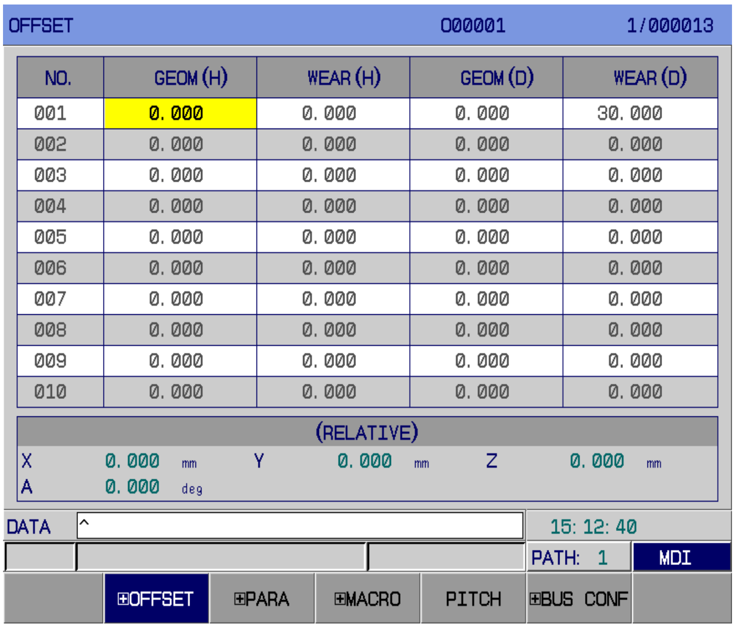


Fig. 3-3-1-1-1

Press the【⊞OFFSET】softkey in the above figure, then enter the operation interface. (Refer to the **Fig. 3-3-1-1-2)**

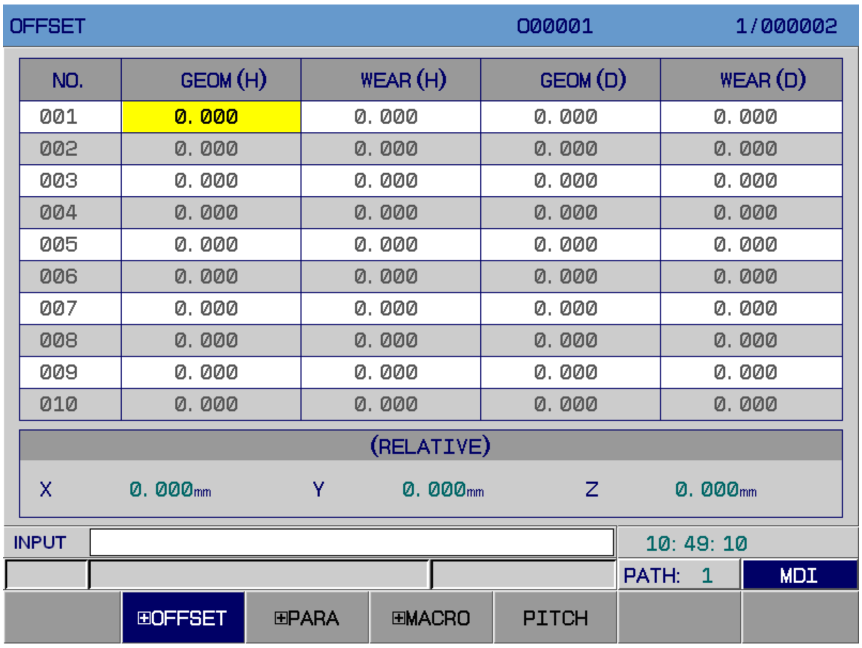


Fig. 3-3-1-1-2

The compensation value can be directly input or performed the addition/subtraction operation with the value at the current position. Shape (H) means the tool length compensation; Wear (H) means tool length abrasion and wear; Shape (D) meand cutter compensation; Wear (D) means cutter radius abrasion and wear.

3.3.1.2 Modification and Setting of Offset Value

The methods for setting the tool offset in the offset interface:

1) Enter the offset display interface by【OFFSET】.

2) Move the cursor to the position where to be inputted the compensation number.

Method 1: Display the page to be modified the compensation value by page up/down button; Move the compensation number position to be modified by cursor.

Method 2: After the compensation number is input, positioning performs by SEARCH.

3) Input the compensation value in any mode, and then confirm it by INPUT or【INPUT】softkey.

4) Input the compensation value in amy mode, and then press【+INPUT】or【-INPUT】softkey; the system will then automatically calculate the compensation value and display it.

Note 1: When the tool offset is changed, the new offset value can be enabled, only when the specified H or D code of compensation number shoul be performed.

Note 2: User can freely alter the tool compensation value during the program operation. When, however, the modification should be completed before performing this tool compensation number, if it can be immediately enabled during program operation.

Note 3: For example, the length compensation value shold be added the relative coordinate value of the Z axis; it is only need to be write the compensation value followed with the Z axis, the system will then automatically overlapped.

For instance, input the Z 10, the compensation value is regarded that the current relative coordinate value of Z axis adds 10.

3.3.2 Display, Modification and Setting of Parameter

3.3.2.1 Parameter Display

Enter the parameter page display by【 PARAMETER】, there are two sub-interface within the parameter page: 【BIT PARAMETER】,【DATA PARAMETER】, which can be viewed or altered by its corresponding softky, is as follows:

1) Bit parameter page Enter the bit parameter interface by【BIT PARAMETER】. (Refer to the Fig. **3-3-2-1-1)**

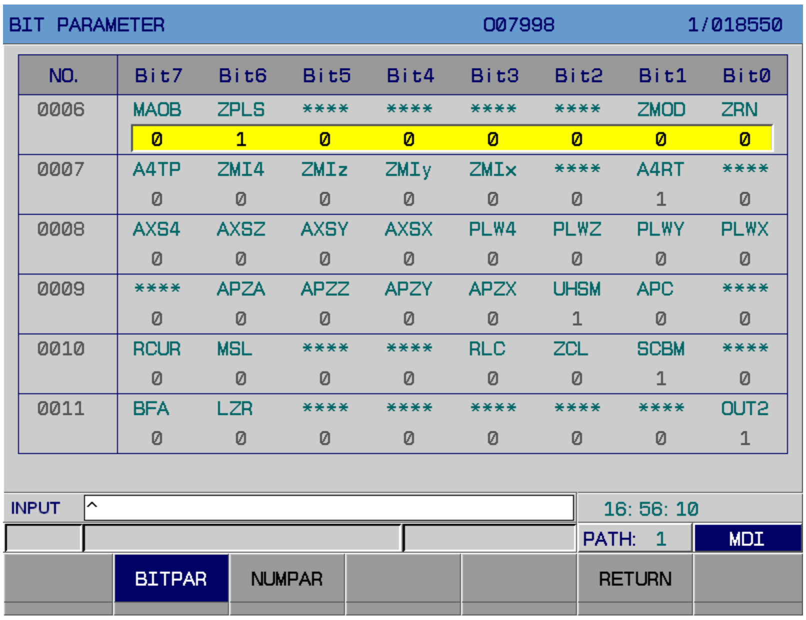


Fig. 3-3-2-1-1

Refer to the *Appendix One Parameter Explanation* for the concrete definitions of each parameter.

2) Data parameter page Enter the data parameter page by【DATA PARAMETER】softkey. (Refer to the Fig. **3-3-2-1-2**):

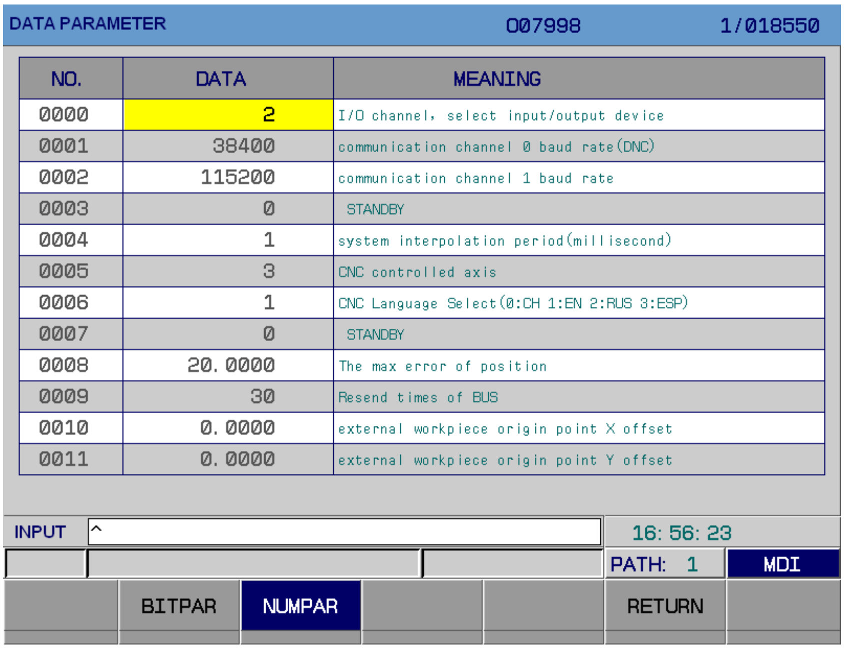
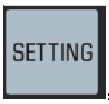


Fig 3-3-2-1-2

Refer to the *Appendix One Parameter Explanation* for the concrete definitions of each parameter.

3.3.2.2 Modification and Setting of Parameter Value

1) Select the <MDI> operation mode.

2) Enter the <SETTING> page by 

, the parameter switch set to “1”.

3) Press 

, then the【⊞ PARAMETER】softkey, lastly enter the parameter display page.

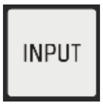
4) Move the cursor to the position where the parameter number to be modified:

Method 1: Display the page to be set the parameter by page up/down button; move the cursor bu direction buttons, and then position the parameter place to be modified.

Method 2: Press 

to position after inputting the parameter number.

5) Input the new parameter value (It is necessary to input the password authority of the corresponding level when different levels parameter are modified) by numerical button.

6) Press the 

to confirm, the parameter value is input and then displays.

7) Close the parameter switch after the overall parameters setting are completed.

3.3.3 Display, Modification and Setting of Macro Variable

3.3.3.1 Display of Macro Variable

Enter the macro variable page display by【⊞ MACRO VARIABLE】softkey, there are two sub-interface within the macro variable page: 【USER VARIABLE】,【SYSTEM VARIABLE】, which can be viewed or modified by its corresponding softkeys, refer to the following concrete descriptions:

1) User variable page Enter the user variable interface by【USER VARIABLE】softkey. (Refer to the Fig. **3-3-3-1-1**):

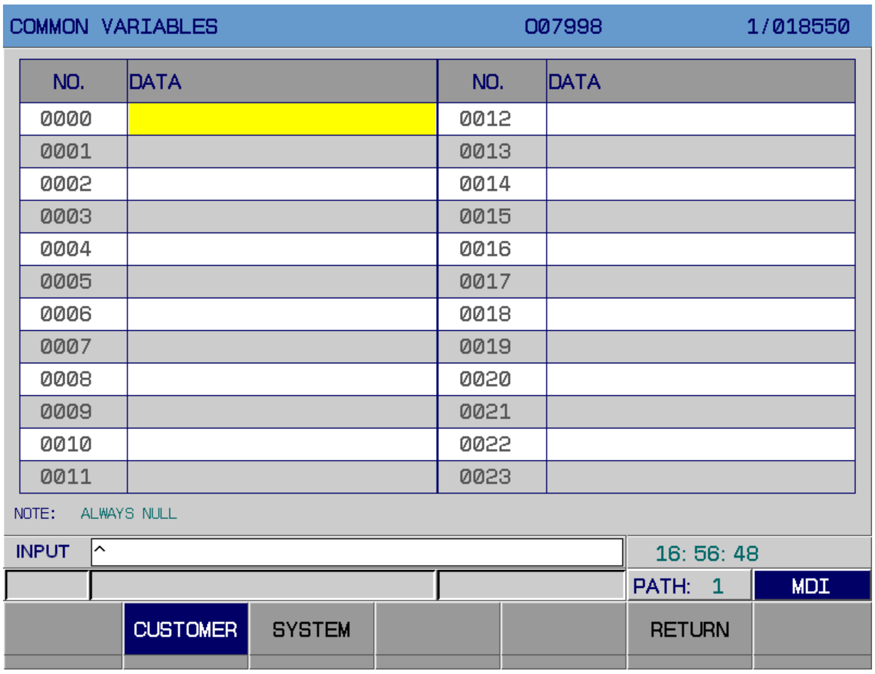


Fig. 3-3-3-1-1

2) System variable page Enter the system variable page by 【SYSTEM VARIABLE】 (Refer to the Fig. **3-3-3-1-2**):

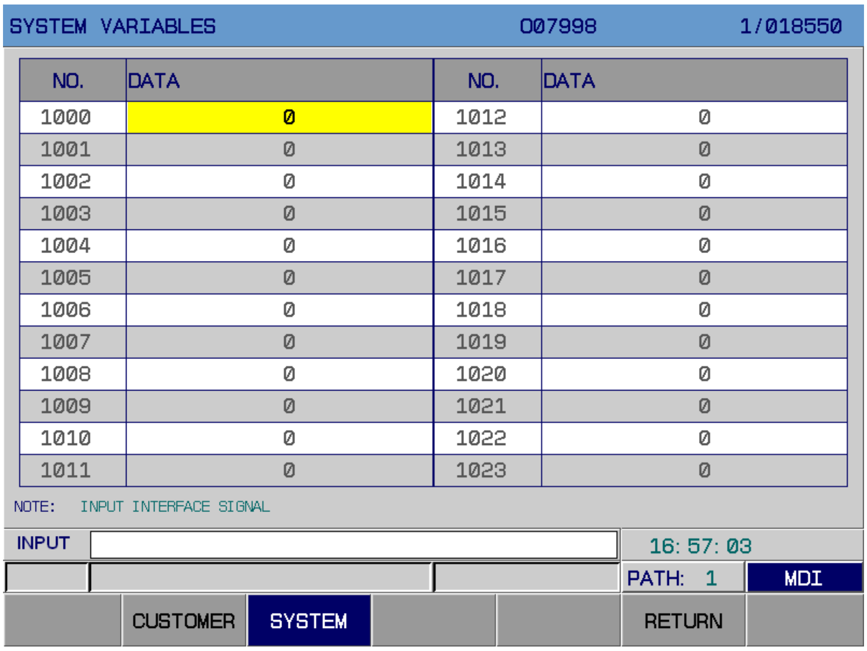


Fig. 3-3-3-1-2

Refer to the *Section 4.7.2 Macro Variable in PROGRAMMING* for the explanation and usage of the macro variable.

3.3.3.2 Modification and Setting of Macro Variable

1) Select the <MDI>operation mode.

2) Press the 

, then【⊞ MACRO VARIABLE】softkey to enter the macro variable display page.

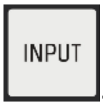
3) Move the cursor to the position where the variable number to be modified:

Method 1: Display the page to be set the variable by page up/down button; move the cursor bu direction buttons, and then position the variable place to be modified.

Method 2: Press the 

to position after inputting the variable number.

4) Input a new numerical value by number buttons.

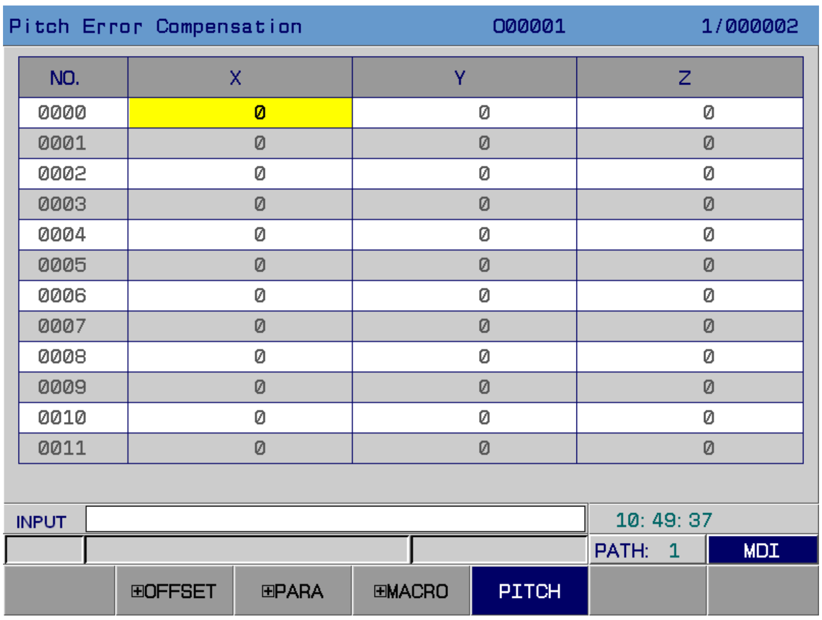
5) Confirm that the numerical value is input and displayed by 

.

3.3.4 Display, Modification and Setting of Ptich Compensation

3.3.4.1 Pitch Compensation Display

Enter the pitch compensation interface by【PITCH COMPENSATION】softkey, the pitch compensation interface is as follows (Refer to the Fig. **3-3-4-1-1**):



NO.	X	Y	Z
0000	0	0	0
0001	0	0	0
0002	0	0	0
0003	0	0	0
0004	0	0	0
0005	0	0	0
0006	0	0	0
0007	0	0	0
0008	0	0	0
0009	0	0	0
0010	0	0	0
0011	0	0	0

Fig. 3-3-4-1-1

3.3.4.2 Modification and Setting of Pitch Compensation

1) The pitch error compensation number of each axis is set by data parameters **P216～P220**; the pitch error compensation interval is set by data parameters **P226～P230.**

2) In the <MDI>, input the compensation amount of each point in turn.

Note: Refer to the INSTALLATION & CONNECTION of the Chapter Four in *GSK218MC CNC System PLC & Installation Connection Manual* for the setting of the pitch compensation.

3.3.5 Display, Modification and Settong of Bus Servo Parameter

Enter the system page by 

, display the【⊞ BUS CONFIGURATION】sub-interface by its corresponding softkey shifting. Refer to the following figure (Fig. **3-3-5-1**):

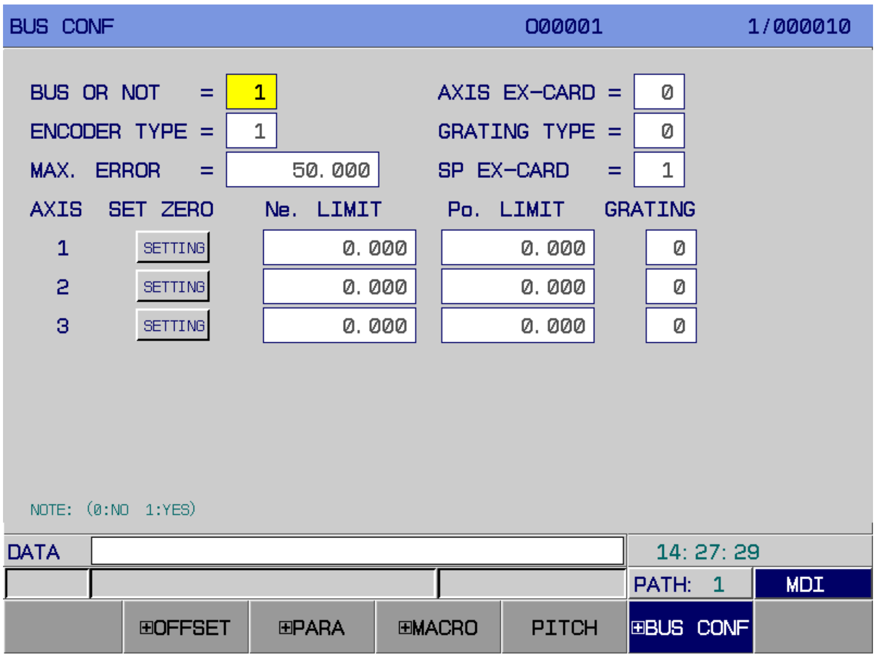


Fig. 3-3-5-1

【⊞**BUS CONFIGURATION**】Interface operation explanations

After entering the bus configuration interface by the【⊞**BUS CONFIGURATION**】softkey,as the Fig. 3-3-5-1, the parameters displayed in this page can be viewed; as well, the relevant parameters can be altered. The concrete operation methods and steps are shown below:

1. Ente <MDI> operation mode;
2. Move the cursor by the up/down/left/right direction buttons to the item to be modified.
3. Modification is performed based upon the following explanations;

   1) Whether it is bus

      0: Drive transmission method is pulse.

1: Drive transmission method is bus.

Note: It can also be set as bus method by the bit 0 of parameter No.: 0.

2) Encoder type

0: Increment 1: Absolute

Note: Set whether use an absolute encoder by the bit 6 of parameter No.: 20.

3) Select the Max. allowable error

Note: System defaults 50.000mm, also, it can be set by data parameter P445.

4) Axis extensive card

0: Without extensive card 1: With extensive card

Note: Set whether use a bus servo card by the bit 6 of parameter No.: 0.

5) Grating type

0: Increment 1: Absolute

Note: Set whether use an absolute grating by the bit 0 of parameter No.: 1.

6) Spindle extensive card

0: Without extensive card 1: With extensive card

Note: Set whether the spindle drive is used the bus control method by the bit 1 of parameter No.: 1.

7) Multi-coil absolute zero setting

b) In the MDI mode, set the "Whether it is bus" to 1 in the bus configuration interface; set the "Encoder type" to 1; set the machine zero position moving each axis manually.

c) Move the cursor to the SETTING, press the <INPUT> twice based upon the prompts, the zero return indicator is ON; the record of the motor absolute encoder along eac axis at the current position is regarded as machine zero; the system is restarted after the power is turned off, and the zero return indicator is still ON. Manually set the negative/positive boundary based upon the Max. stroke of the actual machine tool, so that move the current machine tool an offset value forward or backward, lastly, the bit 6 of parameter No.: 61 is set to 1, the positive/negative is then enabled.

Setting range: -99999.9999~99999.9999, alternatively, the positive/negative boundary along with each axis can be directly set by data parameters **P450~P459.**

D) Whether configure the griding. Separatly set whether each axis is configured with a griding, 0: Without, 1: With. Alternatively, individually set it by **bit 3 and 7 of parameter No.: 1**.

4. Confirm it by INPUT.

Note 1: After the machine tool zero is set, if the zero return direction, feed axis movement direction, servo and system gear ratio of the system are altered, the zero will lost, and therefore, it is necessary to reset the zero of the machine tool.

Note 2: After the machine tool zero is set, the other reference points may be affected, for example, the 2nd and the the 3rd reference points should be set again.

3.3.5.1 Servo Parameter Display

Enter the servo debugging interface by pressing the【⊞ BUS CONFIGURATION】, then enter the servo parameter interface by the【⊞ SERVO PARAMETER】. The content of this interface is shown below (Fig. **3-3-5-1-1**).

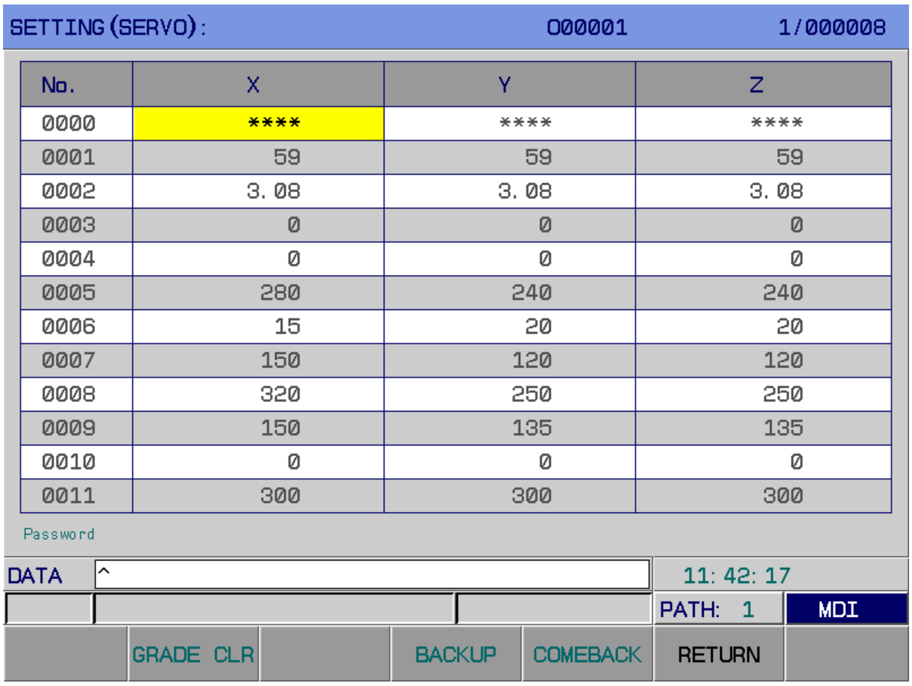


Fig. 3-3-5-1-1

3.3.5.1.1 Modification & Setting of Servo Parameter

1) Select <MDI> operation mode.

2) Enter the <SETTING> interface by SETTING, set the parameter switch to "1".

3) Press the SYSTEM, then the【⊞ BUS CONFIGURATION】to enter the servo debugging interface; enter the parameter setting and display screen by the【⊞ SERVO PARAMETER】.

4) Move the cursor to the selected axis parameter **#0**; input the password 315 (Parameters **0~42** can be viewed or altered); download the drive to the system by input button; the servo parameter can be modified in the【SERVO PARAMETER】interface.

5) Move the cursor to the position where the parameter number is to be modified:

Method 1: Display the page to be set a parameter by page up/down button; alternatively, move the cursor by direction buttons to position the parameter position to be modified.

Mehtod 2: Positioning is performed by SEARCH after inputting a parameter number.

6) Confirm it by INPUT, the parameter value is delivered to drive, and the state column displays "Successful for drive parameter download!".

7) The servo saves the updated parameter by SAVE, and the state column displays "Successful for drive parameter save!".

8) Close the parameter switch after the overall parameters setting are executed.

3.3.5.1.2 Matched Parameter Setting Between Servo & Motor Type

1) Select the <MDI> mode.

2) Enter <SETTING> interface by SETTING, set the parameter switch to "1".

3) Firstly, press the SYSTEM, then enter to the servo debugging interface by【⊞ BUS CONFIGURATION】, lastly, enter the parameter display page by【⊞ SERVO PARAMETER】.

4) Move the cursor to the current selected axis parameter **#0**, input the password **385**, download the drive parameter to the system by input button, modify the servo parameter in the【SERVO PARAMETER】interface.

5) Move the cursor to the parameter **#1**, input the matched numerical value to the motor type:

6) Confirm it by INPUT, the parameter value is delivered to the drive, and the state column displays "Successful for drive parameter download!".

7) Servo saves the updated parameter by SAVE, and the state column displays "Successful for drive parameter save!".

8) Close the parameter switch after the overall parameters setting are completed.

3.3.5.1.3 Servo Parameter Backup

1) Select the <MDI> operation mode.

2) Enter the <SETTING> interface by SETTING, set the parameter switch to "1".

3) Enter the <SETTING> interface by SETTING, input **the terminal user password or above.**

4) Control the SYSTEM, enter the servo debugging interface by【⊞ BUS CONFIGURATION】, and then enter the parameter display screen by【⊞ SERVO PARAMETER】.

5) Select the【BACKUP】button, the parameter of the current selected axis is copied to the file DrvParXX.txt. (XX axis number. For example: the file name is DrvPar01.txt if the X axis is backup.)

6) Close the parameter switch after the overall parameters setting are completed.

3.3.5.1.4 Recovery of Servo Parameter

1) Select the <MDI> operation mode.

2) Enter the <SETTING> interface by SETTING, set the parameter switch to "1".

3) Enter the <SETTING> interface by SETTING, input **the terminal user password or above.**

4) Press the SYSTEM, enter the servo debugging interface by【⊞ BUS CONFIGURATION】, and then enter the parameter display screen by【⊞ SERVO PARAMETER】

5) Select the【RECOVERY】button, the backup file DrvParXX.txt of the current selected axis is recovered to the servo drive. (XX axis number. For example: the file name is DrvPar01.txt if the X axis is backup.)

7) Servo saves the updated parameter by SAVE, and the state column displays "Successful for drive parameter save!".

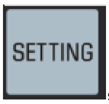
7) Close the parameter switch after the overall parameters setting are completed.

3.3.5.1.5 Servo Level Zering

The machine tool trembles due to the excessive big of the servo parameter rigid when debugging the parameter. To avoid the hazard, the servo parameter can be recovered into 0 level initial state parameter by the function of the servo level zeroing.

1) Select the <MDI> operation mode.

2) Enter the <SETTING> interface by SETTING, set the parameter switch to "1".

3) Enter the <SETTING> interface by 

, input **the terminal user password or above.**

4) Press the 

, enter the servo debugging interface by【⊞ BUS CONFIGURATION】, and then enter the parameter display screen by【⊞ SERVO PARAMETER】

5) The overall servo axis parameters can be recovered into the parameter of the 0 level by the button of【LEVEL ZEROING】.

7) Servo saves the updated parameter by 

, and the state column displays "Successful for drive parameter save!".

7) Close the parameter switch after the overall parameters setting are completed.

3.3.5.2 Spindle Parameter

When the spindle drive is selected as bus control mode by the system (Bit 1 of parameter **No.:1** sets to 1), user can check and set the corresponding servo drive parameters of the spindle in the【SPINDLE PARAMETER】interface.

3.3.5.2.1 Spindle Parameter Display

When the bit 4 of parameter No.: 0 is set to 0, the system uses the single spindle control, enter the spindle parameter interface by【SPINDLE PARAMETER】softkey. The content of this interface is shown below (Refer to the Fig. **3-3-5-2-1-1**).

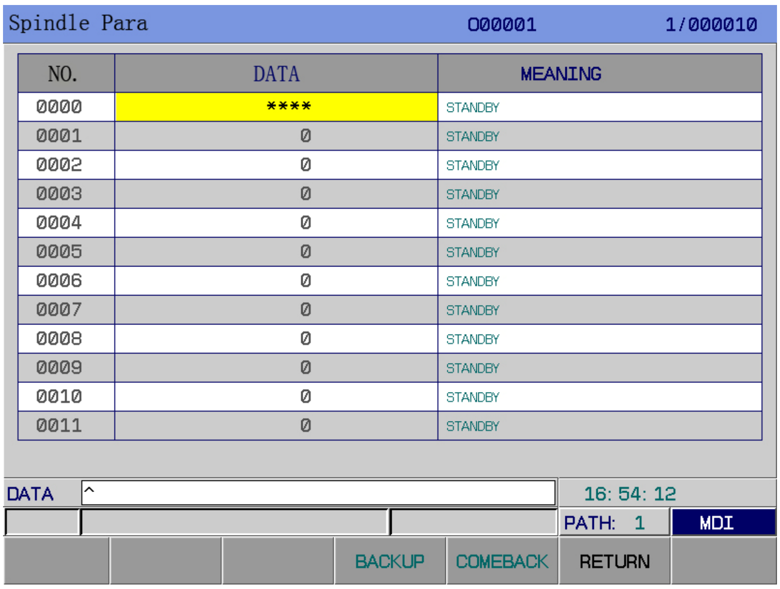


Fig. 3-3-5-2-1-1

VOLUME II OPERATION

When the bit 4 of parameter No.: 0 is set to 1, the system uses the double-spindle control, enter the spindle parameter interface by【SPINDLE PARAMETER】softkey. The content of this interface is shown below (Refer to the Fig. **3-3-5-2-1-2**).

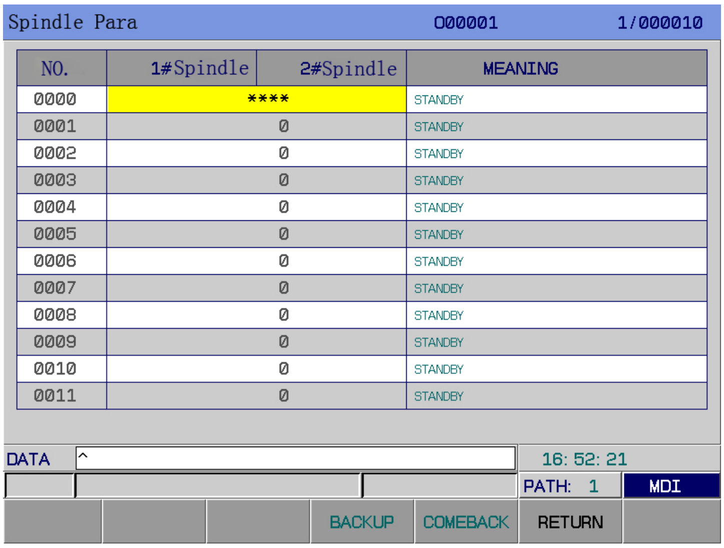
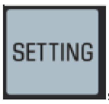


Fig. 3-3-5-2-1-2

3.3.5.2.2 Modification and Setting of Spindle Parameter

1) Select the <MDI> operation mode.

2) Enter the <SETTING> interface by 

, set the parameter switch to “1”.

4) Press the 

, and then enter the parameter display screen by【⊞ SPINDLE PARAMETER】softkey.

4) Move the cursor to the selection axis parameter **#0**, input the password 315 (Parameters **0**～**160** can be viewed and altered); download the drive parameter to the system by input button, and the servo parameter can be modified by【SPINDLE PARAMETER】interface.

5) Move the cursor to the position where to be modified the parameter number:

Method 1: Display the page to be set the parameter by page up/down; or move the cursor by direction buttons, and the position the parameter position where to be modified.

Method 2: Press the 

to position after inputting the parameter number.

6) Input the new parameter value by numerical buttons (Modify the parameters with different levels, and the password authority of the corresponding level should be input accordingly).

7) Confirm it by INPUT, the parameter value is then input, and the state column displays “Successful for drive parameter download!”.

8) Servo saves the updated parameters by SAVE, and the state column displays “Successful for drive parameter save!”.

9) Close the parameter switch after the overall parameters setting are completed.

3.3.5.2.3 Backup of Spindle Parameter

1) Select the <MDI> operation mode.

2) Enter the <SETTING> interface by SETTING, set the parameter switch to “1”.

3) Enter the <SETTING> interface by SETTING, input **the terminal user password or above.**

4) Press the SYSTEM, and then enter the parameter display screen by【⊞ SPINDLE PARAMETER】softkey.

5) Select the【BACKUP】button, the parameter of current selected axis can be copied to the file DrvParXX.txt. (XX axis number. For example: the file name is SP01.txt if the X axis is backup.)

6) Close the parameter switch after the overall parameters setting are completed.

3.3.5.2.4 Recovery of Spindle Parameter

1) Select the <MDI> operation mode.

2) Enter the <SETTING> interface by SETTING, set the parameter switch to “1”.

3) Enter the <SETTING> interface by SETTING, input **the terminal user password or above.**

4) Press the SYSTEM, and then enter the parameter display screen by【⊞ SPINDLE PARAMETER】softkey.。

5) Select the【RECOVERY】button, the backup parameter file SPXX.txt of current selected axis can be recovered to the servo drive. (XX axis number. For example: the file name is SP01.txt if the X axis is backup.)

6) Servo saves the updated parameter by 

, and the state column displays "Successful for drive parameter save!".

7) Close the parameter switch after the overall parameters setting are completed.

3.3.5.3 Servo Debugging Tool STT

In order to guarantee that the servo debugging function is really reacted the servo capacity, it is better to cancel the gear ratio at the drive side and each compensation (Including the pitch error compensation and reverse interval compensation) at the system side.

3.3.5.3.1 Interface Composition

Enter to the servo debugging tool interface by【⊞STT】. The content of this interface is as the Fig. **3-3-5-3-1~ 3-3-5-3-2).**

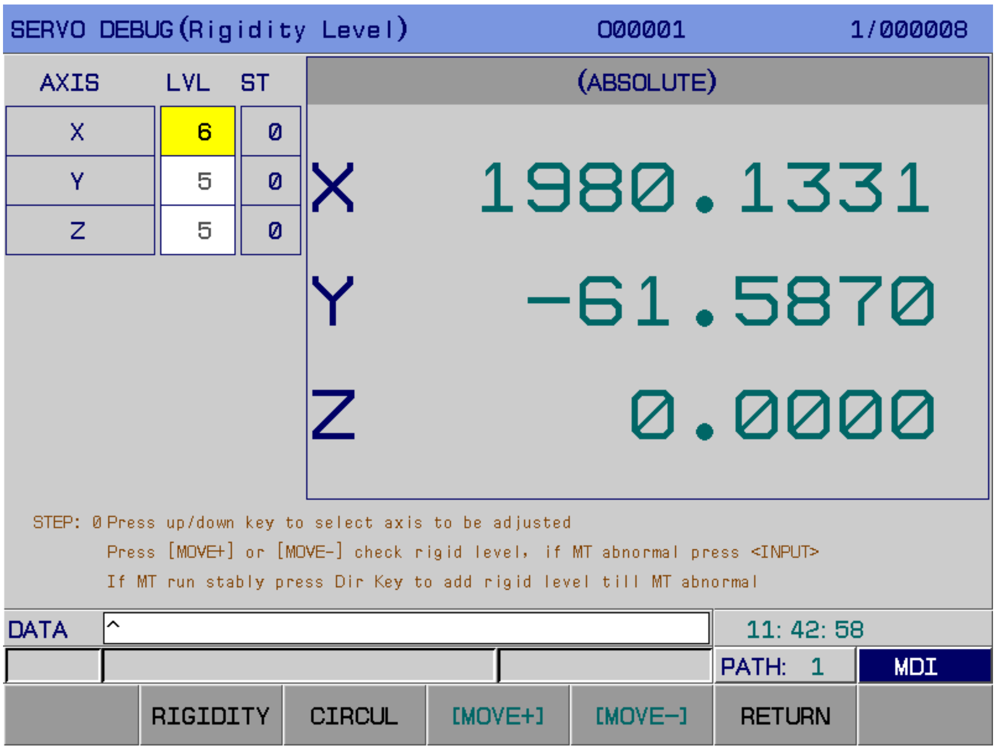


Fig. 3-3-5-3-1 Rigid level interface

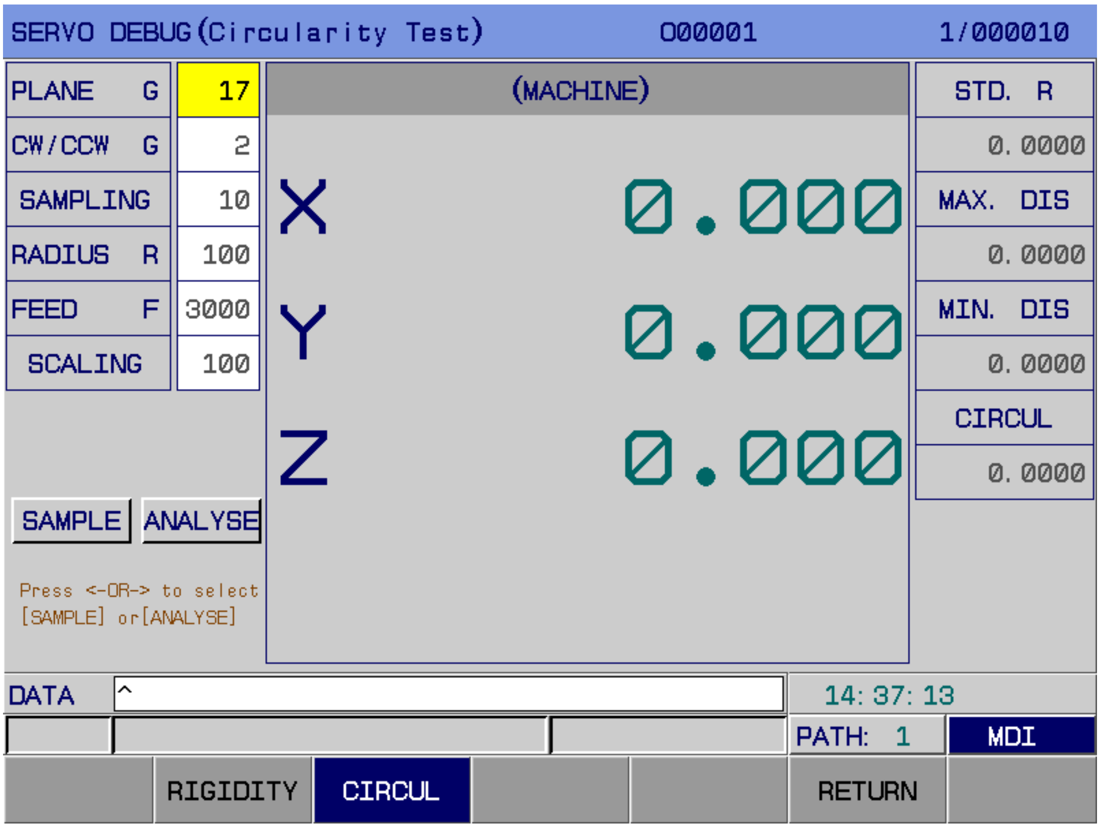


Fig. 3-3-5-3-2 Roundness measure interface

Note: The coordinate axis display on the servo debugging interface is determined by the system control axis numbers and the least number in the bus servo slave-station.

3.3.5.3.2 Function Introduction

1. Rigid grade and parameter optimization operation function

   This function is for setting the servo parameter to the optimization state of the servo performance.

2. Round testing

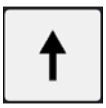
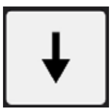
   The round testing can be judged the response synchronism of each servo axis for the machine tool by collecting the motor encoder position information based upon imitating the round of the circle cutting movement.

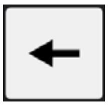
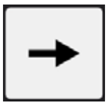
3.3.5.3.3 Operation Explanation

1. Rigid grade debugging operation

Explanation: the debugging and setting of rigid grad, only one axis can be operated once.

Operation buttons:

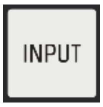
A.  **and**  buttons: Selection axis. (Note: The up/down direction buttons can not be changed the axis in the current operation once entering the optimization schedule.)

B.  **and**  **buttons:** Reduce or increase the rigid grade of the current axis, the rigid level reduces or increases one level by every one push;

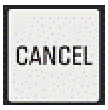
C. 【AXIS SHIFTING+】 and【AXIS SHIFTING -】 softkeys: Move the current axis for a distance positively or reversely based upon the velocity from the data parameter P393 of which this distance is determined by data parameter P392. Check whether the motor is vibrative or abnormal noisy by repeatedly controlling the **【AXIS SHIFTING+】** and **【AXIS SHIFTING -】** softkeys before entering the optimizing schedule; the motor's character can not be gained by pressing the **【AXIS SHIFTING+】** and **【AXIS SHIFTING -】** softkey movement axes consecutively once entering the optimizing schedule.

Note 1: Press the【AXIS SHIFTING+】and【AXIS SHIFTING -】softkey movement axes and collect the data after entering the optimizing schedule.

Note 2: Never attempt to change the data parameters P392 and P393 without the professionals; otherwise, the optimization may not succeed.

D. 

button: Confirm the operation or enter the next step;

E. 

button: Cancel some one operation or return to the previous operation;

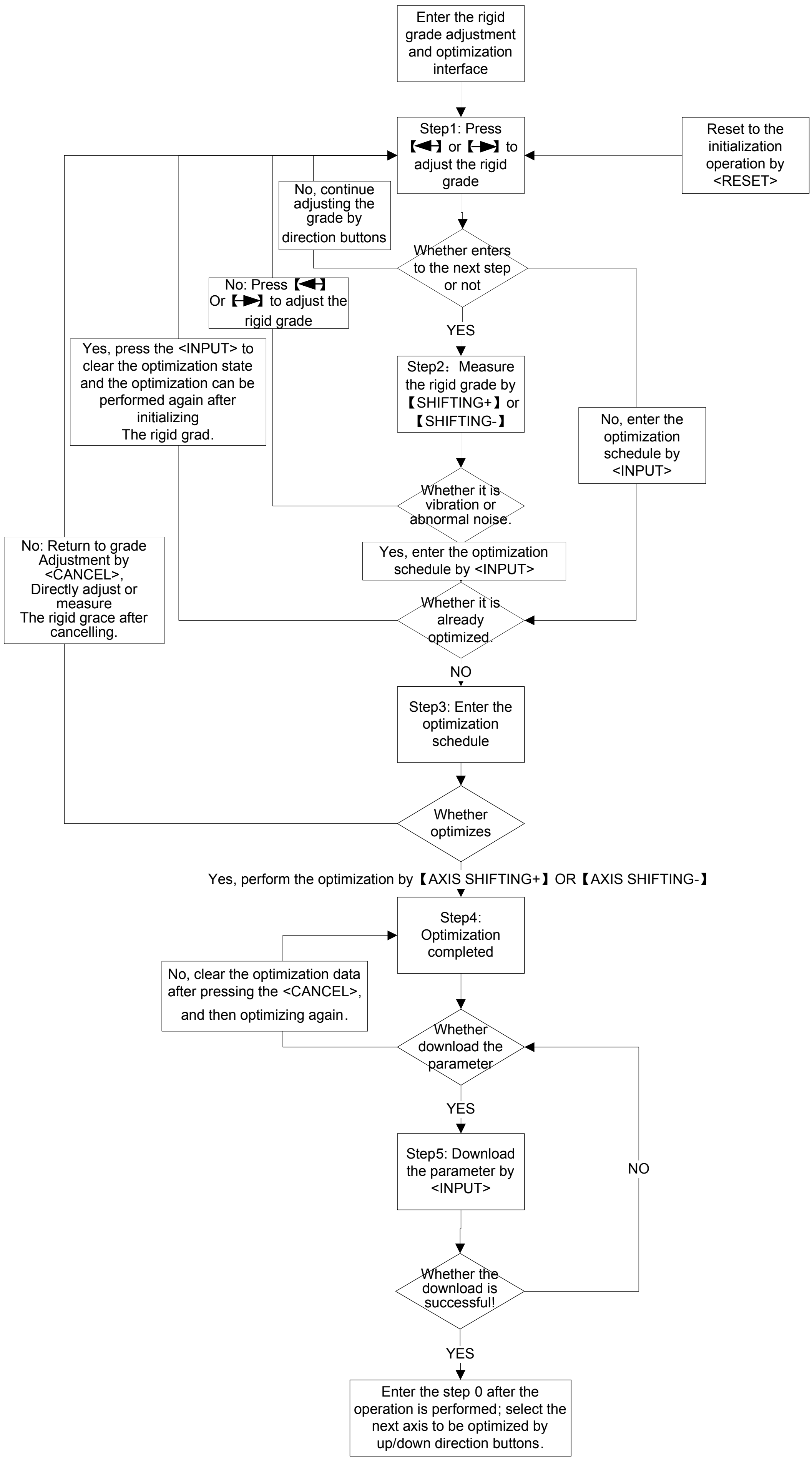
F. 

button: Reset operation, return to the initial operation step.

G. 

button: Save operation, save the optimized parameter.

Operation schedule: refer to the following figure:


Enter the rigid grade adjustment and optimization interface
Step1: Press 【◀】 or 【▶】 to adjust the rigid grade
Reset to the initialization operation by <RESET>
No, continue adjusting the grade by direction buttons
Whether enters to the next step or not
No: Press 【◀】 Or 【▶】 to adjust the rigid grade
YES
Yes, press the <INPUT> to clear the optimization state and the optimization can be performed again after initializing The rigid grad.
Step2：Measure the rigid grade by 【SHIFTING+】 or 【SHIFTING-】
No, enter the optimization schedule by <INPUT>
Whether it is vibration or abnormal noise.
Yes, enter the optimization schedule by <INPUT>
No: Return to grade Adjustment by <CANCEL>, Directly adjust or measure The rigid grace after cancelling.
Whether it is already optimized.
NO
Step3: Enter the optimization schedule
Whether optimizes
Yes, perform the optimization by【AXIS SHIFTING+】OR【AXIS SHIFTING-】
Step4: Optimization completed
No, clear the optimization data after pressing the <CANCEL>, and then optimizing again.
Whether download the parameter
YES
Step5: Download the parameter by <INPUT>
NO
Whether the download is successful!
YES
Enter the step 0 after the operation is performed; select the next axis to be optimized by up/down direction buttons.

2. Round testing

Operation buttons:

A. Number buttons: Input each parameter numerical value;

B. ↑ **and** ↓ **buttons: Parameter selection;**

C. ← **and** → **buttons:** Function selection (Collection and analysis);

D. INPUT **button:** Input parameter value or confirm it, and then perform the operation;

E. INPUT **button:** Clear the data and then reset to the initial state.

Parameter items:

A. Plane: Select the testing plane G17, G18 and G19;

B. CW/CCW circle: Select the circle direction G02, G03;

C. Sampling period: Set the sample period, it is necessary to set based upon the circle radius and feedrate; the bigger the radius is, the longer the sample period is; the slower the feedrate is, the longer the sampling period is;

D. Feedrate: The movement speed when testing.

E. Amplification: The round analysis is the magnification of the error amplification.

Operation steps:

Step 1: Select the collection function by ← or → button after each parameter is set;

Step 2: Start the arc and collect the data by pressing the INPUT, and then select the analysis function by ← or → after the collection is performed.

Step 3: Start the analysis function by INPUT, and then output the roundness data and draw the round error distribution figure; refer to the following drawing.

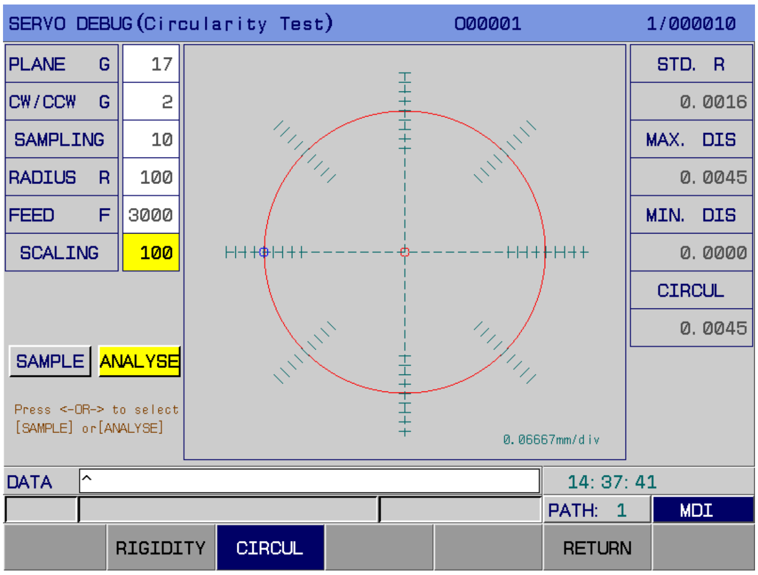


Fig. 3-3-5-3-3-1

Note: Measure the synchronism situation of the current each feed axis by the roundness testing tools after the rigid grade and parameter optimization function debugging are performed. The roundness testing of each plane is within 6u which is regarded as the synchronism of current each servo axis is better, the parameter debugging is successful accordingly.

3.3.5.4 Double-drive Debugging Tool

The double-drive offset generates during the operation of the system; adjust some one axis for the double-drive by the【DOUBLE-DRIVE DEBUGGING TOOL】; at last, adjust the double-drive parallel by observing whether the motor feedback current of the double-drive is consistent (or perform the parallelism).

3.3.5.4.1 Double-drive Function Setting

When using the【DOUBLE-DRIVE DEBUGGING TOOL】, the data parameter **P380** should be set to **1~3** (1: The 4th axis is synchronic with the X axis; 2: The 4th axis is synchronic with the Y axis; 3: The 4th axis is synchronic with the Z axis); also, the **bit 0 of parameter No.:0** should be set to 1, which is treated as the bus transition method of the drive, and it is necessary to debug in the MPG mode.

3.3.5.4.2 Enter to Double-drive Debugging Function

In the double-drive debugging tool interface, after the interface is selected by the 

button on the controllable panel, enter to the parameter page display by【⊞ BUS CONFIGURATION】softkey,

VOLUME II OPERATION

and then enter the double-drive debugging tool interface after selecting the【DOUBLE-DRIVE TOOL】again. Refer to the Fig. **3-3-5-4-2-1.**

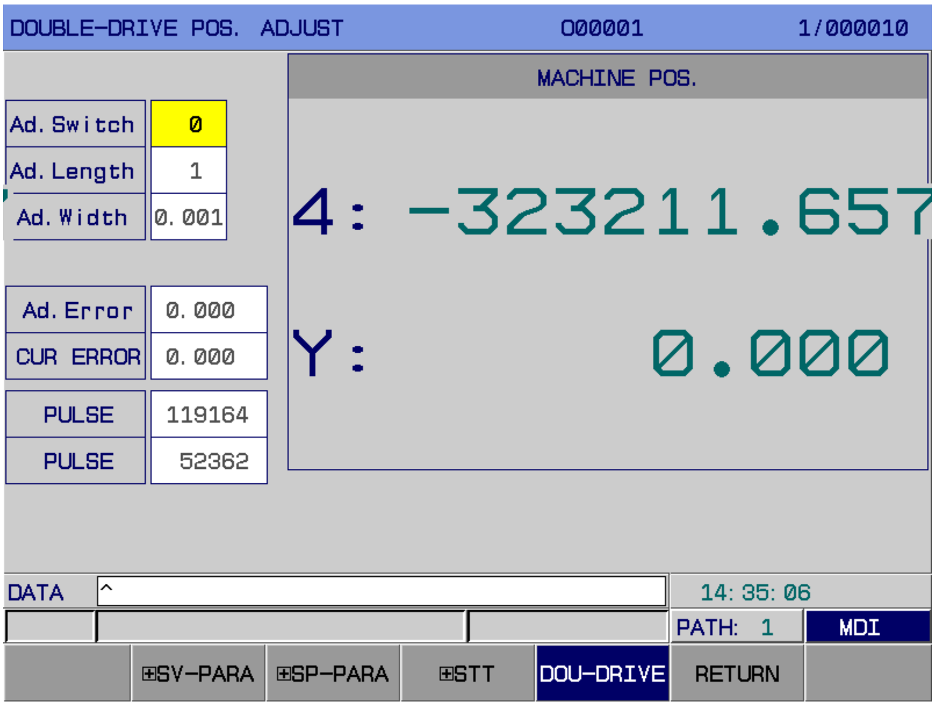


Fig. 3-3-5-4-2-1

3.3.5.4.3 Operation Explanation

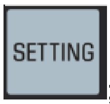
If the parallelism verification should be performed by double-drive tool, the following steps are necessary:

1. Shift to the MPG mode by controllable panel;
2. The step length of the MPG is adjusted to 0.001;
3. Set the【CALIBRATION SWITCH】to 1, in this case, the synchronism axis does not performed at the same; each axis can be adjusted by MPG movement;
4. The MPG movement can be performed based upon the【CURRENT OFFSET】, when the MPG movement exceeds 1mm of 【VERIFICATION LENGTH】, the system may alarm, it may be moved again after the resetting cancels an alarm.
5. Set the【VERIFICATION SWITCH】to 0, in this case, the double-drive synchronism is enabled; judge the motor's current feedback (Alternatively, open the parallelism) is parallel by moving the double-drive manually. Repeated the above steps 1~4 till the verification is performed.

3.4 Setting Display

3.4.1 Setting Page

1. Enter the page

Enter the setting information display interface by 

; there are 5 sub-interface in this interface:【SETTING】,【⊞ WORKPIECE COORDINATE】,【⊞ CENTER TOOL-SETTING】,【⊞ DATA】and【PASSWORD】, check and modification can be performed by the corresponding software, refer to the Fig. **3-4-1-1:**

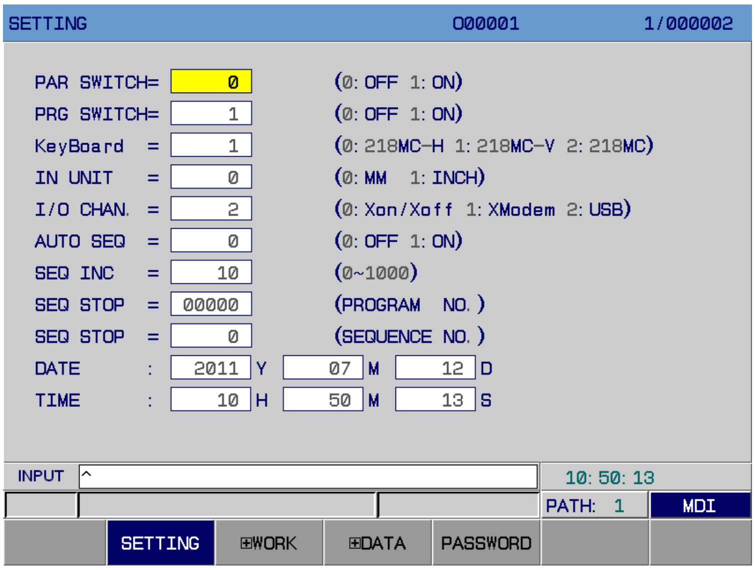


Fig. 3-4-1-1

2. 【SETTING】interface operation explanation

Press the【SETTING】softkey, check its parameters after the interface is set in the Fig. 3-4-1-1, as well, the corresponding parameter can be modified.

The operation methods and steps are shown below:

(a) Enter the <MDI> operation method;

(b) Move the cursor by up/down button to the item to be changed;

(c) Input 0 or 1 according to the following explanations, or modify it by the left or right key;

1) Parameter switch

0: Close the parameter swith

1: Open the parameter switch

When the parameter switch sets to "0", the modification and setting of the system parameter are prohibited, the system alarm (0100: Parameter write is enabled) cancels. When the parameter switch sets to "1", the system alarm occurs (0100: Parameter write is enabled). In this case, the alarm (This operation is only enabled in a setting interface) can be cancelled by SHIFT + RESET buttons.

2) Program switch

0: Close the program switch

1: Open the program switch

When the program switch sets to “0”, the program editing is prohibited.

3) Keyboard selection

0：MC-H 1：MC-V 2：MC 3：MC-U1

Note: In any method, the keyboard selection can be also altered if the ESP button is controlled.

4) Input unit

Set whether the input unit of the program is metric or inch

0: Metric

1: Inch

5) I/O channel

User freely sets it based upon its requirements, for example, the channel sets to 2 if the DNC machining is performed by U disk; the channel sets to 3 if DNC machining is performed by internet access.

0, 1: RS232 (0 selects the Xon/Xoff agreement, 1 selects Xmodem agreement)

2：USB 3：NET

6) Automatic series No.

0: When the program is input by keyboard in the editing mode, the system will not automatically insert the sequence number.

1: When the program is input by keyboard in the editing mode, the system will automatically insert the sequence number. The incremental values of the sequence numbers among each block are set by data parameter P210.

7) Sequence number increment

The incremental value when the automatic sequence number insertion is set, its range is 0~1000.

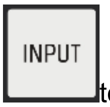
8) Stop sequence number

This function can be set some one program that performs the single block dwell to the specified block, which can be enabled when the program number and the block sequence number are specified at the same time. For example: 00060 (Program number) is program number O00060; 00100 (Sequence number) is block number N00100.

Note: When the stop sequence number is automatically set to -1 after the program operates to the object block, and the single block dwell will not perform.

9) Date and time

User can set the system data and time at this position.

(d) Control the 

to input.

3.4.2 Workpiece Coordinate Setting Page

1. Enter the coordinate system setting interface by【⊞ WORKPIECE COORDINATE】, the content of this page is shown below (Fig. **3-4-2-1**).

Fig. 3-4-2-1

Additionally, there are 50 addition workpiece coordinate systems can be used other than 6 standard workpiece coordinate systems (G54～ G59). Refer to the Fig. **3-4-2-2.** Each coordinate system can be checked or altered by page up/down button. The operation of the additional coordinate system, refer to the Section 4.2.9 *Additional Workpiece Coordinate System* in PROGRAMMING.

Fig. 3-4-2-2

VOLUME II OPERATION

2. There are two methods of coordinate input:

1) After entering this interface in any mode, move the cursor to the coordinate system to be changed, and press the axis name from the desired setting value, and then confirm it by INPUT; the value of the current machine coordinate system sets to the origin of the G coordinate system, for example: press "X", then INPUT or "X0", and then INPUT again. The system may then automatically input the X axis machine coordinate of this point; in addition, for example, input X10(X-10), then press the INPUT, it means the X machine tool coordinate value is +10(-10).

2) After entering this interface in any mode, move the cursor to the coordinate system axis to be changed, directly input the machine tool coordinate value of the origin of the workpiece coordinate system, and then confirm it by INPUT.

3) After entering this interface in any mode, move the cursor to the coordinate system axis to be changed, directly input the machine tool coordinate value of the origin of the workpiece coordinate system, and then confirm it by <INPUT>. In addition, press the softkey <+INPUT> after the coordinate value is input, the system may automatically calculate the new coordinate value and then display it.

3. The method of coordinate system search:

1) In any mode, press SEARCH to position after the coordinate system is input. For example: input "G56".

2) In any mode, for example, input "P6" or "P06", press the SEARCH, the cursor will position at the additional workpiece coordinate system "G54 P06".

3.4.3 Center and Tool-setting Function

Enter the center and tool-setting function by【⊞ CENTER TOOL-SETTING】, the content of this interface displays in the following figure (Refer to the Fig. **3-4-3-1**).

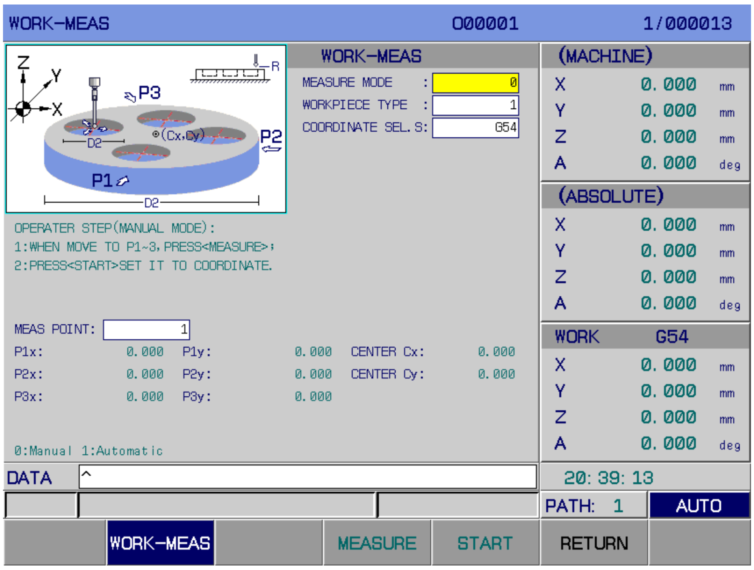


Fig. **3-4-3-1**

3.4.3.1 Center Function Introduction & Operation Explanation

Center measure: There are two center measurements: Manual and Auto. Wherein, only the hole or external circle, boss or groove can be performed the center measurement by the Manual mode; the hole or external circle, boos or groove, vector hole or external, vector boss or groove can be executed center measurement by Auto mode.

I. Manual Center

- **Interface display**

A. Hole or external circle

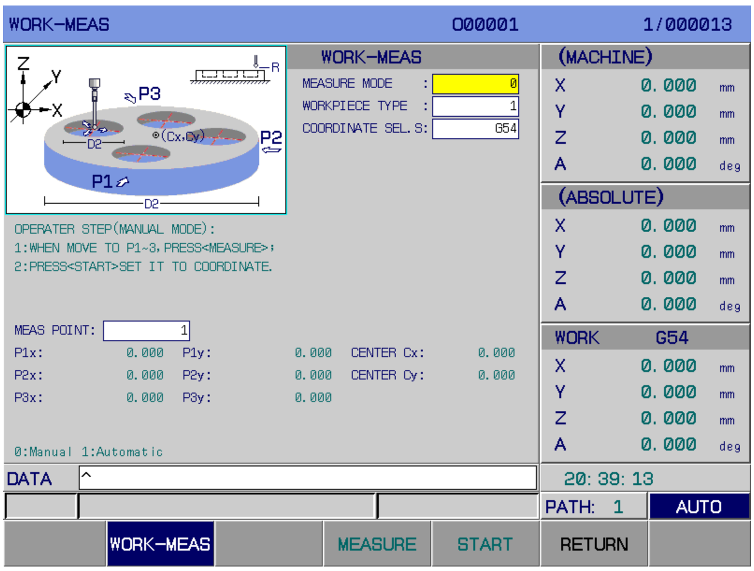


Fig. 3-4-3-1-1

B. Boss or groove

VOLUME II OPERATION

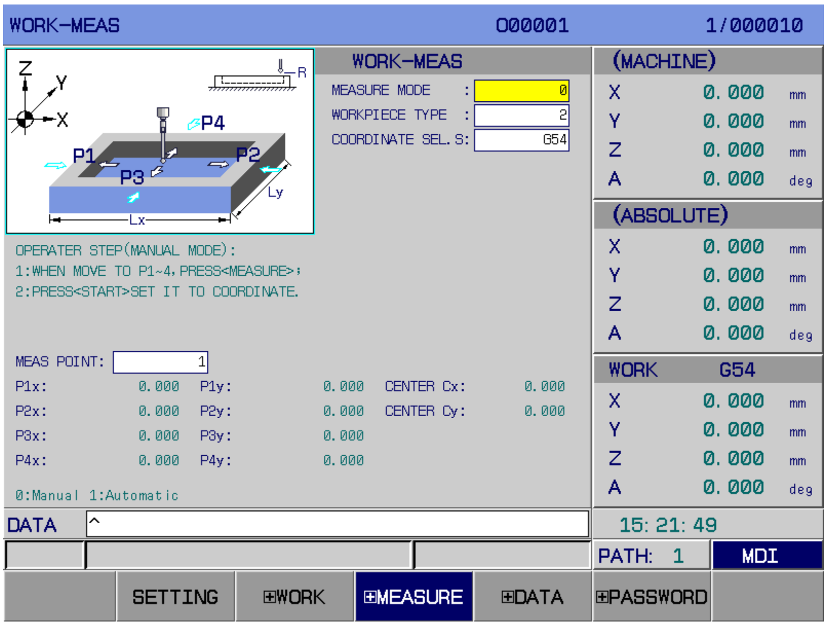


Fig. 3-4-3-1-2

◆ **Manual center operations:**

A. Option explanation:

1. Measurement method:

0: Manual 1: Auto

2. Workpiece type:

1: Hole or external circle 2: Boss or groove

3. Coordinate system selection S:

G54~G59 G54 P1~P50 Set the center to the desired coordinate system after the measurement is completed.

4. Measure point:

A. When the workpiece type is hole or external circle: there are 3 measurement points (P1~P3), and the measurement will perform without particular order; wherein, if there are 3 points are overlapped, one of the point is regarded as the center coordinate; if there are 3 points are shared with a same straight line, the center coordinate can not be calculated, and it is necessary to measure one of the point or the overall points again;

B. When the workpiece type is boss or groove: there are 4 measurement points (P1~P4), and the measurement will perform without particular order; wherein, P1 and P2 are separately two points along the X axis direction; P3 and P4 are separately two points along the Y axis direction. The center coordinate of X axis direction calculates of the X coordinate of P1 and P2; the center coordinate calculates of the Y coordinate of P3 and P4.

B. Operation steps:

Step 1: Press the <MEASURE> softkey after manually moving the tool or center bar to the 1st measurement point.

Step 2: Repeated the step 1 till the overall measurement points are completed (There are 3 points for circle, and 4 points for rectangle).

Step 3: Set the center coordinate to the selected coordinate system by <START> softkey.

II. Auto center

◆ Interface display and parameter option explanation

A. Public parameter option

1. Measurement method:

0: Manual 1: Auto

2. Workpiece type:

±1: Hole & external circle ±2: Boss & groove ±3: Vector hole & external circle ±4: Vector boss & groove

【Note】-1: hole +1: external circle -2: groove +2: Boss -3: vector hole +3: vector external circle -4: vector groove +4: vector boss.

3. Coordinate system selection S:

G54~G59 G54 P1~P50

4. Tool offset number T:

Tool offset number. The radius compensation value of the tool when the interpolation machining is stored within the tool offset number.

5. Experience value tool offset number E:

The tool offset number is already registered the experience value. E and T will not assign the same value when programming.

6. Rough center coordinate Cx:

The absolute coordinate value of the rouch center of the workpiece along X axis. If the current point is set to rough center, directly press <INPUT> key to input a null value.

7. Rough center coordinate Cy:

The absolute coordinate value of the rouch center of the workpiece along Y axis. If the current point is set to rough center, directly press <INPUT> key to input a null value.

8. Measure point coordinate Z:

The absolute position of the Z axis during measuring. If the current point is the measurement point, directly press <INPUT> key to input a null value.

9. Profile dimension tolerance H:

The measured profile dimension tolerance value.

10. Radial interval R:

When measuring the external profile, the probe head is the distance from the destination surface before the Z axis moves. The default is 8mm(0.3149inch) when the power is turned on.

11. Measure head overtravel distance Q:

The overtravel value of the measure head. Input a value by programming, the measure head that uses this value to find a surface is regarded as the extra distance by exceeding the destination dimension. If there is no programming, the default value is 10.0 mm (0.394 inch).

B. Hole & external circle parameter

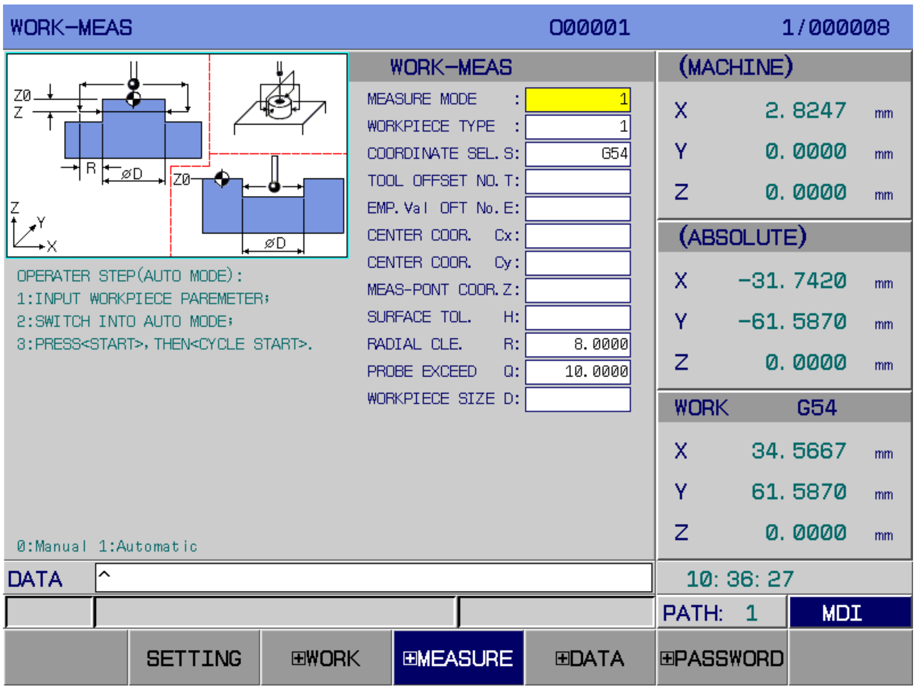


Fig. 3-4-3-1-3

1. Destination dimension D:

The Hole or diameter of the external circle to be measured. This value can not be a null or 0.

C. Boss & groove parameter

VOLUME II OPERATION

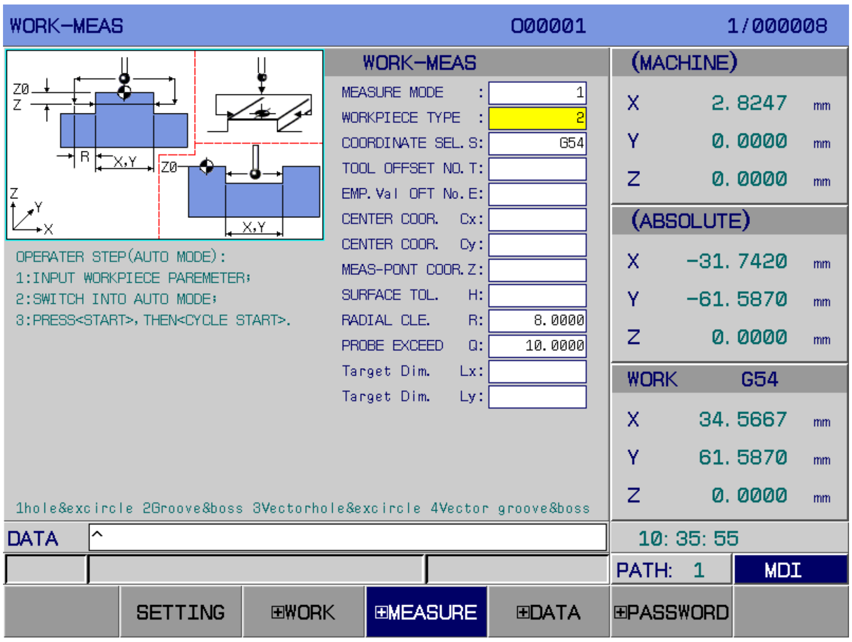


Fig. 3-4-3-1-4

1. Destination dimension Lx:

The profile dimension along X axis direction to be measure. The measurement along this axis can not be performed when this parameter option is null or 0.

2. Destination dimension Ly:

The profile dimension along Y axis direction to be measure. The measurement along this axis can not be performed when this parameter option is null or 0.

【Note】: Lx and Ly can not be set to null or 0 at the same time.

D. Vector hole & external circle

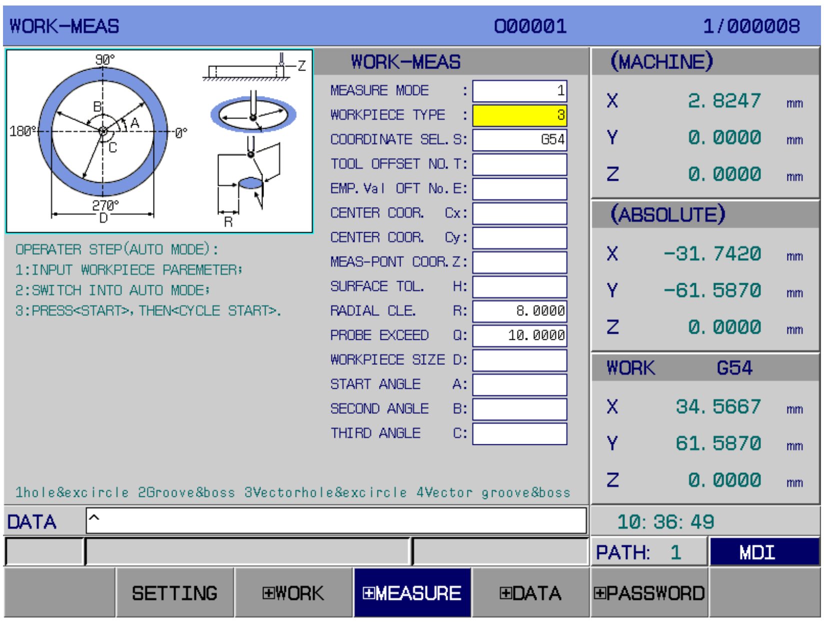


Fig. 3-4-3-1-5

1. Destination dimension D:

The diameter of the hole or external circle to be measured. This value can not be set to null or 0.

2. Start angle A:

The angle by the 1st vector measure, calculate from the X+ direction. If it is ignored, the alarm issues.

3. The 2nd angle B:

The angle of the 2nd vector measure, calculate from the X+ direction. If it is ignored, the alarm issues.

4. The 3rd angle C:

The angle of the 3rd vector measure, calculate from the X+ direction. If it is ignored, the alarm issues.

【Note】The least D-value between any two point angle is determined by “#5=____” in program O09729, the default is 5. If the least D-value should be changed, modify the “#5=____”.

E. Vector boss & groove

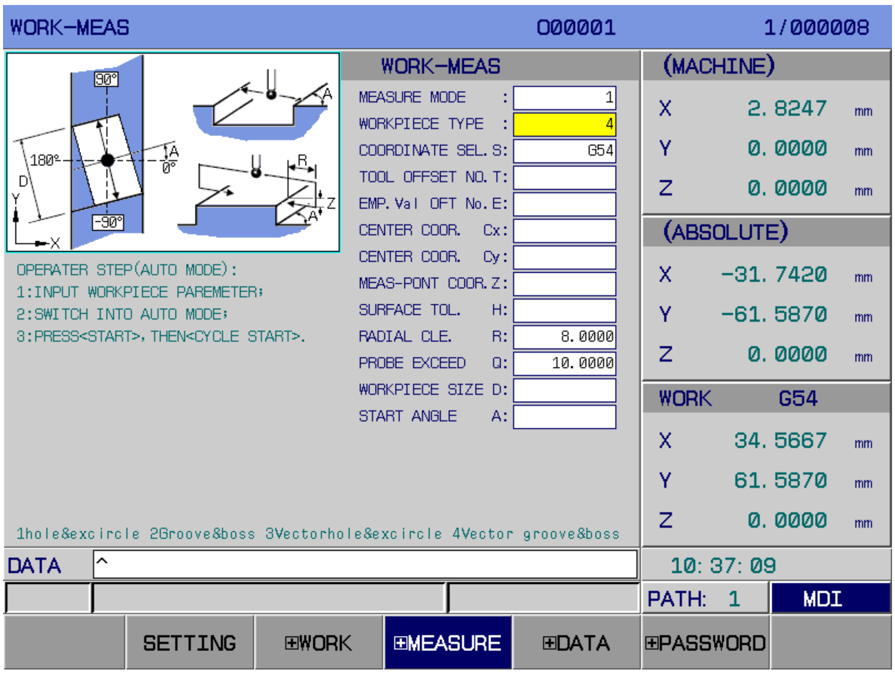


Fig. 3-4-3-1-6

1. Destination dimension D:

The profile diemension to be measured. The measurement of this axial will not perform when the parameter is null or 0.

2. Start angle A:

Measured plane calculates the angle from the X+ direction.

◆ **Data input**

A. The condition of the data input

When the Auto center measurement does not start, the data can be input in any operation method.

B. Input format

1. Data +<INPUT> inputs the desired data;
2. Directly control the <INPUT> button to enter the null value;
3. When the current line is the rough center coordinate X, rough center coordinate Y, measurement point coordinate Z, the following formats can be input:

① Directly input null value by <INPUT>;

② X/Y/Z+ <INPUT> input the current absolute coordinate value;

③ X/Y/Z+ data + <INPUT> input the current absolute coordinate value of the selected axis + data.

④ Directly press the [MEASURE] software to input the absolute coordinate value of current axis;

⑤ X/Y/Z+ [MEASURE] input the absolute coordinate value of current axis;

⑥ X/Y/Z+ data + [MEASUREMENT] input the current absolute coordinate value of the selected axis + data;

◆ **Operation steps:**

Step 1: Set the center parameters of each item by turn;

Step 2: Shift to the Auto Mode.

Step 3: Start the automatic center program by <START>softkey, then measure the macro program by <CYCLE START> button; the system will automatically set the center coordinate to the selected workpiece coordinate system after the measurement is performed.

3.4.3.2 Introduction and Operation Explanation of Tool-setting Function

◆ **Interface display and function introduction**

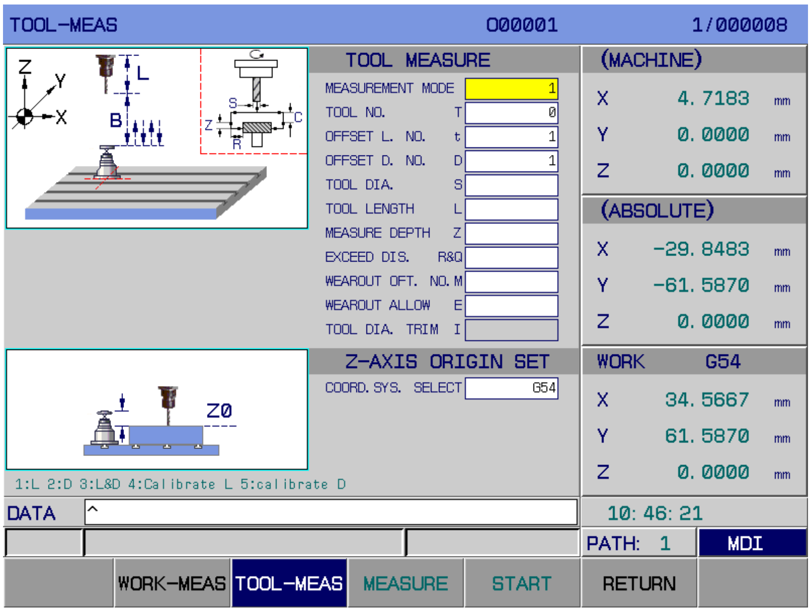


Fig. 3-4-3-2-1

Tool-setting function contains of the Auto tool length measurement and Z axis workpiece origin setting.

A. Tool measurement:

Auto tool length measurement is performed the measurement to the different tool length and dirameter by the tool-setter which is installed on the workpiece. The length and diameter of each tool is automatically set to the specified tool offset register, so that the tool can be correctly machined even if the different length and diameter are used at the same program.

B. The setting of the coordinate origin along the Z axis:

After the tool length measurement is completed, move the tool to the workpiece surface, in this case, in this case, the current machine coordinate value that is regarded as the origin sets to the selected workpiece coordinate (G54~G59 G54 P1~P50) by pressing the <MEASUREMENT> softkey.

◆ **Tool measurement**

A. Parameter option explanation

1. Measurement mode selection:

1: Length 2: Diameter 3: Length & Diameter calibration 4: Length calibration

2. Tool number T:

The current tool number to be measured.

3. Tool length offset number H:

Store the offset number of the current tool length (It's default is same with the T).

4. Cutter diameter offset number D:

Store the offset number of the current tool diameter (It's default is same with the T).

5. Cutter diameter S:

The diameter for the measured tool, when S is "+" value that means the tool is the right-handed direction cutting; when S is "-" value that means the tool is the left-handed direction cutting. When the cutter radius offset No. D in the register is already treated as the nominal tool diameter, the value may not be input. (This parameter value is cleared after the tool No. T is altered.)

6. Tool length estimated measurement L:

The tool length is being measured. When the tool length offset No. H in the registered is already regarded as the nominal tool length, the value may not be input. (This parameter value is cleared after the tool No. T is altered.)

【Note 1】: When the measurement mode is selected as the length carlibration, this length should be input and regarded as the standard tool (Refer to the mandrel) length.

7. Measurement depth Z:

The depth from the prober surface to the diameter measurement position (Default value –5.0mm [–0.20 inch]), negative value means the downward.

8 Overtravel value R&Q:

The overtravel value and the radial interval moving to the prober side downward. (Default value 4.0 mm [0.16 inch]).

【Note 2】It is the overtravel value along the length direction when the length measurement is performed; it is the radial overtravel value in the diameter measurement; and the overtravel value along the length direction and the radial overtravel value are same when the length & diameter measurement are executed.

9. Worn mark tool offset number:

A null tool offset number is used as the position of the tool wore mark.

10. Wore allowrance tolerance I:

Cutter dimension adjustment compensates the cutting state of the tool. The positive value brings that the actual radius is less than the specified value, for example I=.01 means that the tool radius is less than 0.01. Also, set the nominal cutter radius value to 0 by input its value.

【Note 3】: It is used for setting the tool-setting prober diameter when the diameter calibration is executed.

B. Measurement parameter inputs the operation steps:

1. The selection of item: it can be selected by moving the cursopr up/down button.

2. Data input: Whn the Auto tool measurement does not start, in any operation mode, input the data, then the ENTER, and the each data can be modified accordingly.

C. Operation steps:

Step 1: Set each tool measurement parameter in turn

Step 2: Shift to the Auto mode.

Step 3: Start the automatic tool-setting main program by <START> softkey, then operate the macro program measurement by <CYCLE START>again; automatically specifiy the tool length and radius write-in to the offset registered.

◆ **Workpiece origin setting along Z axis**

Notice: Ensure that the current tool is already performed the automatic tool measurement before the Z axis workpiece origin is set; otherwise, the machining error may occur, and the tool and equipment may be damaged, as well the personnel safety.

A. The selection of the coordinate system:

1. Setting range: G54~G59 G54 P1~P50

2. Data input: When the automatic tool measurement does not start, in any operation mode, move the cursor to the coordinate system option, and then input the data based upon the following formats:

a. The integer of 54~59

b. G54~G59；

c. P1~P50. Then press the**<INPUT>** button。

B. The setting of the workpiece origin:

1. Setting range: -9999.999~9999.999

2. Data input: When the automatic tool length measurement does not start, in any operation mode, move the cursor to the coordinate system selection option, set the current Z axis moachine coordinate value to the Z axis of the current selected workpiece coordinate system by directly pressing the [MEASUREMENT] software, alternatively, input the data based upon the following formats:

a. Input format: Z;

b. Z+ data; Set the current machine coordinate value along Z axis + the input data to the Z axis of the current selected workpiece coordinate system by [MEASUREMENT] softkey accordingly.

3.4.4 Backup, Recovery and Transmission of Data

Enter the setting (Data treatement) interface by【⊞ DATA】. The user data (Ladder diagram, ladder diagram parameter, system parameter value, tool compensation value, pitch compensation value, system macro variable, user macro program and CNC component program) can perform both the backup (save) and recovery (read) and the data output and input operations by U disk or PC machine. Simultaneously, the data back and recovery are performed, the component program memoried in the CNC will be regardless.

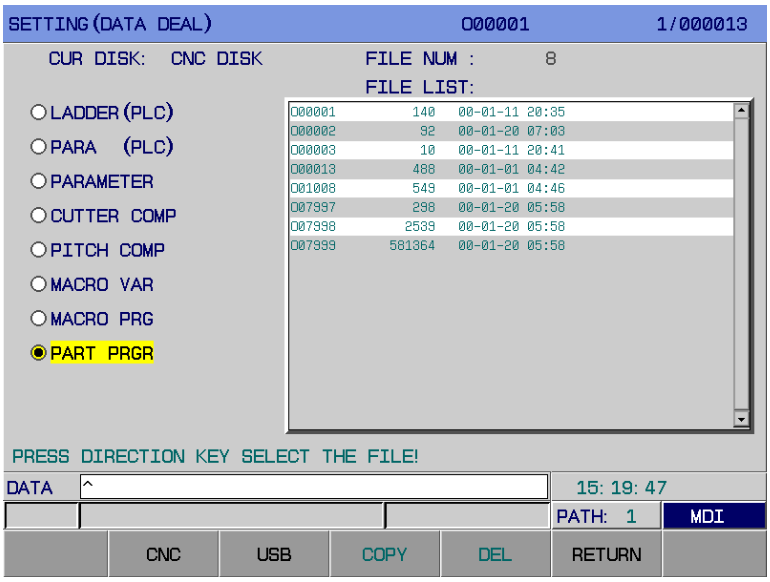


Fig. 3-4-4-1

Operation method:

1. Set the corresponding level password in the password interface by【PASSWORD】softkey. Refer to the Setting and Modification of Password Authority in the Section 3.4.5 for the corresponding password level of each data operation.

2. Enter the data treatment operation interface by【⊞ DATA】twice, refer to the Fig. **3-4-4-2.**

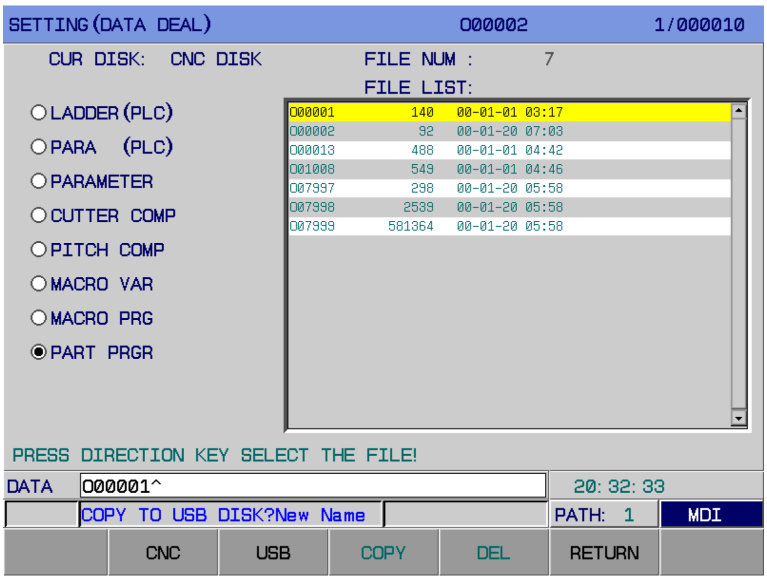


Fig. 3-4-4-2

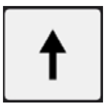
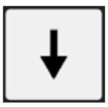
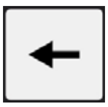
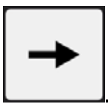
Enter the next page 

by 【▶】.

The function of each operation items are shown below: (Table **3-4-4-1**)

Table 3-4-4-1

Operation Item	Function Explanation
Data backup	Separatly perform the data backup, such as the ladder diagram (PLC), parameter (PLC), system parameter value, cutter compensation value, pitch compensation value and system macro variable etc. in the system disk. The system will generate .bak backup file after the data backup is executed.
Data recovery	Separatly perform the data recovery, such as the ladder diagram (PLC), parameter (PLC), system parameter value, cutter compensation value, pitch compensation value and system macro variable etc. in the system disk. The system recovery operation is read and recovered the backup file from system.
Data output	The data output operation can be output the data in the system disc to the external storage device.
Data input	The data input operation can be input the data in the external storage device to the system disc.
One-Touch backup	Simultaneously backup the multiplication data to the system disk.
One-Touch recovery	Simultaneously recover th+e backup file for the multiplication data items
One-Touch output	Simutaneously copy the file for the multiplication data items in the system disk to the U disk.
One-Touch input	Simultaneously copy the multiplication file in the U disk to the corresponding data item in the system disk.

3. Select the destination file by  or  direction button; Shift the data item list and file list table by  or .

4. The operations, such as the data backup, data recovery, data output, data input, one-touch backup, one-touch recovery, one-touch output and one-touch input, are performed by its corresponding softkey.

Note:

1) The softkey functions both the data output and the data input are consistent when I/O channel sets to

U disk.

2) For the output/input operation of the data, guarantee the the correctly set of the I/O channel. The I/O channel should be set to 0 when using U disk; I/O channel should be set to 0 or 1 when using the transmission software by PC machine.

3) The content of one-touch operation is determined by password authority; refer to the Section 3.4.5 of OPERATION EXPLANATION for the corresponding relationships between the each data item and password authority.

4) Relevant parameter:

① It is set by the bit 7 of parameter No.: 54: Whether the one-touch input/output

② It is set by bit 0 of parameter No.:27: Whether forbid the subprogram editing of program numbers 80000-89999.

③ It is set by bit 4 of parameter No.:27: Whether forbid the subprogram editing of program numbers 90000-99999.

5) During the treatment of the data, the system sets a relevant opration prompt, its prompt content shows below: (Table 3-4-4-2).

Table 3-4-4-2

Series No.	Prompt Information	Reason	Troubleshooting
1	One-touch operation completion	Successful operation	Transmission completion
2	One-touch operation completion, system prompts: The copy can be performed after the parameter is altered.	The input/output operation of the macro program is performed, however the relevant parameter of the system does not set.	Skip the input/output operation of this file.
3	The system alarm after one-touch operation is completed: The parameter that should be cut off the primary power-source is modified.	The ladder diagram and the parameter update of the ladder diagram are performed, it is better to power on again.	The transmission is executed, the power should be turned on again.
4	Fail to read the file	File error	Interrupt input/output operation
5	Fail to write the file	File error	Interrupt input/output operation
6	Fail to copy the file	File error	Interrupt input/output operation
7	File excessive big, it is better to use DNC	Component program is more than 4M	Interrupt input/output operation
8	The remainder space is absent	Adequate space	Interrupt input/output operation

6) The LADCHI**.TXT file is disabled after transiting the system, which can be enabled after the power is turned on again.

3.4.5 Setting & Modification of Password Authority

GSK218MC system provides authority setting function to avoid the machining program or CNC parameter is maliciously modified; password divides into 5 levels: the 1st level (System factory), the 2nd level (Machine tool factory), the 3rd level (System debugging level), the 4th level (Terminal user) and the 5th level (Machining operation) based upon the high-to-low. The default is lowest leve when the machine's power is turned on (Refer to the Fig. **3-4-5-1).**

The 1st level, 2nd level: The state parameter, data parameter, cutter compensation parameter and the transmission PLC ladder diagram, etc. of the CNC can be modified.

The 3rd level: The state parameter, data parameter and cutter compensation data and pitch compensation, etc. of the CNC can be modified.

The 4th level: The state parameter, data parameter of partial CNC can be modified.

The 5th level: Without password level. It can be both modified the cutter compensation data, macro program and performed the machine tool operation panel, instead of altering the state parameter, data parameter and pitch compensation data of the CNC.

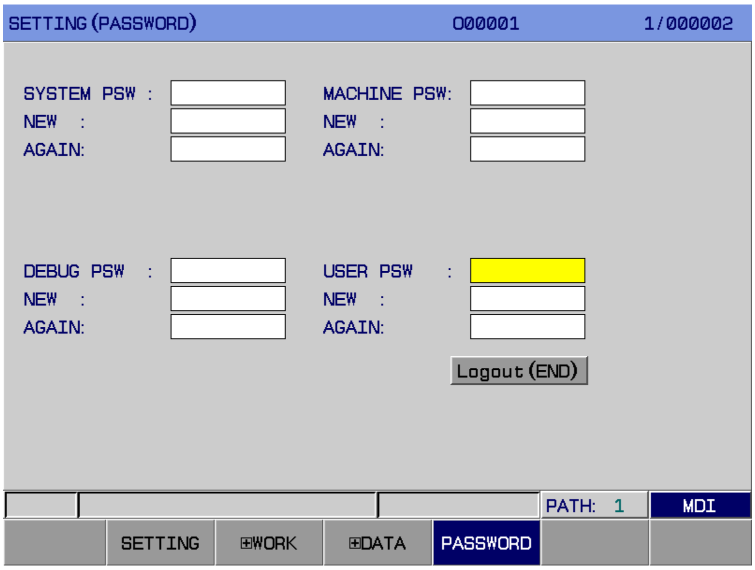
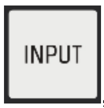
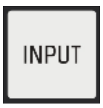


Fig. 3-4-5-1

1) After entering this interface by <MDI MODE>, position to the destination position by moving the cursor.

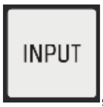
2) Input the password of the corresponding level, then press the 

, if it corrects, the system will prompt "Password Correct".

3) Input the digits 0~6 or letters when the system passwords are modified, then press the 

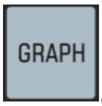
to confirm it.

4) Press the 

key after altering, cursor moves to the “CANCEL (END)” button, the interface prompts: “Press <INPUT> button to confirm the cancel!”. After controlling the 

, the interface prompts: “Cancel completion!”, simultaneously, cursor returns to the password setting column. The password will be automatically cancelled after the power is turned off till to the reatart is performed.

3.5 Figure Display

Enter the figure page by 

. There are two display interfaces【**FIGURE PARAMETER**】and【**⊞FIGURE**】, which can be displayed by its corresponding softkey. Refer to the following figure (Fig. **3-5-1**):

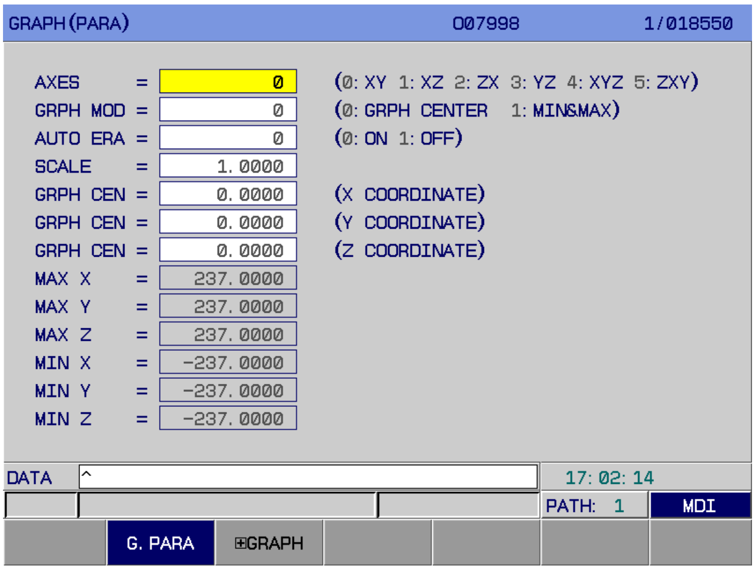


Fig. 3-5-1

1) Figure parameter interface Enter the figure interface by【FIGURE PARAMETER】, refer to the Fig. 3-5-1.

A. The meaning of the figure parameter:

Coordinate selection: Set the drawing plane, there are 6 methods (0~5), for example, the 2nd line shows.

Figure mode: Set the figure display mode.

Auto erasion: When it is set to 1, the program figure is automatically erased at the next circular start performed till to the program is ended.

Scaling: Set the proportion of the drawing.

Figure center: Set the corresponding workpiece coordinate value of the LCD based upon

VOLUME II OPERATION

the workpiece coordinate.

The Max./Min. value: After the Max./Min. value of the display axis is set, the CNC system may automatically set the scaling proportion or figure center value.

The Max. value of X: The Max. value along X in figure display (Unit: 0.0001mm / 0.0001inch)

The Min. value of Y: The MIN. value along Y in figure display (Unit: 0.0001mm / 0.0001inch)

The Max. value of Y: The Max. value along Y in figure display (Unit: 0.0001mm / 0.0001inch)

The Min. value of Y: The Min. value along Y in figure display (Unit: 0.0001mm / 0.0001inch)

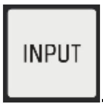
The Max. value of Z: The Max. value along Z in figure display (Unit: 0.0001mm / 0.0001inch)

The Min. value of Z: The Max. value along Z in figure display (Unit: 0.0001mm / 0.0001inch)

B. The setting method of figure parameter:

a. Move the cursor under the set parameter;

b. Input the corresponding numerical value based upon the actual requirement;

c. Confirm it by 

.

2) Figure interface Enter the interface interface by【⊞FIGURE】. (Refer to Fig. **3-5-2):**

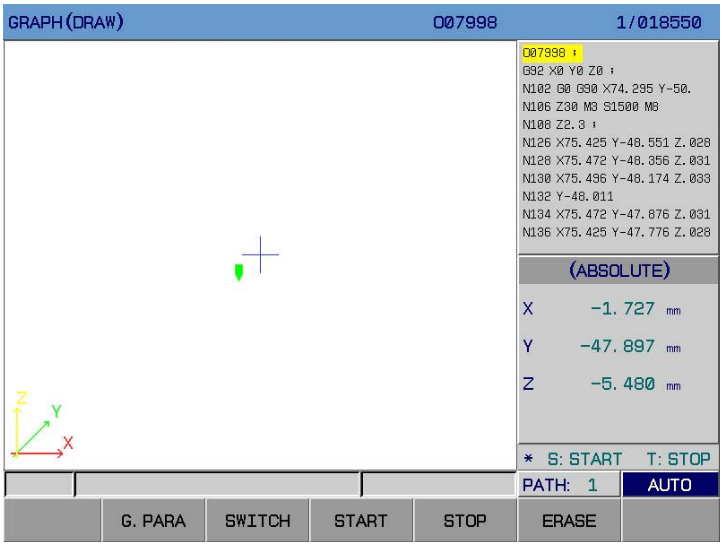


Fig. 3-5-2

The machining path of the operated program can be monitored in the figure page.

A. Enter to the drawing state by【START】or 

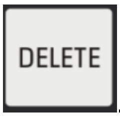
, in this case, '*' moves to **S: Before the drawing start;**

B. Enter the stop drawing state by【STOP】or 

, in this case, '*' moves to **T: Before the**

drawing stop;

C. The figure shifts in the corresponding figures **0**～**5** by pressing【SHIFT】every time.

D. Clear the drawn figure by【Clear】or 

.

3.6 Diagnosis Display

The state of DI/DO signal between the **CNC** and machine tool, the signal state for transmitting between the CNC and PLC, and the PLC internal data and CNC internal state are displayed on the diagnosis page. The corresponding meanings and setting methods for each diagnosis number are shown the ***GSK 218MC*** *CNC System* ***PLC*** *& Installation Connection Manual*.

This diagnosis uses for detecting the CNC interface signal and internal signal operation state, which can not be modified.

Enter the diagnosis display interface by 

; there are 5 sub-interface in this page:【⊞SIGNAL】,【SYSTEM】,【BUS】,【DSP】and【⊞FLUCTUATION】, which can be viewed by its corresponding software (Refer to the Fig. **3-6-1).**

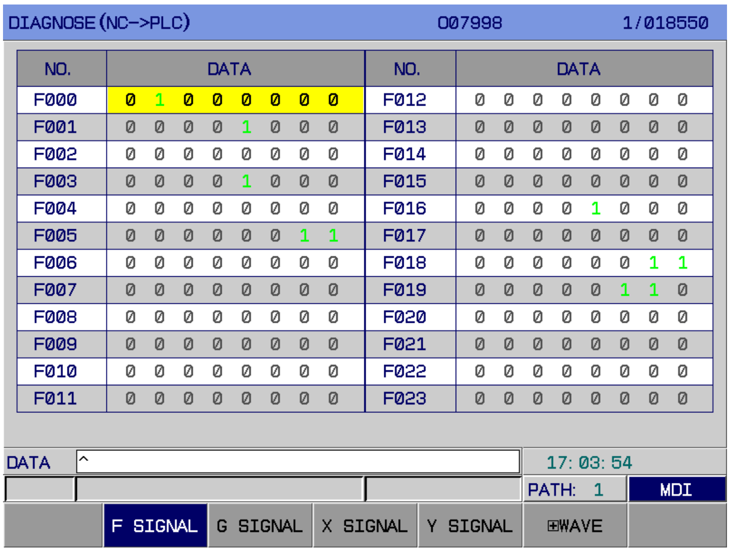


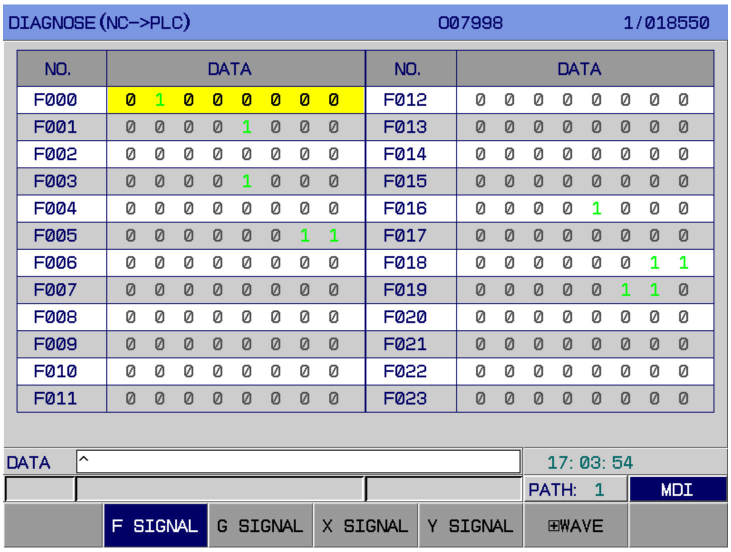
Fig. 3-6-1

3.6.1 Diagnosis Data Display

3.6.1.1 Signal Parameter Display

Enter the signal diagnosis interface by【SIGNAL】softkey. The content of this interface is shown below (Refer to the Fig. **3-6-1-1-1**~Fig.**3-6-1-1-4).**

1. F signal interface Enter the diagnosis (NC→PLC) interface by【F SIGNAL】in the <DIAGNOSIS> interface. Refer to the Fig. **3-6-1-1-1.**



DIAGNOSE (NC->PLC) 007998 1/018550

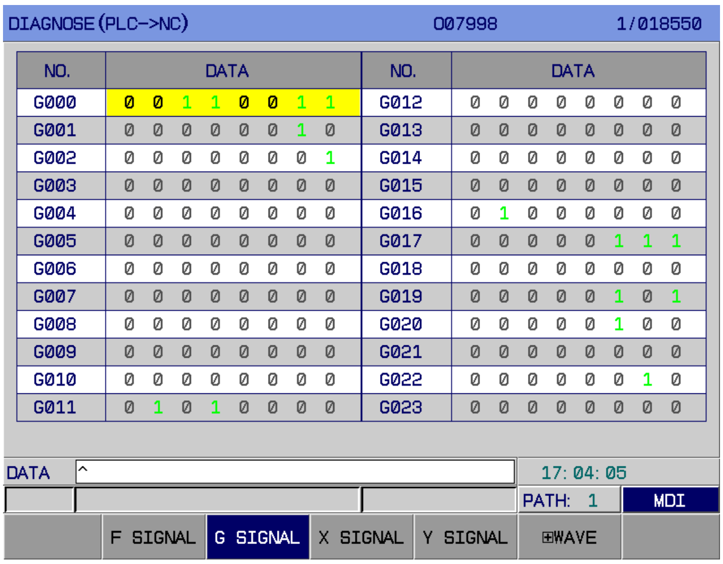
NO.	DATA	NO.	DATA
F000	0 1 0 0 0 0 0 0	F012	0 0 0 0 0 0 0 0
F001	0 0 0 0 1 0 0 0	F013	0 0 0 0 0 0 0 0
F002	0 0 0 0 0 0 0 0	F014	0 0 0 0 0 0 0 0
F003	0 0 0 0 1 0 0 0	F015	0 0 0 0 0 0 0 0
F004	0 0 0 0 0 0 0 0	F016	0 0 0 0 1 0 0 0
F005	0 0 0 0 0 0 1 1	F017	0 0 0 0 0 0 0 0
F006	0 0 0 0 0 0 0 0	F018	0 0 0 0 0 0 1 1
F007	0 0 0 0 0 0 0 0	F019	0 0 0 0 0 1 1 0
F008	0 0 0 0 0 0 0 0	F020	0 0 0 0 0 0 0 0
F009	0 0 0 0 0 0 0 0	F021	0 0 0 0 0 0 0 0
F010	0 0 0 0 0 0 0 0	F022	0 0 0 0 0 0 0 0
F011	0 0 0 0 0 0 0 0	F023	0 0 0 0 0 0 0 0

DATA ^ 17: 03: 54
PATH: 1 MDI
F SIGNAL G SIGNAL X SIGNAL Y SIGNAL ⊞WAVE

Fig. 3-6-1-1-1

This signal is delivered to PLC from system, its meanings and setting methods of each diagnosis number are shown in the matched ***GSK 218MC*** *CNC System* ***PLC*** *& Installation Connection Manual*.

2. G signal interface Enter the diagnosis (PLC→NC) interface by【G SIGNAL】softky in the <DIAGNOSIS> interface. Refer to the Fig. **3-6-1-1-2:**



DIAGNOSE (PLC->NC) 007998 1/018550

NO.	DATA	NO.	DATA
G000	0 0 1 1 0 0 1 1	G012	0 0 0 0 0 0 0 0
G001	0 0 0 0 0 0 1 0	G013	0 0 0 0 0 0 0 0
G002	0 0 0 0 0 0 0 1	G014	0 0 0 0 0 0 0 0
G003	0 0 0 0 0 0 0 0	G015	0 0 0 0 0 0 0 0
G004	0 0 0 0 0 0 0 0	G016	0 1 0 0 0 0 0 0
G005	0 0 0 0 0 0 0 0	G017	0 0 0 0 0 1 1 1
G006	0 0 0 0 0 0 0 0	G018	0 0 0 0 0 0 0 0
G007	0 0 0 0 0 0 0 0	G019	0 0 0 0 0 1 0 1
G008	0 0 0 0 0 0 0 0	G020	0 0 0 0 0 1 0 0
G009	0 0 0 0 0 0 0 0	G021	0 0 0 0 0 0 0 0
G010	0 0 0 0 0 0 0 0	G022	0 0 0 0 0 0 1 0
G011	0 1 0 1 0 0 0 0	G023	0 0 0 0 0 0 0 0

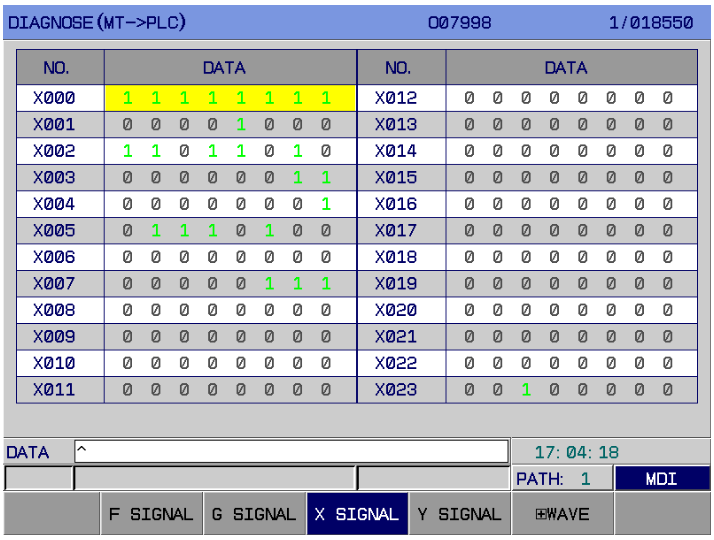
DATA ^ 17: 04: 05
PATH: 1 MDI
F SIGNAL G SIGNAL X SIGNAL Y SIGNAL ⊞WAVE

Fig. 3-6-1-1-2

VOLUME II OPERATION

This signal is delivered to PLC from system, its meanings and setting methods of each diagnosis number are shown in the matched ***GSK 218MC*** *CNC System* ***PLC*** *& Installation Connection Manual*.

3. X signal interface Enter the diagnosis (MT→PLC) interface by【X SIGNAL】softkey in the <DIAGNOSIS> interface. Refer to the Fig. **3-6-1-1-3:**

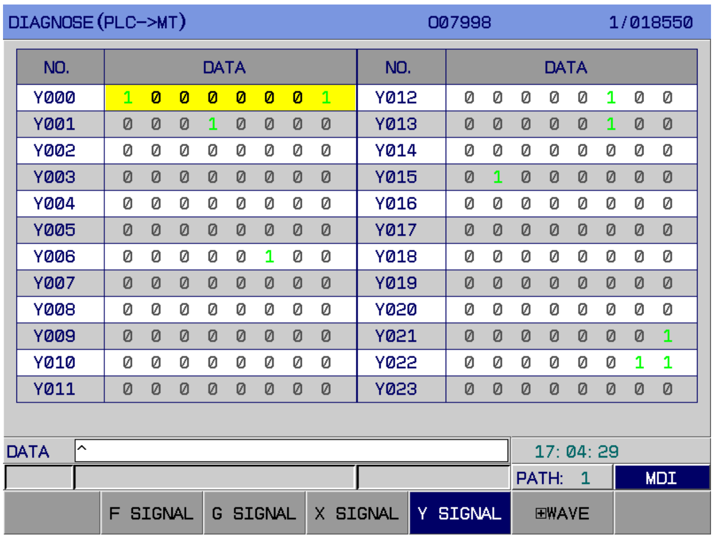


NO.	DATA	NO.	DATA
X000	1 1 1 1 1 1 1 1	X012	0 0 0 0 0 0 0 0
X001	0 0 0 0 1 0 0 0	X013	0 0 0 0 0 0 0 0
X002	1 1 0 1 1 0 1 0	X014	0 0 0 0 0 0 0 0
X003	0 0 0 0 0 0 1 1	X015	0 0 0 0 0 0 0 0
X004	0 0 0 0 0 0 0 1	X016	0 0 0 0 0 0 0 0
X005	0 1 1 1 0 1 0 0	X017	0 0 0 0 0 0 0 0
X006	0 0 0 0 0 0 0 0	X018	0 0 0 0 0 0 0 0
X007	0 0 0 0 0 1 1 1	X019	0 0 0 0 0 0 0 0
X008	0 0 0 0 0 0 0 0	X020	0 0 0 0 0 0 0 0
X009	0 0 0 0 0 0 0 0	X021	0 0 0 0 0 0 0 0
X010	0 0 0 0 0 0 0 0	X022	0 0 0 0 0 0 0 0
X011	0 0 0 0 0 0 0 0	X023	0 0 1 0 0 0 0 0

Fig. 3-6-1-1-3

This signal is delivered to PLC from system, its meanings and setting methods of each diagnosis number are shown in the matched ***GSK 218MC*** *CNC System* ***PLC*** *& Installation Connection Manual*.

4. Y signal interface Enter the diagnosis (PLC→MT) interface by【Y SIGNAL】softkey in the <DIAGNOSIS> interface. Refer to the Fig. **3-6-1-1-4:**



NO.	DATA	NO.	DATA
Y000	1 0 0 0 0 0 0 1	Y012	0 0 0 0 0 1 0 0
Y001	0 0 0 1 0 0 0 0	Y013	0 0 0 0 0 1 0 0
Y002	0 0 0 0 0 0 0 0	Y014	0 0 0 0 0 0 0 0
Y003	0 0 0 0 0 0 0 0	Y015	0 1 0 0 0 0 0 0
Y004	0 0 0 0 0 0 0 0	Y016	0 0 0 0 0 0 0 0
Y005	0 0 0 0 0 0 0 0	Y017	0 0 0 0 0 0 0 0
Y006	0 0 0 0 0 1 0 0	Y018	0 0 0 0 0 0 0 0
Y007	0 0 0 0 0 0 0 0	Y019	0 0 0 0 0 0 0 0
Y008	0 0 0 0 0 0 0 0	Y020	0 0 0 0 0 0 0 0
Y009	0 0 0 0 0 0 0 0	Y021	0 0 0 0 0 0 0 1
Y010	0 0 0 0 0 0 0 0	Y022	0 0 0 0 0 0 1 1
Y011	0 0 0 0 0 0 0 0	Y023	0 0 0 0 0 0 0 0

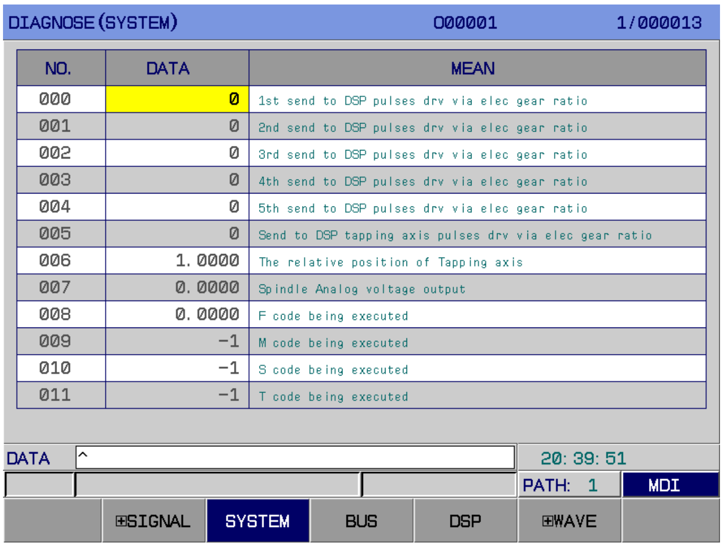
Fig. 3-6-1-**1**-4

This signal is delivered to PLC from system, its meanings and setting methods of each diagnosis number are shown in the matched ***GSK 218MC*** *CNC System* ***PLC*** *& Installation Connection Manual*.

VOLUME II OPERATION

3.6.1.2 System Parameter Display

Enter the system signal diagnosis interface by【SYSTEM】softkey. The content of this interface is shown below (Refer to Fig. **3-6-1-2-1**).

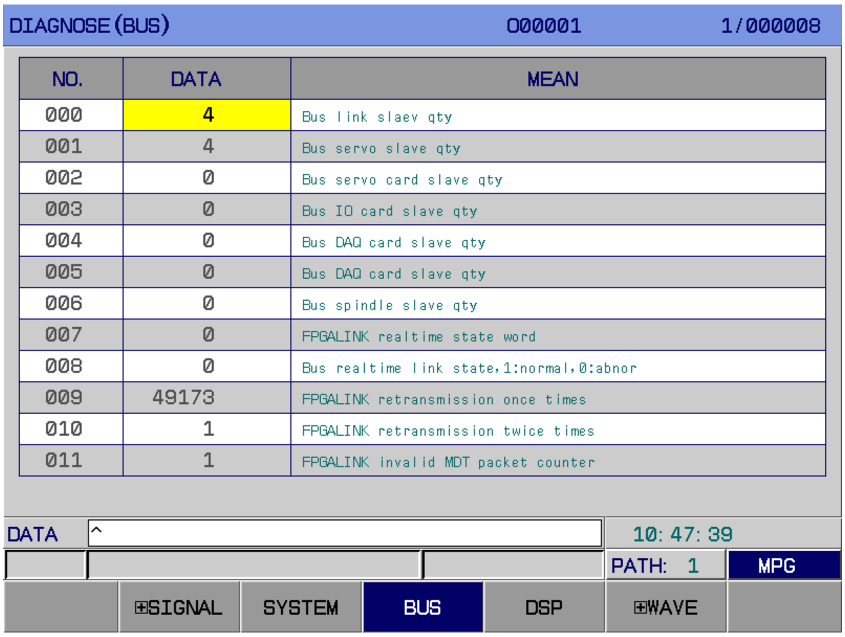


NO.	DATA	MEAN
000	0	1st send to DSP pulses drv via elec gear ratio
001	0	2nd send to DSP pulses drv via elec gear ratio
002	0	3rd send to DSP pulses drv via elec gear ratio
003	0	4th send to DSP pulses drv via elec gear ratio
004	0	5th send to DSP pulses drv via elec gear ratio
005	0	Send to DSP tapping axis pulses drv via elec gear ratio
006	1.0000	The relative position of Tapping axis
007	0.0000	Spindle Analog voltage output
008	0.0000	F code being executed
009	-1	M code being executed
010	-1	S code being executed
011	-1	T code being executed

Fig. 3-6-1-2-1

3.6.1.3 Bus Parameter Display

Enter the bus signal diagnosis interface by【BUS】. The content of this page is shown below (Refer to the Fig. **3-6-1-3-1).**



NO.	DATA	MEAN
000	4	Bus link slaev qty
001	4	Bus servo slave qty
002	0	Bus servo card slave qty
003	0	Bus IO card slave qty
004	0	Bus DAQ card slave qty
005	0	Bus DAQ card slave qty
006	0	Bus spindle slave qty
007	0	FPGALINK realtime state word
008	0	Bus realtime link state,1:normal,0:abnor
009	49173	FPGALINK retransmission once times
010	1	FPGALINK retransmission twice times
011	1	FPGALINK invalid MDT packet counter

Fig. 3-6-1-3-1

3.6.1.4 DSP Parameter Display

Enter the bus signal diagnosis interface by【DSP】. The content of this page is shown below (Refer to the Fig. **3-6-1-4-1).**

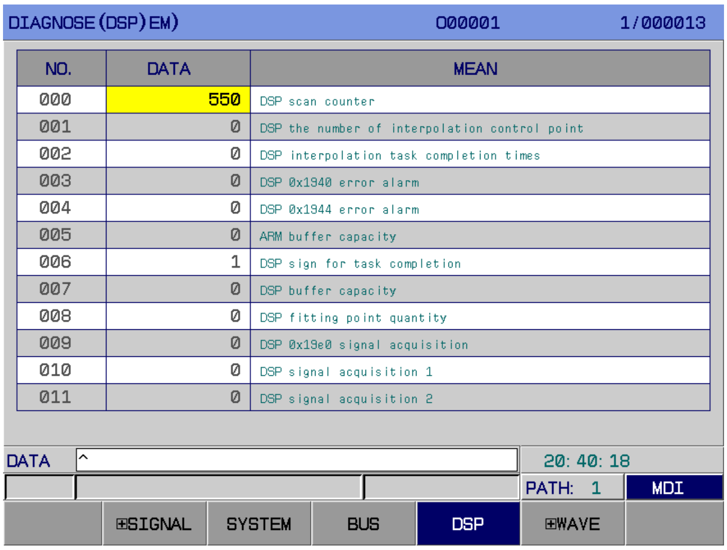


Fig. 3-6-1-4-1

3.6.1.5 Fluctuation Parameter Display

Enter the fluctuation interface by【FLUCTUATION】softkey. Refer to the Fig. **3-6-1-5-1**:

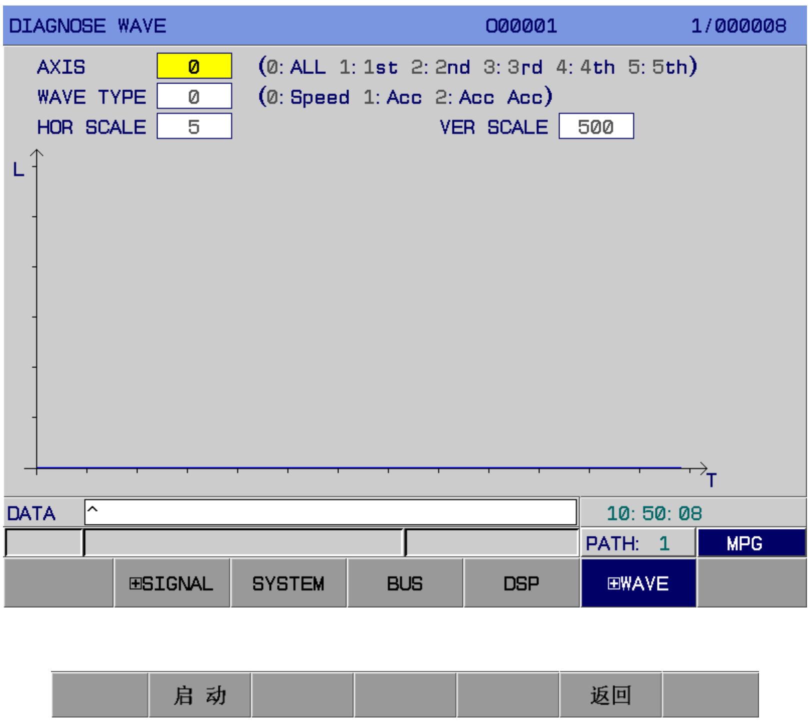


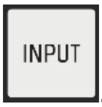
Fig. 3-6-1-5-1

VOLUME II OPERATION

Axis selection: Select the axis to be performed the fluctuation diaganosis.

Fluctuation selection: Select the content of the diagnosed fluctuation.

Proportion of horizontal axis, vertical axis: Select the drawn proportion

Data: In any mode, input the corresponding data, then confirm it by 

.

Signal monitoring performs by <START> button, and stop it by <STOP> button.

3.6.2 Check Signal State

1) Select the corresponding display interface by 

.

2) The corresponding address explanation and meaning at the lower left of the screen when moving the cursor left or right.

3) You can find the destination address by moving the cursor or input the desired search parameter address, then control the 

button.

4) **【FLUCTUATION】** interface can be displayed the velocity, acceleration and addition acceleration for debugging, and then find the optimized adapted parameter of the drive and motor.

3.7 Alarm Display

The “alarm” information displays at the lower-left-corner on the screen when the system alarm issues. In this case, the display page appear by 

. There are four display interfaces in this page:【ALARM】,【USER】,【HISTORY】,【RECORD】, which can be checked (Refer to the Fig. **3-7-1**～Fig.**3-7-4)** by its corresponding softkey. Alternatively, whether shifting to the alarm interface when bit 6 of parameter No.: 24 alarm occurs.

1. Alarm interface　Enter the alarm interface by【ALARM】softkey in <ALARM> interface, refer to the Fig. **3-7-1**:

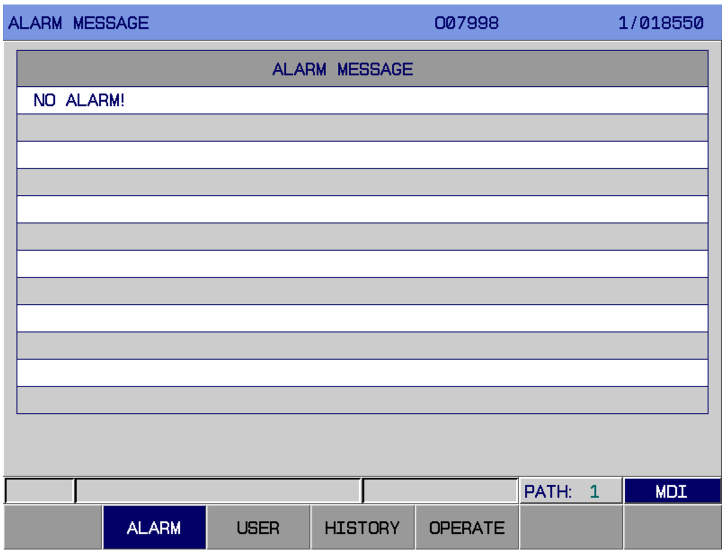


Fig. 3-7-1

Display the detailed content of the current P/S alarm number on the alarm display screen. Refer to the appendix two for the detailed alarm content.

2. User interface Enter the external alarm interface by【USER】softkey in <ALARM> interface, refer to the Fig. **3-7-2:**

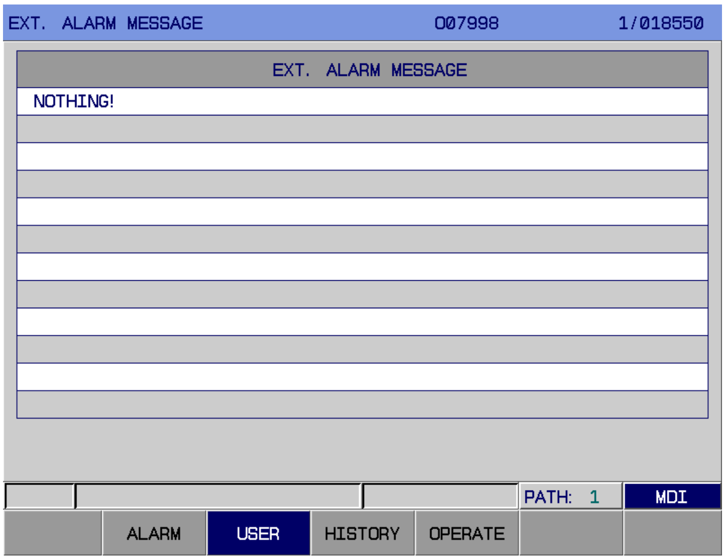


Fig. 3-7-2

The detailed content of each user alarm information shows in the ***GSK 218MC CNC System PLC & Installation Connection Manual***.

Note: The external alarm can be set and edited the alarm number based upon the on-site actual circumstance by the user, and the alarm content after edited inputs to the system by the system transmission software. The external alarm is the A of the edited file LadChi.txt, the followed two-digit are set by the**

value of the bit parameters 53.0~53.3. (Default vaue is 01, that is, the file name is LadChi01.txt).

3. History interface Enter the history alarm information interface by【HISTORY】softkey in <ALARM> interface, refer to the Fig. **3-7-3:**

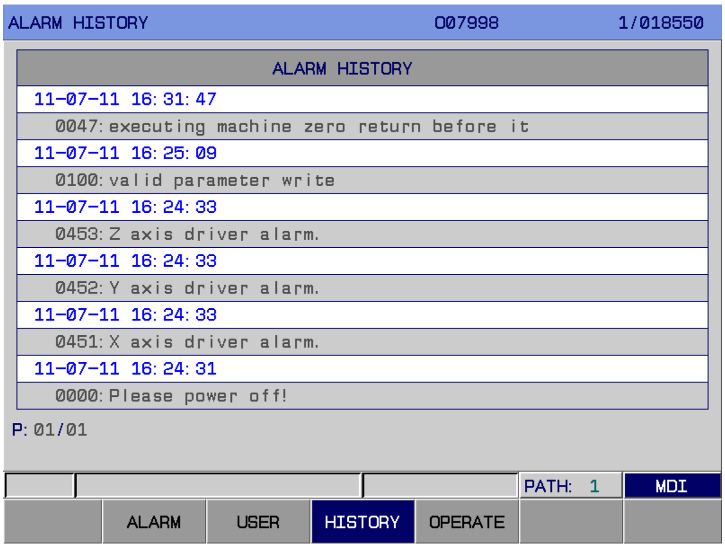


Fig. 3-7-3

This items within this interface are arranged from near to far based upon the time sequence, so that the user can easily view it.

4. Record interface Enter the external alarm interface by【RECORD】softkey in <ALARM> interface, refer to the Fig. **3-7-4:**

The content of operation record interface is the concrete modification information for the system parameter and ladder diagram, such as the modification content and time, etc.

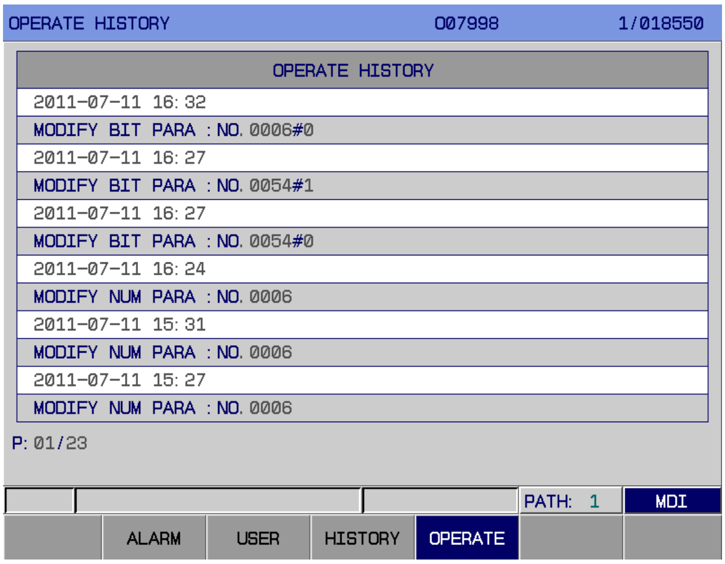


Fig. 3-7-4

Operation record can be display 34 pages, and history alarm information can be shown 9 pages, such as, the alarm time, alarm number, alarm information, page, etc. which can be viewed by page up/down button.

The record of history and record can be deleted (The password level is the debugging or more than the debugging level) by DELETE button.

3.8 Program-Control Display

Enter the program-control display page by PLC, and there are 5 display interfaces in the page:【INFO】,【⊞PLCGRA】,【⊞PLCPAR】,【PLCDGN】and【⊞PLCTRACE】, which can be shifted by the corresponding softkey. The concrete content is shown below (Refer to the Fig. **3-8-1**～**3-8-5).**

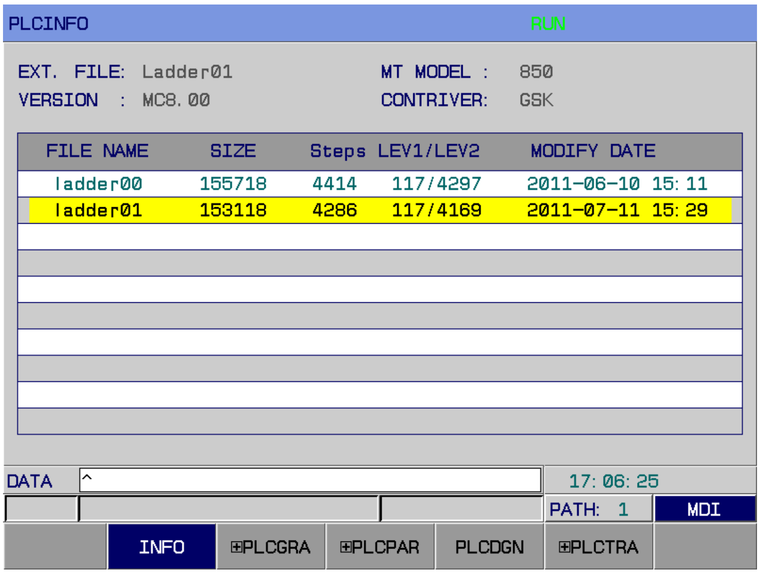


Fig. 3-8-1

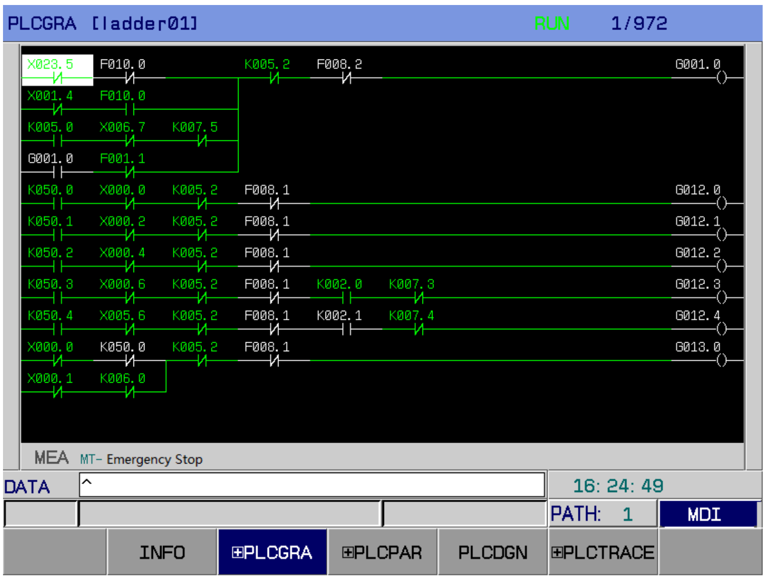
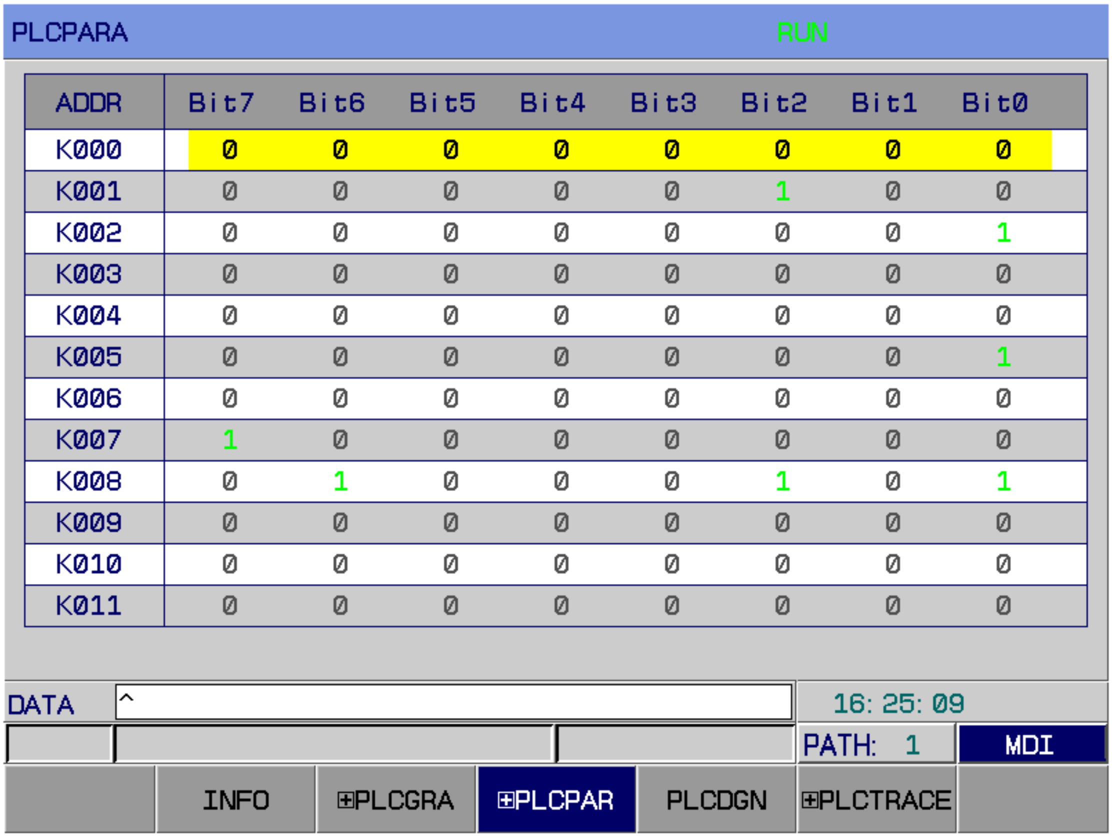


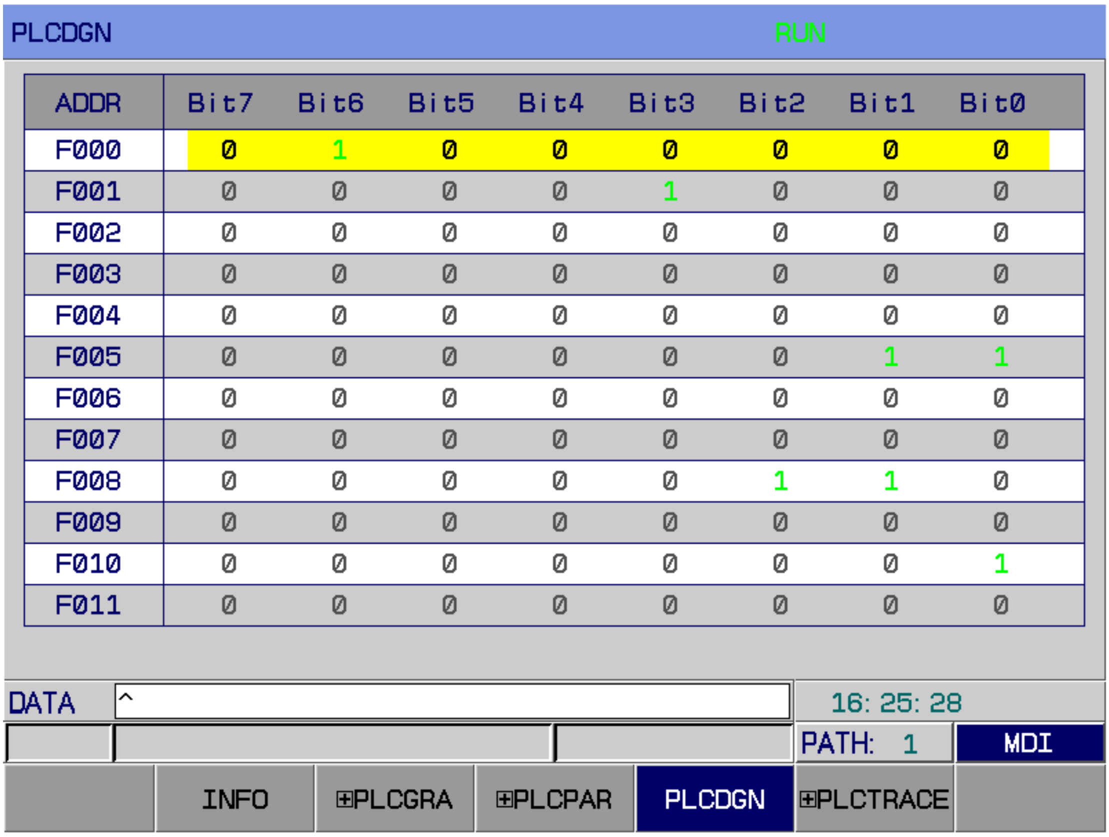
Fig. 3-8-2

VOLUME II OPERATION



ADDR	Bit7	Bit6	Bit5	Bit4	Bit3	Bit2	Bit1	Bit0
K000	0	0	0	0	0	0	0	0
K001	0	0	0	0	0	1	0	0
K002	0	0	0	0	0	0	0	1
K003	0	0	0	0	0	0	0	0
K004	0	0	0	0	0	0	0	0
K005	0	0	0	0	0	0	0	1
K006	0	0	0	0	0	0	0	0
K007	1	0	0	0	0	0	0	0
K008	0	1	0	0	0	1	0	1
K009	0	0	0	0	0	0	0	0
K010	0	0	0	0	0	0	0	0
K011	0	0	0	0	0	0	0	0

Fig. 3-8-3



ADDR	Bit7	Bit6	Bit5	Bit4	Bit3	Bit2	Bit1	Bit0
F000	0	1	0	0	0	0	0	0
F001	0	0	0	0	1	0	0	0
F002	0	0	0	0	0	0	0	0
F003	0	0	0	0	0	0	0	0
F004	0	0	0	0	0	0	0	0
F005	0	0	0	0	0	0	1	1
F006	0	0	0	0	0	0	0	0
F007	0	0	0	0	0	0	0	0
F008	0	0	0	0	0	1	1	0
F009	0	0	0	0	0	0	0	0
F010	0	0	0	0	0	0	0	1
F011	0	0	0	0	0	0	0	0

Fig. 3-8-4

VOLUME II OPERATION

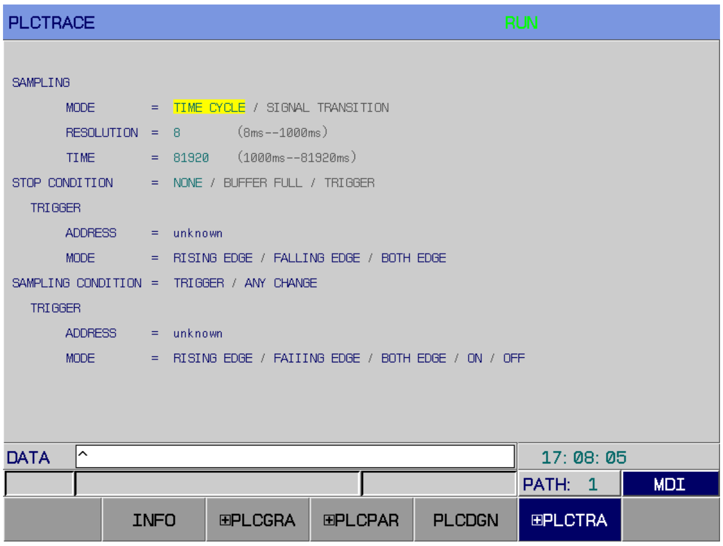


Fig. 3-8-5

The modification methods and relevant information for the PLC ladder diagram are shown in the ***GSK 218MC CNC System PLC & Installation Connection Manual***.

3.9 Help Display

Enter the help display page by , there are eight display interfaces in this page:【SYSTEM INFORMATION】【OPERATION TABLE】【ALARM TABLE】【G CODE TABLE】【PARAMETER TABLE】【MACRO COMMAND】【⊞PLC.AD】【COUNTER】, which can be viewed by its corresponding softkey. The displayed content shows below (Refer to Fig. **3- 9- 1**～**3- 9- 12).**

1. System information interface Enter the system information interface by【SYSTEM INFORMATION】softkey in <HELP> interface; refer to the Fig. **3-9-1:**

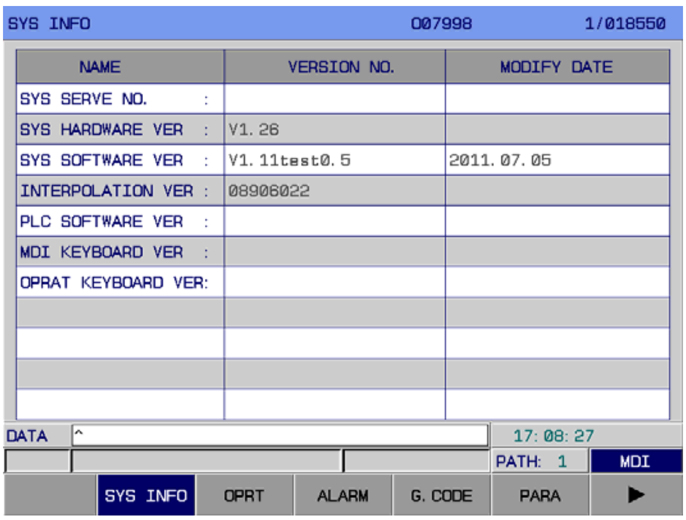


Fig. 3-9-1

2. Operation table interface Enter the help information (Operation list) page by【OPERATION TABLE】softkey in <HELP> interface; refer to the Fig. **3-9-2:**

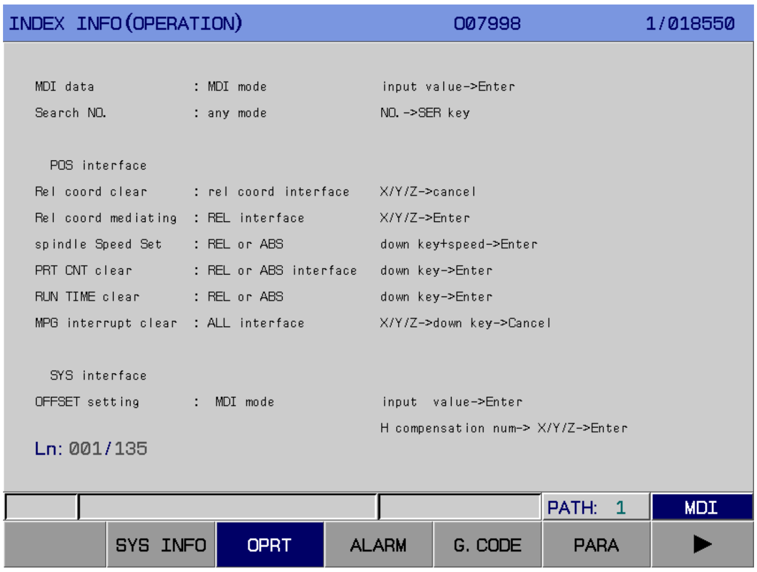


Fig. 3-9-2

In the help information (Operation list) interface, the operation steps and methods in each interface are detailed. The unfamiliar or unclear operations can be searched and compared in the help interface.

3. Alarm table interface Enter the help information (Alarm list) page by【ALARM TABLE】softkey in <HELP> interface; refer to the Fig. **3-9-3:**

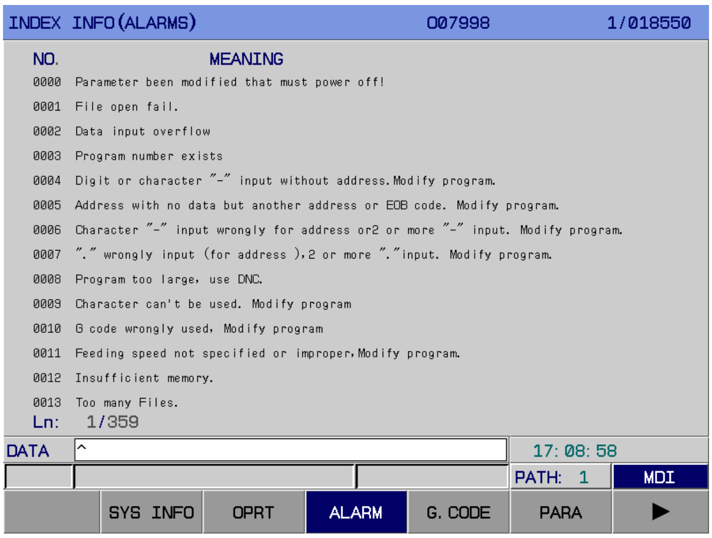
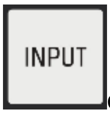


Fig. 3-9-3

The meanings and troubleshootings in this interface are detailed.

4. G code table interface Enter the help information (G code list) page by【G CODE TABLE】softkey in <HELP> interface; refer to the Fig. **3-9-4:**

Fig. 3-9-4

The definition of each G code used by the system is introduced in G code interface, select the G code to be check by cursor, and the G code definition shows at the lower-left-corner of the interface. Refer to the Fig. **3-9-4**. If you want to know the concrete format and usage of the G code, press the 

on the panel after selecting the G code, and then return by 

; refer to the Fig. **3-9-5**:

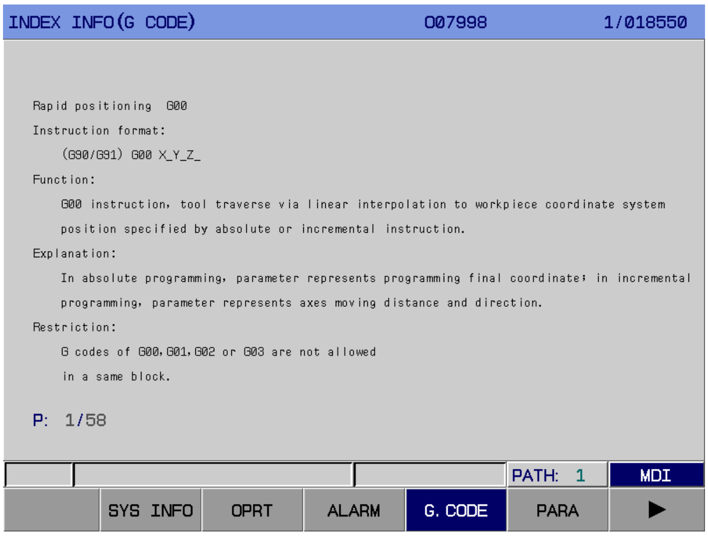


Fig. 3-9-5

The format, function, explanation and restriction of the code are detailed in this interface, as for the unfamiliar or unclear code can be searched and compared in this page.

5. Operation table interface Enter the help information (Parameter/Diagnosis table) page by 【PARAMETER TABLE】softkey in <HELP> interface; refer to the Fig. **3-9-6:**

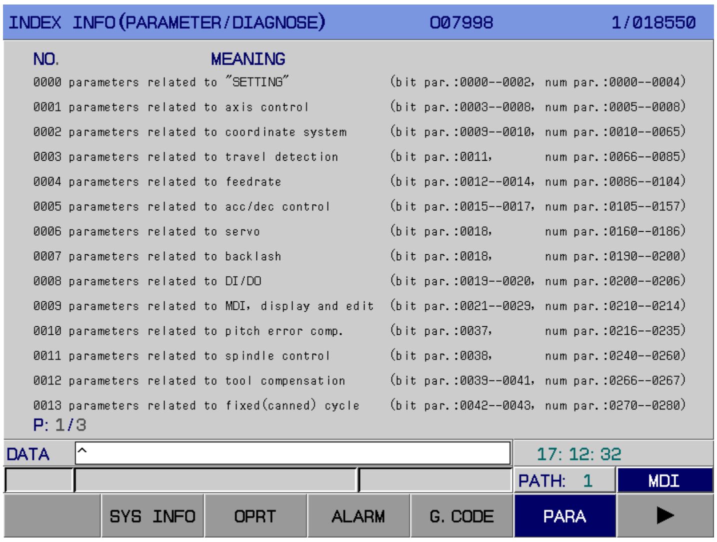


Fig. 3-9-6

The parameter setting of each function is detailed in this interface, the unfamiliar or unclear parameter setting can be searched and compared in the interface.

6. Macro command interface Enter the help information (Macro command list) page by【MACRO COMMAND LIST】softkey in <HELP> interface; refer to the Fig. **3-9-7:**

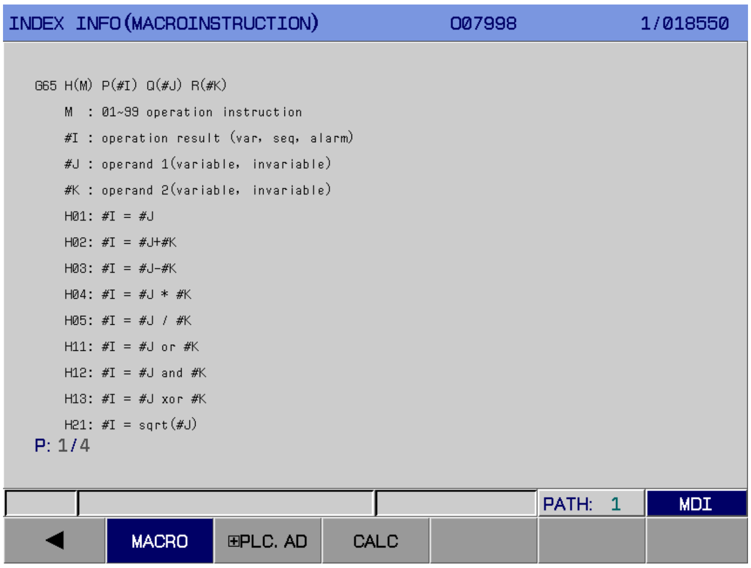


Fig. 3-9-7

The formats of macro commands and each operation code are introduced in this interface, and the setting range of the local variable, universal variable and the system variable are provided. As for the unfamiliar or unclear macro command operations can be searched and compared in the interface.

7. PLC.AD interface Enter the help information (PLC address list) page by【PLC.AD】softkey in <HELP> interface. There are four sub-interface【F ADDRESS】,【G ADDRESS】,【X ADDRESS】 and【Y ADDRESS】for the PLC address interface. The content shows below (Refer to the Fig. **3-9-8**～**3-9-11).**

VOLUME II OPERATION

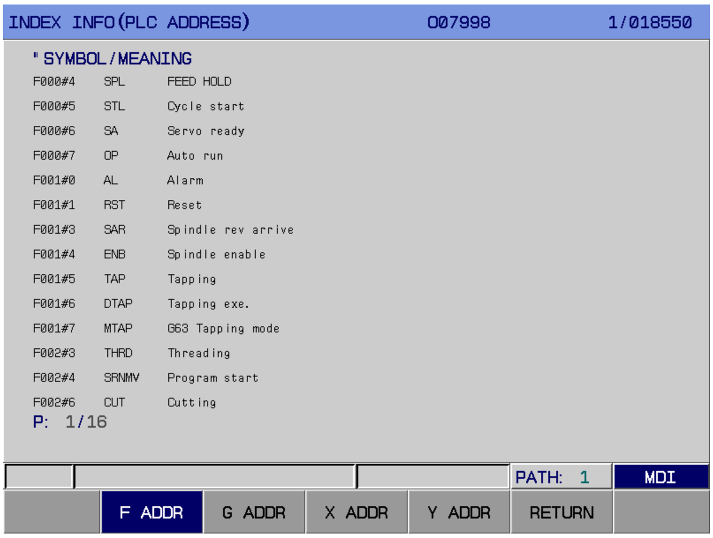


Fig. 3-9-8

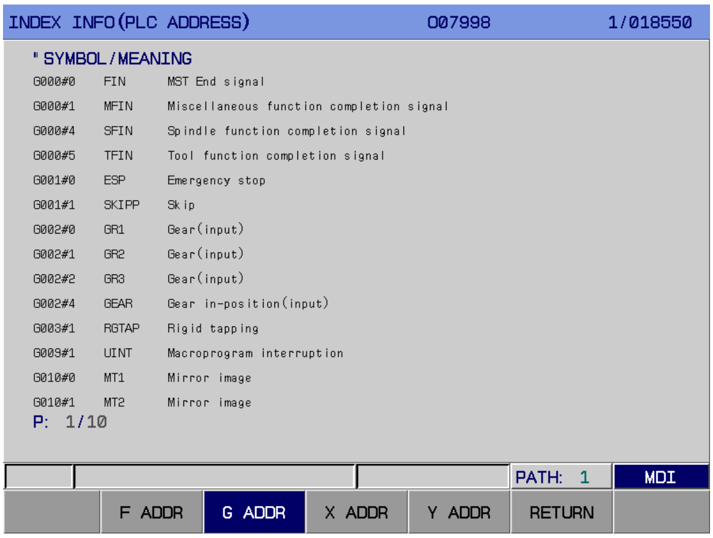


Fig. 3-9-9

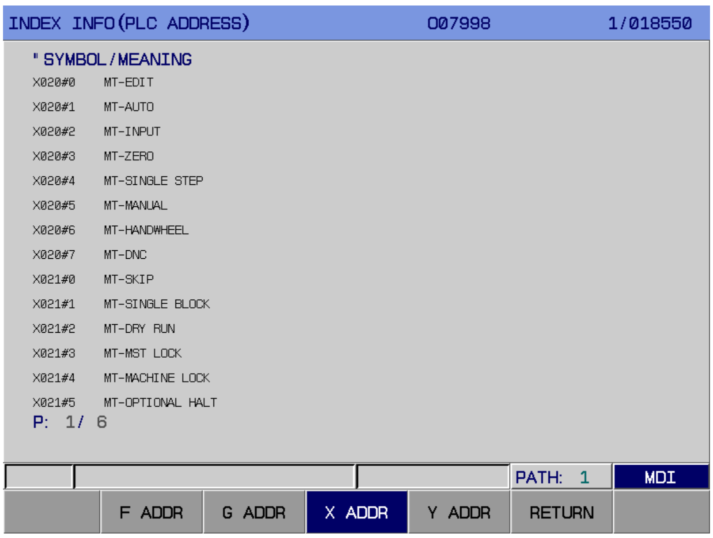


Fig. 3-9-10

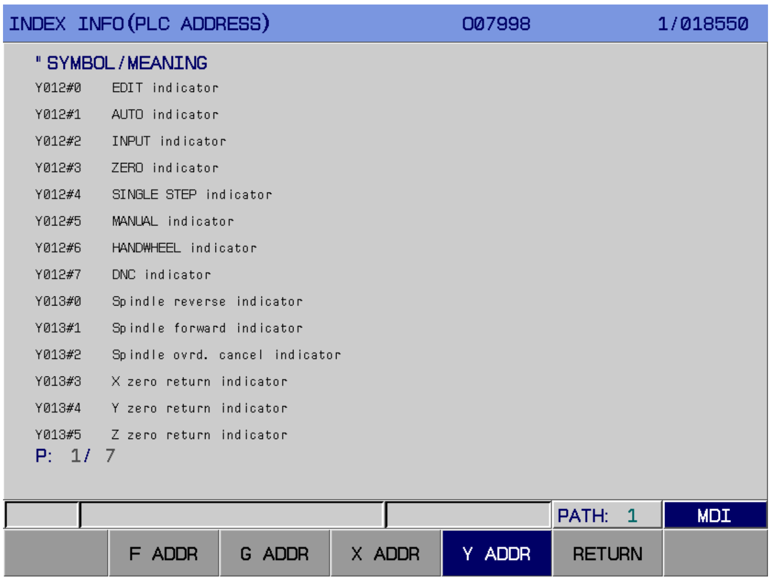


Fig. 3-9-11

The address, symbol and meaning of the PLC are detailed in the PLC address interface, the unfamiliar or unclear PLC addresses can be searched and compared in this page.

8. Counter interface Enter the counter information page by【COUNTER】softkey in <HELP> interface; refer to the Fig. **3-9-12:**

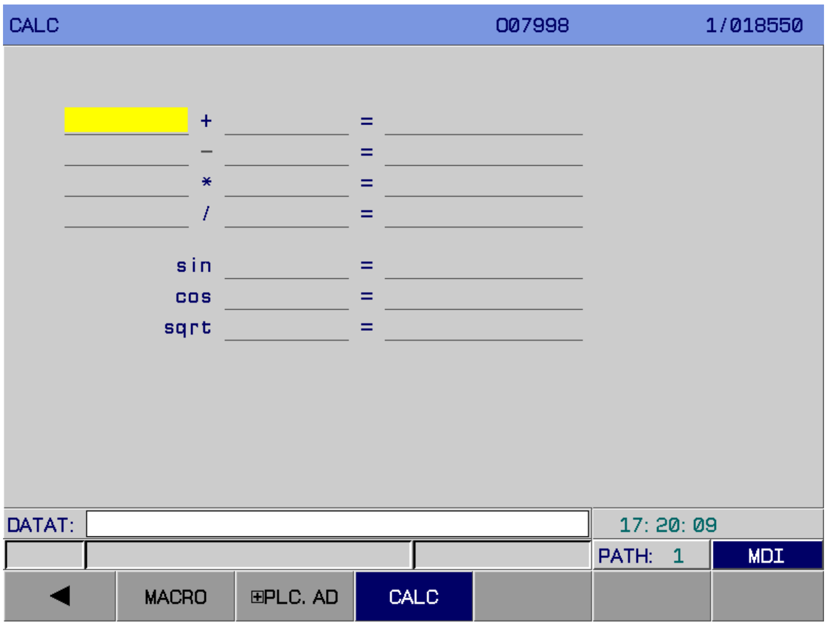
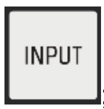


Fig. 3-9-12

In this page, the system provides the operation formats in this interface, such as the addition, subtraction, multiplication, devision, sine, cosine and extraction, move the cursor positioning to the space to be intput the data, and then input the data, then confirm it by 

; the system may automatically calculate the result after the desired data are input to the space followed the "=".

VOLUME II OPERATION

Pressing the 

can be input the data again to calculate, the overall data in this interface are then cleared.

CHAPTER FOUR MANUAL OPERATION

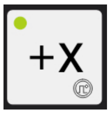
Enter the manual operation mode by 

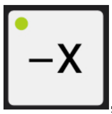
, which contains of the manual feed, spindle control and machine panel control, etc.

4.1 Coordinate Axis Movement

In the manual operation mode, each axis can be separately performed by the manual feedrate or manual rapid traverse rate.

4.1.1 Manual Feed

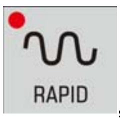
In the <MANUAL MODE>, press the feed axis 

or 

, the corresponding axis begins moving, and its movement speed can be changed by adjusting the feedrate; the axis movement stops after releasing the button; the same as the Y axis and Z axis. This system, temporarily, does not support the multi-axis movement at the same time, instead of the simultaneous zero return along each axis.

Note: The manual feedrate of each axis is set by parameter P98.

4.1.2 Manual Rapid Traverse

Press the 

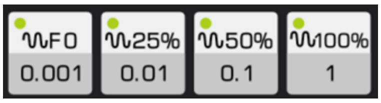
, the indicator is on, then enter the manual rapid traverse state, and then each axis operates at the rapid traverse rate pressing the button of the feed direction axis.

Note 1: The manual rapid traverse rate is set by P170~ P173.

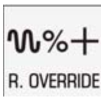
Note 2: The bit 0 of parameter No.: 12 can be set whether the manual rapid traverse is enabled before the reference position return.

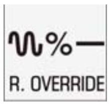
4.1.3 Manual Feed & Manual Rapid Traverse Rate Selection

In the manual feed, select the manual feedrate by band switch, total 21 levels (0%--200%).

In the manual rapid traverse rate, 218MC, 218MC-H or 218MC-V can be selected override of the manual rapid traverse rate by 

, and total 4 gears for the rapid override: Fo, 25%, 50% and 100%. Fo speed is set by data parameter **P93**).

218MC-U1 can be modified by 

and 

buttons.

Note: The rapid override selection can be available for the following traverse rate.

(1) G00 rapid feed

(2) Rapid feed in canned cycle

(3) The rapid feed in G28

(4) Manual rapid feed

For example: When the rapid feedrate is 6 m/min, if the override is 50%, the speed is then 3 m/min.

4.1.4 Manual Intervention

In the Auto, MDI and DNC modes, the program is converted to the manual mode during operating after the feed holde is performed, and then the manual intervention should be operated. After moving each axis, then convert at the previous operated method; when this program is operated by 

, each axis rapidly returns to the origin manual intervention point based upon the G00 mode, and then continue the operation program.

Detailed explanations:

1. If single block operation is switched on during the return operation, the tool will perform the single dwell at the manual intervention point.
2. If the alarm or reset occurs in the manual intervention or return, this function will be cancelled.
3. When using the manual intervention, it is important to carefully use the machine lock, mirror and scaling function.
4. When the manual intervention is performed, it is note that the machining procedure and machining shape to avoid damage the tool or machine tool.

The manual intervention operation is shown below:

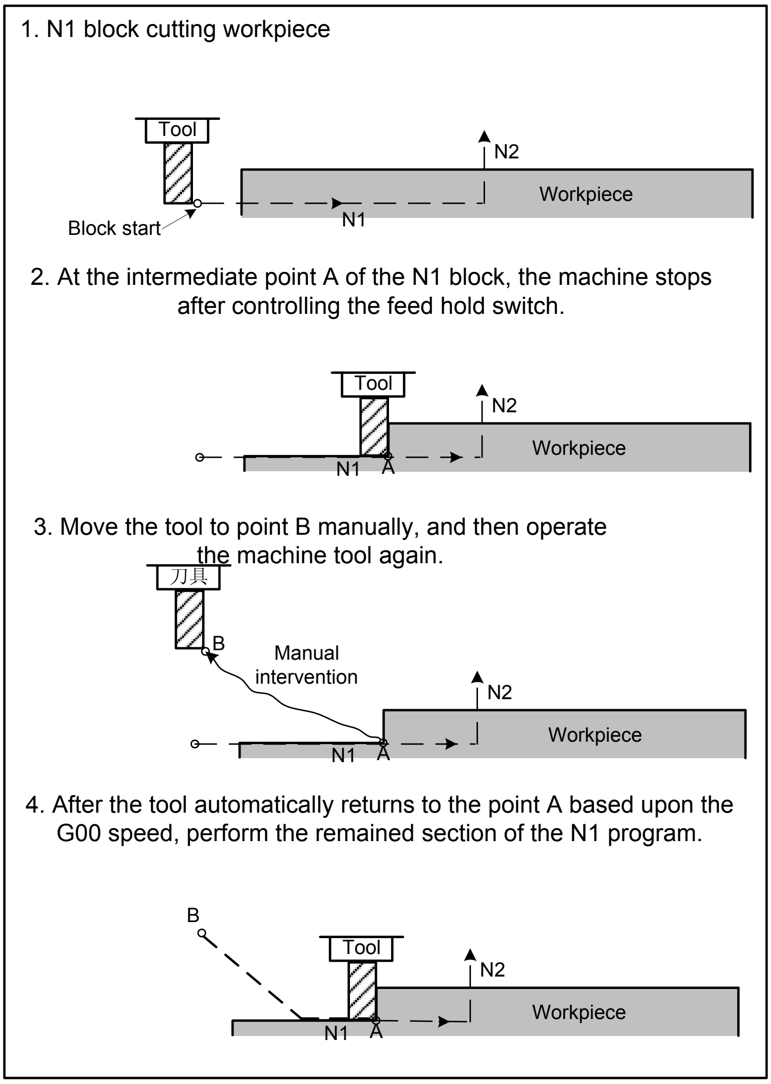


Fig. 4-1-4-1

4.1.5 Workpiece Correction

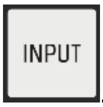
In order to guarantee the machining accuracy (dimension, shape and position precision) of the component and surface quality, it is necessary to correct the position of the workpiece or the fixture for clamping the workpiece.

The common correction method contains: drawing, trial cut, etc. as for its character, **GSK 218MC** CNC designs the correcting operation method for specially using the tool. For example, the center of a rectangle workpiece X-Y plane is positioned based upon the trial cut halving correction method (It is also called as Center Correction), the operation steps are shown below:

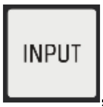
1) Start the spindle with a certain rotation speed.

2) System shifts to the display interface of the relative coordinate. Firstly correct the X direction: Position to the one side of the workpiece along X axis based upon the operation of the movement axis with manual mode; and then move downward to Z axis, so that the tool nose is

lower the workpiece surface; lastly, move toward the negative of the workpiece with the lower speed (Usually, use the MPG feed mode), till the tool is just cut to stop of the workpiece. In this case, press the  in the editing panel area, then the 

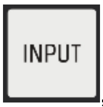
, and the X coordinate sets to zero. (If it sets to other values, you can use the same method, for example, input "X20" to confirm by 

.)

3) Similiarly, move the tool and cut one side along the negative direction, press  after positioning, then the press 

, and then complete the center operation. It is note that the setting of the center does not change the absolute and machine tool coordinate values.

4) Move the tool to the axis which the relative coordinate displays as 0, that is, the center along the X direction.

5) In the "Setting" interface, select the "Workpiece coordinate" page, press the  then the 

, and then the zero setting along X axis is executed.

6) The floating coordinate system can be set by G92 at the center of the XY (The XY value of the relative coordinate is 0, the positioning point of the machine tool), as well, the XY machine coordinate of this point records within the parameter of the G54～G59 coordinate coordinate system for calling by system.

7) The operation is completed for correcting the rectangular workpiece center by using the trial cutting center method.

Flexibly understand the method of the relative assignment and the center function setting to improve the correction speed and enhance the convenient of operation.

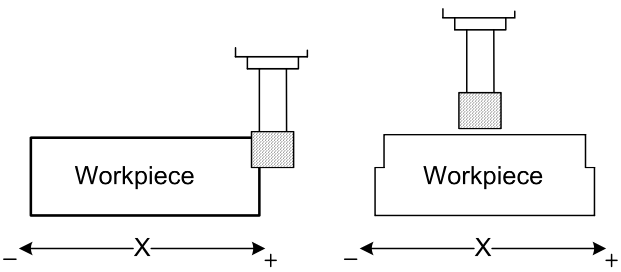


Fig. 4-1-5-1

Note 1: This system is only input the coordinate at the relative position displayed. (The place where can be modified the offset can also be set the position of the relative coordinate.)

VOLUME II OPERATION

Note 2: It owns the operation function, which can be performed the addition and subtraction operations to the displayed coordinate value, before the displayed coordinate is set.

Note 3: After the coordinate system is set, if the coordinate system set by G92 will lost due to the mechanical zero return or coordinate systems G54～G59 calling; it may not lose if the mechanical coordinate records to G54～G59 by parameter. The operator may flexibly set based upon their requirements; usually, it is suggested to use the latter.

4.2 Spindle Control

4.2.1 Spindle Positive

Specify the S speed in MDI mode; press this button in the Manual/MPG/single-step mode, spindle rotates CCW.

4.2.2 Spindle Negative

Specify the S speed in MDI mode; press this button in the Manual/MPG/Single-step mode, spindle rotates CW.

4.2.3 Spindle Stop

In the Manual/MPG/Single-step mode, spindle stops rotation by pressing this button.

4.2.4 Automatic Shift of Spindle

Select the spindle whether is the frequency control or I/O point control by bit 2 of parameter No. 1. When NO:1#2=0, the spindle speed is controlled by S speed command to carry out the automatic shift; at present, the system can be performed 3-shift control, and its corresponding top rotation speed is set by parameter (P246, P247 or P248). When NO:1#2=1, the automatic shifting of the spindle speed is controlled by I/O point; at present, the system can be both performed 3-shift control (S1, S2 or S3) and modified the shifting addition output of the ladder diagram. The system may automatically perform the corresponding shifting selection after executing the S speed command.

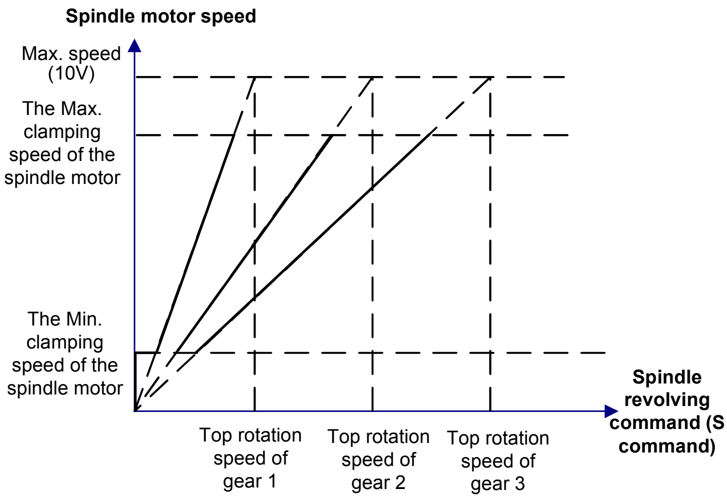


Fig. 4-2-4-1

Note: The automatic shifting is enabled, detect the spindle shift and perform the S code by the shifting in-position signal.

4.3 Other Manual Operations

4.3.1 Coolant Control

: The coolant can be shifted between the ON and OFF. The indicator ON means that the power is turned on, OFF is turned off.

4.3.2 Lubrication Control

: It is turned on by pressing the lubrication button; it is turned off by releasing it. The indicator ON means that the power is turned on, OFF is turned off.

4.3.3 Chip-removal Control

: The chip-revmoval can be shifted between the ON and OFF. The indicator ON means that the power is turned on, OFF is turned off.

4.3.4 Workping Indicator Control

: 

The working indicator can be shifted between the ON and OFF. The indicator ON means that the power is turned on, OFF is turned off.

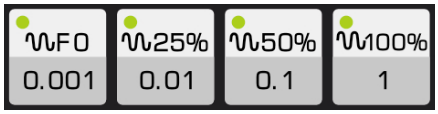
CHAPTER FIVE SINGLE-STEP OPERATION

5.1 Single-step feed

Enter the signle-step mode by , 

machine tool moves based upon the step-length defined by system each time in this mode.

5.1.1 Movement Amount Selection

Select a movement increment by any of the 

, and the movement increment can be displayed on the page; for example, press the 

, the single step length (0.100) displays in the <POSITION> interface. (Refer to the Fig. **5-1-1-1**)

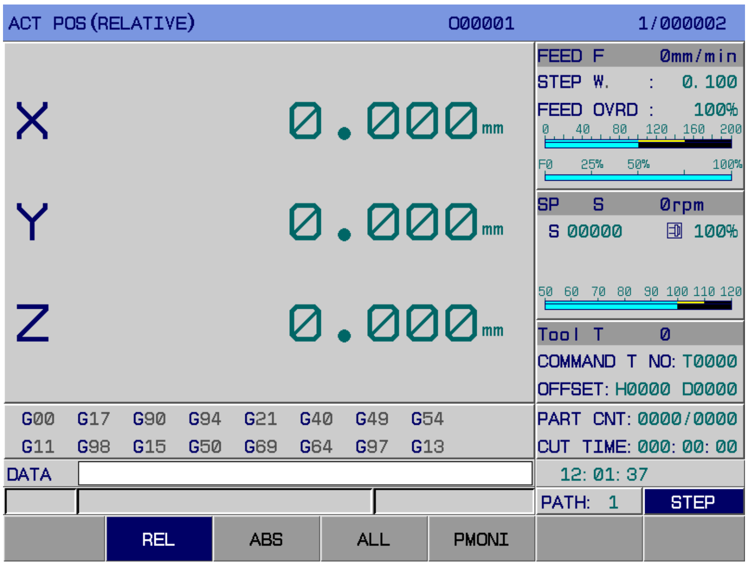


Fig. 5-1-1-1

The machine tool corresponding axis moves 0.1mm by pressing the movement axis once.

218MC-U1 can be controlled by 

buttons, and the modification and debugging methods are identical with the above-mentioned.

5.1.2 Selection of Movement axis and Movement Direction

The feed axis and its direction buttons +X or −X, the X axis can be moved positively or negative along the X axis direction button, press its corresponding button once, and the corresponding axis moves the distance of the system single-step definition; the same as the Y and Z axes. This system does not temporarily support the manual 3-axis movement at the same time, but the 3-axis can be performed the zero return simultaneously.

5.1.3 Single-step Feed Explanations

The top clamping speed of the single-step feed is set by data parameter **P155**.

The single-step feedrate does not be controlled by the feedrate, rapid override.

5.2 Single-step Interruption

When the program is operated in the Auto, MDI and DNC mode, shift the single-step mode to perform the single-step interruption function after dwells. Single-step interrupition coordinate system is same as the MPG one, and the operation function is also similar as the MPG (The electric MPG is the Hand Pulse Generator---MPG, similarly hereinafter) one, refer to the *Control of MPG Interruption Operation* in Section 6.2 of OPERTION.

5.3 Auxiliary Control in Single-step Operation

It is same as the manual operation mode, refer to the Section 4.2 and 4.3 in this manual.

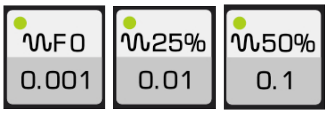
CHAPTER SIX MPG OPERATION

6.1 MPG Feed

Enter the MPG mode by 

, the machine tool movement can controlled by MPG in this feed mode.

6.1.1 Selection of Movement Amount

Select a movement increment by any of the 

, and the movement increment can be displayed on the page; for example, press the 

, the MPG increment (0.100) displays in the <POSITION> interface. (Refer to the Fig. **6-1-1-1**)

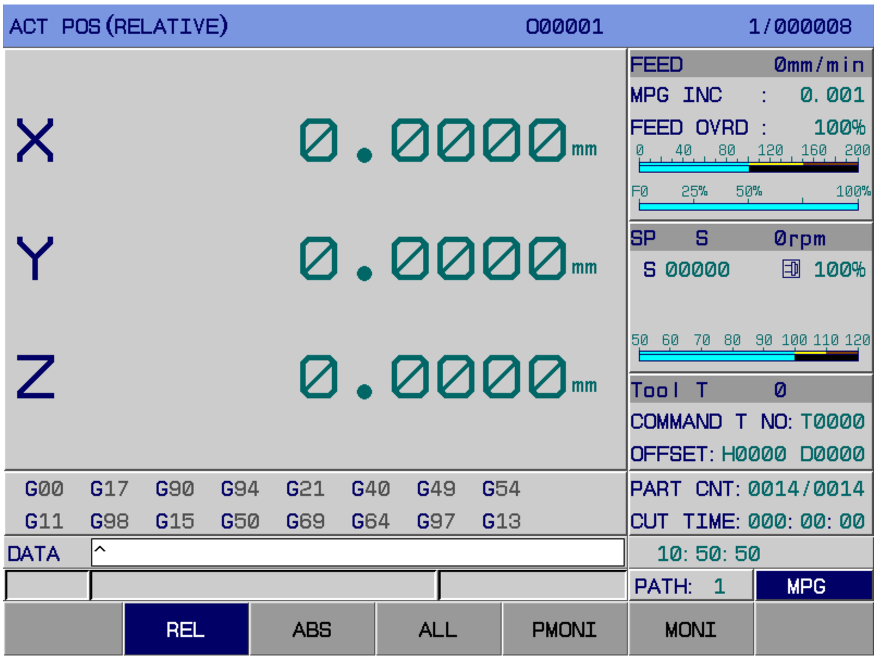


Fig. 6-1-1-1

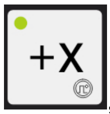
218MC-U1 can be controlled by 

buttons, and the modification and debugging methods are identical with the above-mentioned.

VOLUME II OPERATION

6.1.2 Selection of Movement Axis and Direction

In the MPG operation mode, select a axis to be controlled by MPG, press the corresponding button, and then this axis can be moved by MPG.

In the MPG operation mode, if you want to move the X axis by MPG, press , in this case, the X axis can be moved by operating the MPG.

The feed direction is controlled by MPG rotation direction; refer to the manual made by machine tool manufacturer. Generally, MPG CW is positive direction feed, and the CCW is the negative feed.

6.1.3 MPG Feed Explanations

1. The relationshisp between the MPG graduation and the Machine tool movement amount; refer to the following table:

Table 6-1-3-1

	The movement amount of each garaduation on MPG		
MPG increment (mm)	0.001	0.01	0.1
Machine tool movement value (mm)	0.001	0.01	0.1
MPG increment (inch)	0.001	0.01	0.1
Machine tool movement value (inch)	0.0001	0.001	0.01

2. The numerical values from the above-mentioned table are differ from the different mechanical drivings; refer to the User Manual from the machine tool manufactory.
3. When the bit 0 of parameter No.: 56 sets to 1, the MPG movement value selects the absolute operation. The speed for rotating the MPG can not more than the 5r/s, if does, the graduation and movement value may inconsistent.

6.2 Control for MPG Interruption Operation

6.2.1 Operation of MPG Interruption

The operation during the MPG can be overlapped the automatic movement in the Auto operation mode.

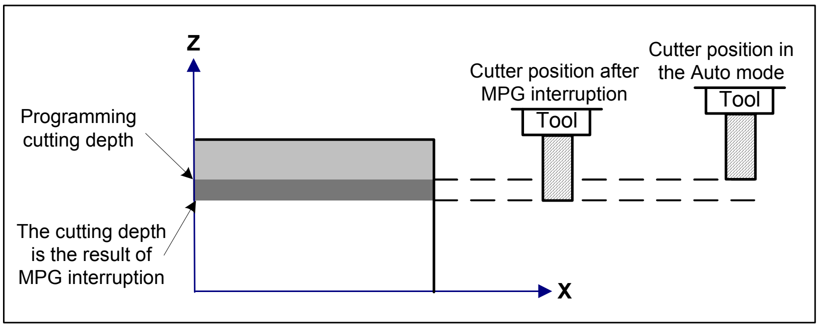


Fig. 6-2-1-1

The operations are shown below:

1) Shift to MPG mode after dwelling during the operation in the Auto mode.

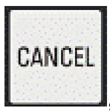
2) Move the tool postion by MPG, for example, performing the up/down movement along Z axis or the translation along X or Y axis, as wll the rotation along A axis can be reached to the destination of the coordinate system modification.

3) Start it after shifting to the Auto mode, and workpiece coordinate holds invariable till to coordinate recovers the actual value after the mechanical zero return is performed.

Note: Set whether using the MPG/Single-step interruption function by bit 3 of parameter No.: 56.

When a program is performed in the Auto, MDI or DNC mode, shift to the MPG mode after feed hold, and then perform the MPG interruption function, refer to the **Fig. 6-2-1-2** for MPG interruption coordinate system .

Fig. 6-2-1-2

The operation steps of MPG interruption coordinate system clear: Move the cursor to MPG interruption coordinate system X flash by the X button, the coordinate system is cleared by 

, and the Y, Z axis shares with the same operation; When the zero return operation is performed, the coordinate system is then automatically cleared.

VOLUME II OPERATION

Note: When the alarm or resetting is generated during the coordinate system is adjusted by using the MPG interruption function, the MPG interruption is then cancelled.

6.2.2 Relationships Between MPG Interruption and Other Fucntions

Table 6-2-2-1

Display	Relationship
Machine lock	MPT interruption movement machine tool is disabled after the machine locking is enabled.
Absolute coordinate value	Absolute coordinate value does not change in MPG interruption
Relative coordinate value	Relative coordinate value does not change in MPG interruption
Machine tool coordinate value	The change value of the machine tool coordinate value is shifting value caused by MPG rotation.

Note: The movement amount of MPG interruption is cleared when manual reference position return along each axis is performed.

6.3 Auxiliary Control During MPG Operation

Its operation method is absolutely same as the Manual, refer to the Section 4.2 and 4.3 in this User Manual.

6.4 Electric MPG Drive Function

The component program operation can be controlled by rotating the MPG with hand, the mechanism is operated along the tool path specified by machining program command, this function is used for workpiece trial cutting and machining program detection.

Operation method:

The electric MPG drive function is enabled by setting the bit 1 of parameter No.: 59. In the Auto mode, open the dry run, each axis of the system will not move after pressing the <CYCLE START> button; in this case, the operation of the component operation can be controlled by rotating the MPG; the faster the MPG rotation is, the faster the speed executed by program is; the slower the MPG rotation is, the slower the speed executed by program is.

Note 1: The dry run is enabled after using the electric MPG drive function.

Note 2: In the single mode, the executiong of the single dwell is enabled.

CHAPTER SEVEN AUTOMATIC OPERATION

7.1 Selection of Auto Operation

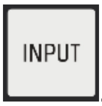
1. Program loading in Auto mode

(a) Enter the Auto operation mode by 

;

(b) Enter the【LIST】page display by 

, find a destination program by moving the cursor;

(c) Confirm it by 

.

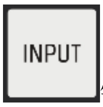
2. Program loading in Edit Mode

(a) Enter the editing operation mode by 

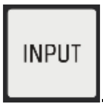
;

(b) Enter the【LIST】page display by 

, find a destination program by moving the cursor;

(c) 按 

键进行确认。

(c) Confirm it by 

.

(d) Enter the Auto operation mode by 

;

7.2 Start of Auto Operation

After the desired program to be started is selected by the two methods in Section 7.1, the program operates in Auto mode by pressing the 

, and then observe the program operation shifting to the <POSITION>, <MONITORING> or <FIGURE> interfaces, etc.

The program operation starts at the line where the cursor locates, and therefore, firstly check

VOLUME II OPERATION

whether the cursor is on the desired line before pressing the , and confirm whether the modal value is correct. If it starts from start line but the cursor does not locate at this line, press the , then the  to carry out the automatic operation program from start line.

Note: In the Auto mode, the workpiece coordinate system and basis offset value can not be modified during the operation program.

7.3 Stop of Auto Operation

During the program Auto operation, the system provides 5 methods to stop the automatic program:

1. Program sto (M00)

After the block with M00 is executed, program operation dwell, and the overall modal information is saved. The program continues after controlling the .

2. Program selection stop (M01)

Before the program operation, if the  is pressed, the program dwells when it executes the block containing the M01, and then the overall modal information is saved. The program continues after controlling the .

3. Press the 

After pressing the  in the Auto operation mode, the machine tool shows the following states:

1) Machine tool feed decelerates then stops;

2) When the dwell (G04 code) is performed, the timer stops then enters the feed hold state;

3) The rest of modal information is saved;

4) The program consecutively performs after pressing 

.

4. Press the 

Refer to the Section 2.3.1 OPERATION EXPLANATION VOLUME.

5. Press the ESP button

Refer to the Section 2.3.1 in this OPERATION EXPLANATION VOLUME.

In addition, when the program is operated in the Auto, DNC, MDI and MDI interface, the machine tool can be stopped after shifting to other modes. Refer to the following items:

1) Shift to the Edit, MDI or DNC interface, the machine tool stops after performing the current block.

2) Shift to the Manual, MPG or Single-step interface, the machine tool immediately stops after the operation is interrupted.

3) Shift to the mechanical zero return interface, the machine tool decelerates then stops.

7.4 Automatic Operation from Any Block

The system supports the automatic operation of any block from current machining program. Refer to the following operation steps:

1. Enter the Manual mode by 

, start the spindle and other miscellaneous function;

2. Operate each modal value of the program in the MDI mode, it is necessary to guarantee the modal value is correct;

3. Enter the Edit operation mode by 

; enter the program page display by 

, and the find the desired program to be machined in【LIST】.

4. Open the program, move the cursor before the block to be operated;

5. Enter the Auto operation mode by 

;

6. Automatically operate the program by 

.

Note 1: Confirm the current coordinate point is the previous block operation end positon (The current coordinate point need not to be confirmed if the operated block is absolute programming and the G00/G01 movement) on its operation block.

Note 2: For example, if the this operated block is tool-change movement, firstly conform the current position will not impact with the workpiece, etc.; to avoid the machine tool damage and personal injury.

7.5 Dry Run

The program can be inspected by “Dry run” before the program is machined; generally, it is used matching with the “M.S.T Lock” or “Machine Lock”. Enter the Auto operation mode by 

, and then press the 

(The indicator-ON on the button means the machine enters the dry run state).

In the rapid feed, the program speed = dry run speed x rapid feedrate.

In the cutting feed, the program speed = dry run speed x cutting feedrate.

Note 1: Dry run speed is set by data parameter P86;

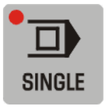
Note 2: Whether the dry run is enabled is set by bit 5 of parameter No.:12 during tapping.

Note 3: Whether the dry run is enabled is set by bit 6 of parameter No.:12 when the cutting feed is performed.

Note 4: Whether the dry run is enabled is set by bit 7 of parameter No.:12 in the rapid positioning.

7.6 Single Block Operation

If you want to check the operation circumstance of the single bock, the “Program single block” operation can be selected.

Press the 

button (The indicator-ON on the button means that the single operation state is already performed) in Auto, DNC or MDI mode. When the single block is operated, the system stops operation after performing one block, and then continue performing the next block by 

, till the program operation is completed.

Note: During the G28, also, the single block can be stopped on the intermediate point.

7.7 Machine Locking Operation

Press the 

button (The indicator-ON on the button means that the single operation state is already performed) in the <AUTO> operation mode. In this case, each axis of machine tool does not move, but the display of the position coordinate is identical with the machine movement, as well the M, S or T can be performed; this function is used for program verification.

7.8 M.S.T Function Locking Operation

Press the 

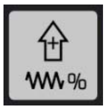
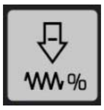
button (The indicator-ON on the button means that the single operation state is already performed) in the <AUTO> operation mode. In this case, M, S or T code does not perform, which is used for the program verification with the machine tool locking function together.

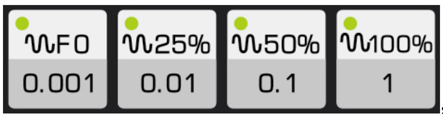
Note: M00, M01, M02, M30, M98 and M99 can be performed normally.

7.9 Feed, Rapid Trimming in Auto Operation

In the <AUTO>operation, the system can be changed the movement velocity by trimming feed or rapid traverse rate.

Select the feedrate by 

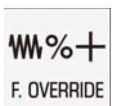
in the Auto mode, its feed override can be carried out 21-level real-time adjustment. Press  once, the feedrate increases one level, each level is 10%, it will not increase any more till to 200%; the feedrate decrease one level by  once, each level is 10%; the bit 4 of parameter No.: 12 set whether the axis is stopped if the override regards to Fo; if it sets to 0 without stopping, the actual rapid traverse rate is set by parameter P93 (General-purpose for overall axes).

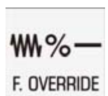
In the Auto mode, select the rapid traverse rate by 

, the rapid override can be carried out four-step adjustment: Fo, 25％, 50％ and 100％.

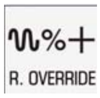
GSK218MC-H and GSK218MC-V CNC System can be selected the feedrate by its feed override wave band switch 

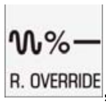
, and the feed override can be carried out 21-level real-time adjustment.

GSK218MC-U1 is performed the feed override trimming by 

and 

, and

performed the rapid traverse override for trimming by 

and 

, its operations are same as the above-mentioned.

Note 1: The value set by F in feed override trimming program

Actua feedrate = The value set by F x feed override

Note 2: The calculation of the rapid traverse speed value gained at the eventually trim by data parameter P88, P89 and P92 and the rapid override:

X axis actual rapid traverse speed = The value set by P88 x rapid override

The calculation method of the actual rapid traverse rate of Y and Z axes are shown as the above-mentioned.

7.10. Spindle Speed Trimming in Auto Mode

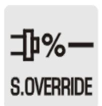
In the Auto mode, when the spindle speed controls by selecting the analog value, the spindle then can be performed the trimming.

In the Auto mode, change the spindle speed by adjusting the spindle override based upon the , 

and the spindle override can be carried out **50**％～**120**％ real-time adjustments, total 8 levels.

The revolving override increases one level by pressing 

once, each level is 10%, it will not increase any more till to 120%.

The revolving override decreases one level by pressing 

once, each level is 10%, the spindle speed stops till to 50%.

GSK218MC-H/-V CNC System can be changed the spindle speed by adjusting the spindle override by spindle override wave band switch 

, and the spindle override can be carried out the feed override can be carried out **50**％～**120**％ real-time adjustments, total 8-level.

The actual speed of spindle = program command speed x spindle override. The top spindle speed is determined by data parameter **P258**. It will be rotated based upon this speed if it exceeds this numerical value.

7.11 Background Editing in Auto Mode

System supports the background editing function during the machining procedure.

In the Auto mode, enter the program display interface in <PROGRAM> during the program operation, and then press the【PROGRAM】softkey, then enter the background editing interface; refer to the Fig. **7-11-1:**

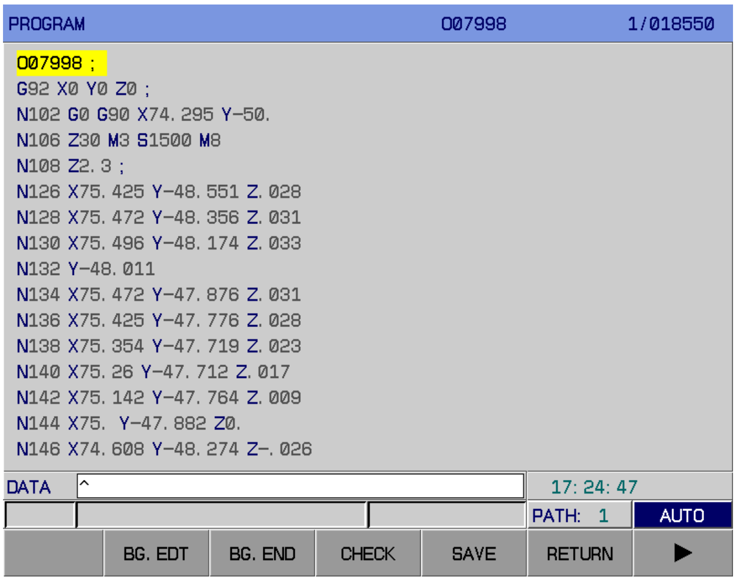


Fig. 7-11-1

Enter the program background editing interface by【B. EDITING】softkey, the editing of the program is indentcial under the editing mode. Refer to the *CHAPTER TEN PROGRAM EDITING OPERATION* of the *OPERATION MANUAL*; save and retreat from this interface by【B. EDITING】editing program.

Note 1: It is suggest that the background editing file size is less than 3000 lines; otherwise, the machining result may be affected.

Note 2: The background editing can be opened the foreground program, instead of editing or delecting the foreground program.

Note 3: The background editing can not compile the operating foreground program.

CHAPTER EIGHT MDI OPERATION

System can be both performed the MDI, parameter modification or offset, etc. and offered the MDI operation function in the MDI mode, and it can be operated by directly inputting the code based upon this function. The MDI data, parameter modification or offset, etc describes in the Interface Display and Data Modification and Setting in CHAPTER THREE, as well, the MDI operation function is also introduced in this chapter.

8.1 MDI Code Input

There are two input methods in MDI mode:

1. 【MDI】can be consecutively input multiple-block program;

2. **【CUR/MOD】**can be input only one program.

In the【MDI】mode, the program input is same as the same editing mode, refer to the PROGRAM EDITING OPERATION in CHAPTER TEN; The following will introduce the input in the【CUR/MOD】.

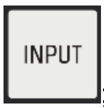
For example: Input a block G00 X50 Y100 from the【CUR/MOD】operation page, the operation steps are shown below:

1) Enter the MDI operation mode by 

;

2) Enter the program interface by 

, then enter the【CUR/MOD】operation page by the【CUR/MOD】softkey (refer to Fig. **8-1-1**) :

3) After inputting the G00X50Y100 on keyboard in turn, confirm it by 

; in this case, you can view that the program is already input to the interface.

Refer to the following figure (Fig. **8-1-1**):

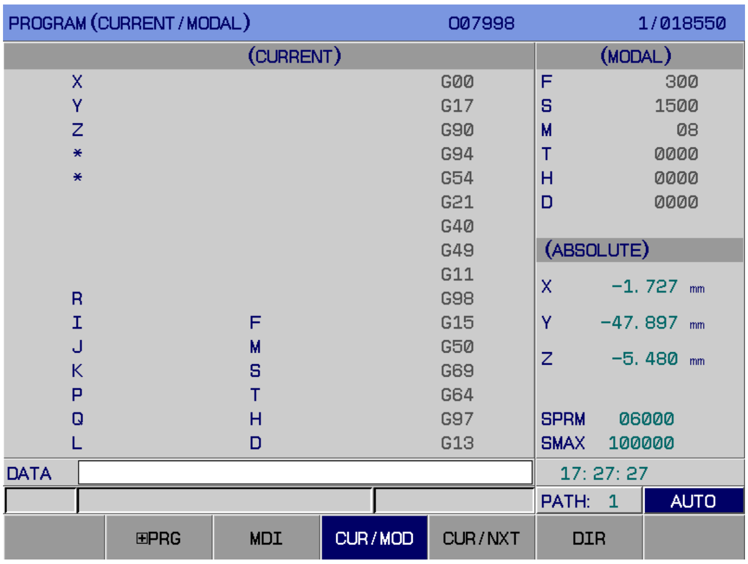


Fig. 8-1-1

8.2 Operation and Stop of MDI Code Block

After the code block is inputted based upon the Section 8.1, the MDI can be operated by 

.

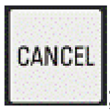
The code block operation can be stopped by 

during the operation.

Note 1: The operation of MDI must be performed in MDI operation method!

Note 2: When the operation program in the MDI and CUR/MOD interface is performed in MDI mode, firstly treat the inputted program in the CUR/MOD interface.

8.3 Modification and Clear of Filed Value in MDI Code Block

If the error occurs during the field input, cancel the input by 

; if the error occurs after the input is performed, input again the correct content to replace the error one or clear the overall inputted contents by 

, and then input again.

8.4 Conversion of Each Operation Mode

In the Auto, MDI or DNC mode, some programs are converted to MDI, DNC, Auto or Editing mode during the operation; the system stops the program after performing the current block.

In the Auto, MDI or DNC mode, the program converts to the single-step mode after dwells, the single-stop function is then performed, refer to the single-step interruption in Section 5.2 of the *OPERATION MANUAL*. Shift to MPG mode after dwells, and then the MPG interruption function is performed, refer to the Manual Interruption in Section 6.2 of the *OPERATION MANUAL*. Shift to MPG mode after dwells, and then the manual intervention function is performed, refer to the Manual Intervention Function in Section 4.1.4 of the *OPERATION MANUAL*.

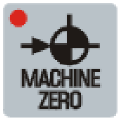
In the Auto, MDI or DNC mode, when the program is directly shifted to the single-step, MPG, Manual, Zero-return mode during the operation, and the program is then stopped after decelerates.

CHAPTER NINE ZERO RETURN OPERATION

9.1 Concept of Machine Tool Zero (Mechanical Zero)

The machine tool coordinate system is the inherent one for a machine, its origin is called the mechanical zero (or machine tool zero), also, it regards as **Reference Point** in this Manual which described the mechanical origin by machine tool manufacturer; usually, it installs at the top traversal along X, Y, Z, as the 4th and 5th axes positively. The mechanical zero point does not execute when the CNC equipment is power on, it is better to perform the Auto or Manual mechanical zero return.

9.2 Operation Steps for Pulse Servo Mechanical Zero Return

1. Enter the mechanical zero return operation method by 

, in this case, the "Mechanical zero return" displays at the right corcer on LCD screen.

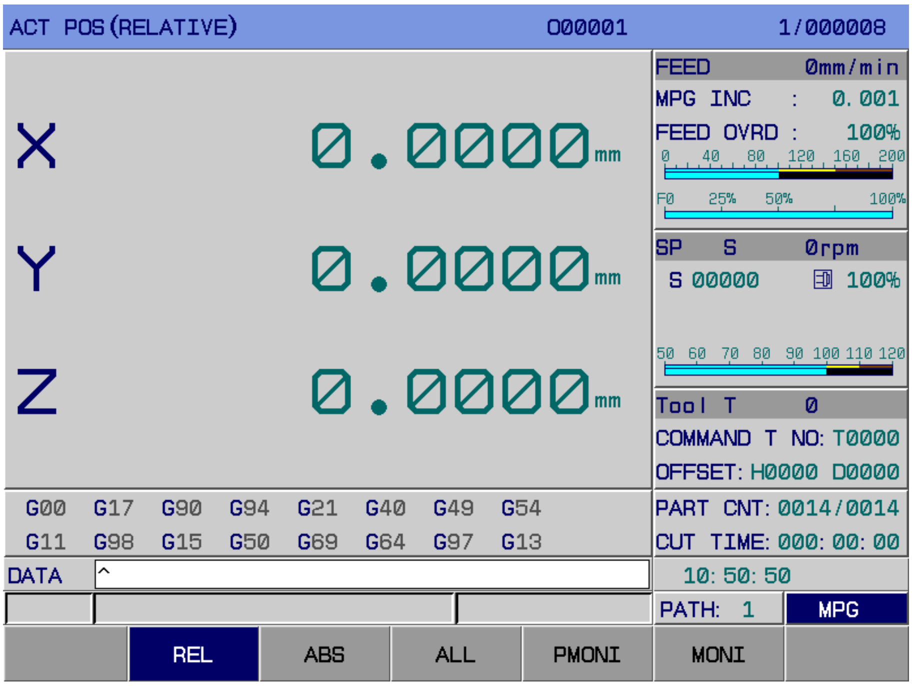


Fig. 9-2-1

2. Select the X, Y, Z, 4th or 5th axis to be returned the mechanical zero, the zero return direction is determined by **bit 0~4 of parameter No.7**.

3. Machine moves along with the mechanical zero point. The machine moves (The movement

speed is determined by data parameter **P100～P104 P100～P104.**) rapidly before the deceleration point. The data parameters **P342~P346** are set the zero return speed along each axis after touching the deceleration switch. The speed of FL (It determines by data parameter **P099**) moves to mechanical zero (that is, reference point) after departing from the stopper. The coordinate axis stops to move and zero return indicator lights on after returning to the mechanical poont.

For example:

The 1st axis common incremental zero return is regarded as an example; the 1st axis begins to impact the stopper with the higher speed F4000 (Data parameter **P100** sets as 4000), and then pass the stopper based upon the F500 (Data parameter **P342** sets to 500) after touching the deceleration switch, then slowly search the one-turn Z pulse signal of servo with F40 (Data parameter **P99** sets to 40) after departing from the stopper, lastly, it immediately stops after gaining, refer to the Fig. 9-2-2.

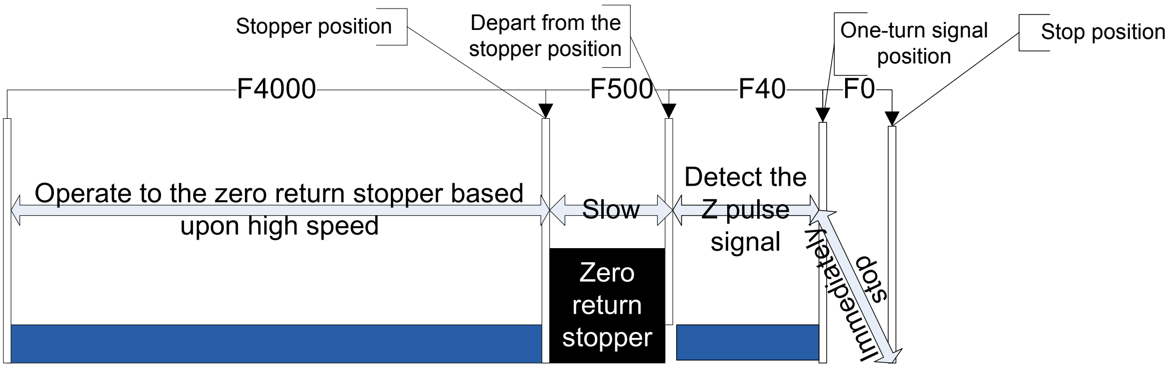


Fig. 9-2-2

9.3 Operation Steps for Mechanical Zero Return Specified by Program

A G28 zero turn can be specified by program after bit 3 of parameter No.: 6 sets to 0, because the detection of stroke stopper is shared the same effect with the manual mechanical zero turn.

9.4 Bus Servo Zero-Return Function Setting

There are three kinds of zero return methods when the system configures a bus servo, which are separately common, high speed and multi-core absolute setting zero. We will separately introduce the following setting methods.

9.4.1 Common Zero Turn

Set the bit parameter **No：0#0=1** and **No：5#4=0,** the system is performed the zero return based upon the common one, which can be selected the zero return method such as the one-turn signal or one-shot signal, and this zero return method can be used in the version of the system configuration DA98B, GE2000 series increment method. Zero return along each axis is enabled in the zero return mode.

The concrete operation steps are basically identical with the pulse servo zero turen; refer to the *Section 9.2 Operation Steps of Pulse Mechanical Zero Return*.

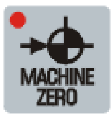
9.4.2 High Speed Increment Zero Return

Set the bit parameter **No：0#0=1** and **No：5#4=0,** the system is performed the zero return based upon the high speed one, which can only be selected the one-turn signal zero return method, and this method can be used in the version of the system configuration GE2000 series increment method.

Each axis zero valid in zero mode

Set the bit parameter No：20#7=0、No：20#6=1、No：20#5=0 to configure the GE2000 series single-core absolute, multi-core absolute version. The setting of parametr value of the data parameter **P347~P351** can be modified the single-core zero signal postion of the absolute encoder; Zero turn along each axis is enabled in the zero return method.

Zero return steps:

1. Enter the mechanical zero return operation method by 

, in this case, the "Mechanical zero return" displays at the right corner of the LCD.

2. Select the X, Y, Z, 4th or 5th axis of the desired mechanical zero return, its direction of zero return is set by bit parameters **N0:7#0～N0:7#4.**

3. Machine tool moves along with the mechanical zero direction at the rapid traverse rate (its movement velocity is set by data parameter **P100～P104**) before the deceleration point; set the zero return speed along each axis is set by data parameter **P342~P346** after touching the deceleration switch; Inquire the Z pulse one-turen signal position based upon the speed of the data parameter **P342~P346** after departing from the stopper, it decelerates and stops after detecting, and then return to the mechanical zero (that is the reference point) based upon the speed of the data parameter **P354**. The coordinate axis is stopped moving when returning to the mechanical zero, and the zero indicator is turned on.

For example:

The 1st axis high-speed incremental zero return is regarded as an example; the 1st axis begins to impact the stopper with the higher speed F4000 (Data parameter **P100** sets as 4000), and then pass the stopper based upon the F500 (Data parameter **P342** sets to 500) after touching the deceleration switch; search the one-turn Z pulse signal position of the servo based upon the F5000 speed after departing from the stopper, it decelerates then stops after detecting, and then return to the mechanical zero based on the F200 (The data parameter **P354** sets to 200) speed, refer to the Fig. 9-4-2-1.

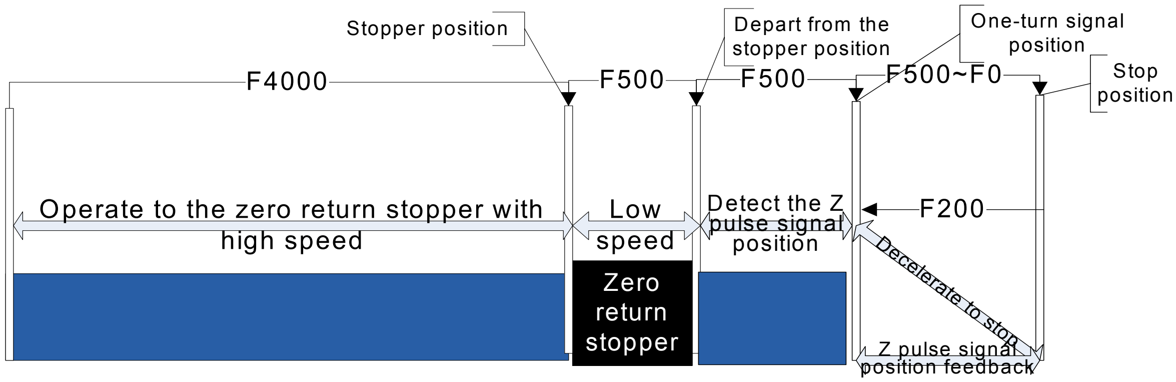


Fig. 9-4-2-1

9.4.3 Multi-Core Absolute Zero Setting

Set the bit parameters **No：0#0=1**, **No：20#7=1, No：20#6=1 and No：20#5=1,** configure the GE2000 series multi-core absolute version, manually move each axis to the granted position of the machine tool zero, then set the 1st axis zero position when bit parameter **No：21#0=1, the 2nd axis** zero position when **No: 21#4=1**, the 3rd axis zero position when **No: 21#2=1**, the 4th axis zero position when **No: 21#3=1** and the 5th axis zero position when **No: 21#4=1** in the MDI mode. In the zero return mode, if the zero return indicator is turned on, the machine zero point setting is successful.

Note: It is hard to set the zero by this method, it is more convenient to set on the bus configuration interface.

This zero return mode can be directly set on the【⊞ BUS CONFIGURATION】interface, refer to the Display, Modification and Setting of Bus Servo Parameter in Section 3.3.5 for details.

For example:

The absolute encoder setting zero can be set the zero position based upon the absolute position from the motor feedback. Set the bit parameter #20.7=1, #20.6=1, #20.5=1. Refer to the Fig. 9-4-3-1.

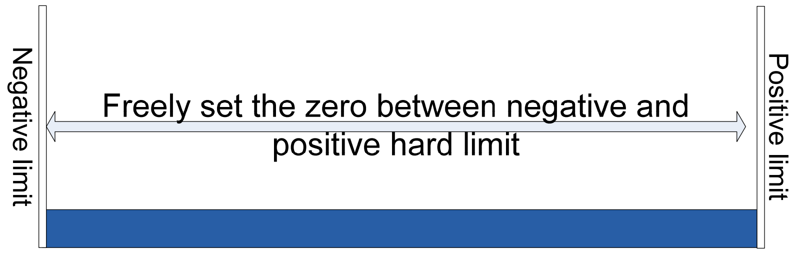


Fig. 9-4-3-1 Absolute encoder zero setting

Note 1: Never attempt to operate the mechanical zero return, if your machine tool does not install the zero return deceleration switch or set the mechanical zero.

Note 2: The indicator of corresponding axis is turned on when the mechanical zero return is ended.

Note 3: The zero return indicator is turned off when the corresponding axis does not at the mechanical zero point.

Note 4: Refer to the machine tool User Manual made by the manufacturer for the mechanical zero (reference point) direction.

Note 5: Do not modify the zero return direction, feed axis direction and gear ratio dimension along each axis after the mechnical zero is set,

Note 6: Refer to the Section 4.8 for *CHAPTER FOUR INSTALLATION & CONNECTION* in *PLC & INSTALLATON CONNECTION VOLUMN* for the relevant parameter of mechanical zero return and all kinds of methods of mechanical zero return.

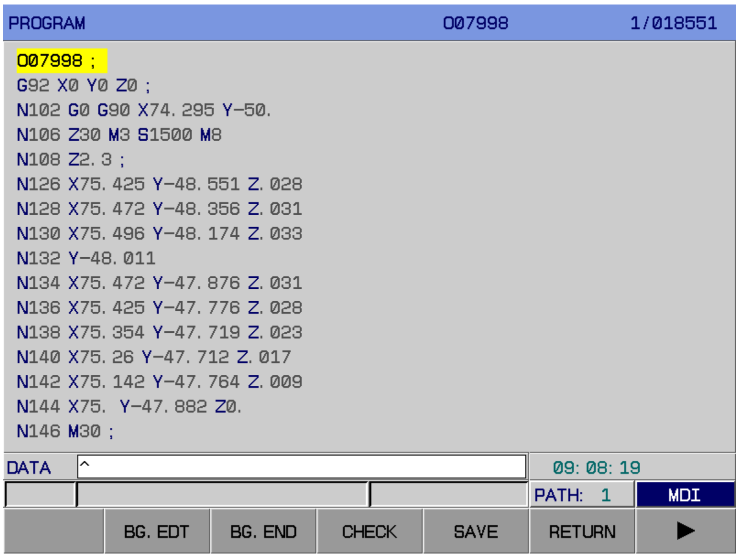
CHAPTER TEN EDIT OPERATION

10.1 Edit of Program

The editing of the component program should be performed in the Edit operation mode. Enter the editing operation method by 

; enter the program interface by 

on the panel; then enter the editing and modification interface after pressing the【⊞ PROGRAM】softkey (Refer to the Fig. **10-1-1**):



Press【▶】to enter the next page 

Press【▶】to enter the next page 

Pres【◀】to enter the next page 

Fig. 10-1-1

Each operation, such as the replacement, cutting, copy, paste and restart, etc. can be performed by pressing the corresponding softkey.

Before the program is compiled, the editing operation can only be performed by opening the program switch. Refer to the **Section 3.4.1** for details.

Note 1: The top lines of single program file are 100 thousand (lines).

Note 2: Refer to the Fig. 10-1-1, when the initial symbol "/" of the block is more than 1, the system still skips this block even if the skip function does not start.

Note 3: The other methods can not be shifted during the correction is performed in the Auto mode; otherwise, the unexpected circumstance may occur.

Perform the correction function in Auto mode, when the "/" symbol locates at the beginning of the block, the program followed with the "/" will perform the correction function regardless of whether the skip function is started or not.

10.1.1 Establishment of Program

10.1.1.1 Automatic Generation of Sequence Number

The "Automatic sequence number" sets to 1 based upon the Section 3.4.1 in OPERATION (Refer to the Fig. **10-1-1-1-1**).

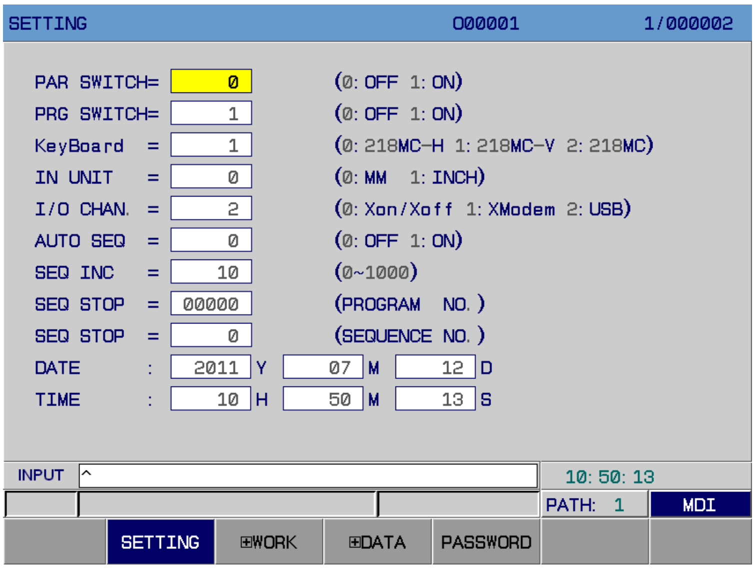


Fig. 10-1-1-1-1

In this case, the system will automatically insert a sequence number among blocks when the program is edited, and the number incremental value of the sequence number can be set in the series number incremental value.

10.1.1.2 Input of Program Content

1. Enter the Editing operation method by 

;

2. Enter the program page display by 

. (Refer to the Fig. **10-1-1-2-1**):

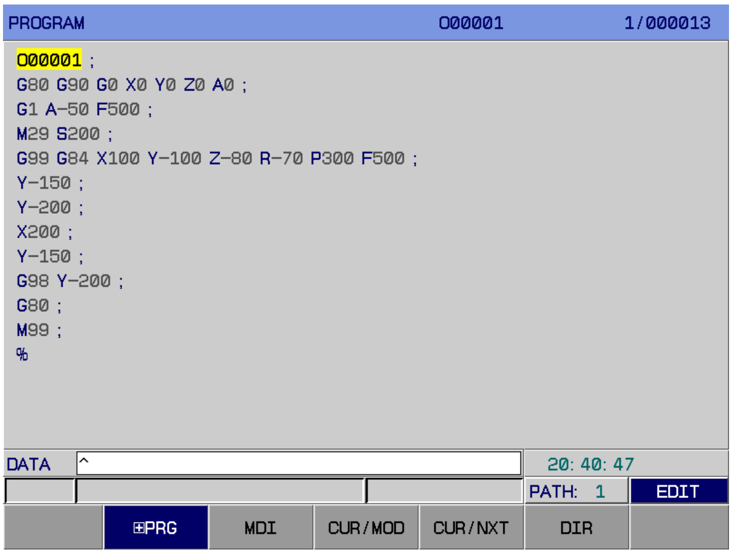


Fig. 10-1-1-2-1

3. Input the number buttons , , ,  and  (For example, the establishment of the O00002 program name) in turn by the address button , display the O00002 followed with the data column, refer to the following Fig. **10-1-1-2-2:**

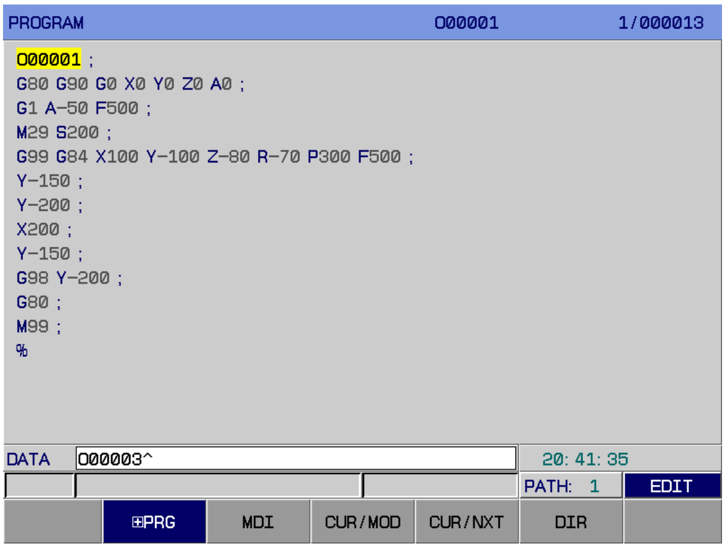


Fig. 10-1-1-2-2

4. Establish a new program name by , refer to the following Fig. **10-1-1-2-3:**

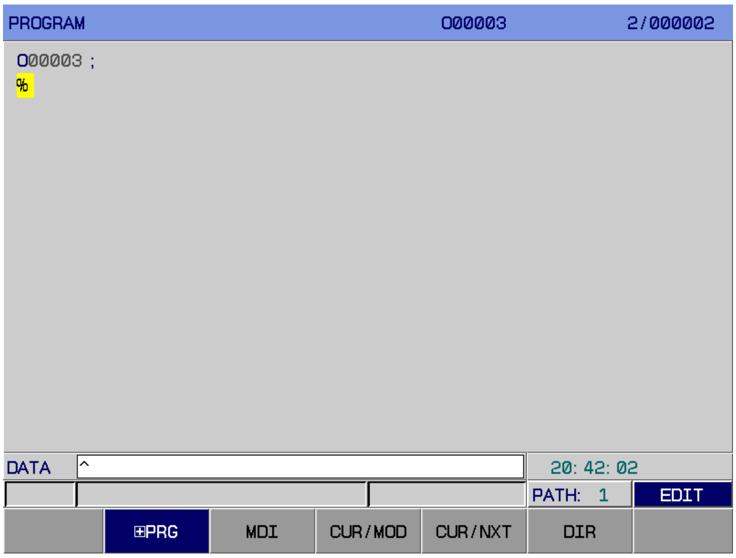


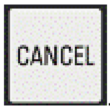
Fig. 10-1-1-2-3

5. Input the compiled program step by step, simultaneously, the program is automatically stored after the complete to shift other working methods; if you want to shift other interfaces (such as the 

), firstly press the 

to save, and then complete the program input.

Note 1: In the Editing method, the system is temporarily not supported the single number.

Note 2: If you find the code word is incorrect when the program is input, cancel the input code by 

.

Note 3: Up to 65 characters for the single input of the block.

VOLUME II OPERATION

10.1.1.3 Index of Sequence Number, Word and Line Number

Sequence number index is a sequence number within the index program. Generally, the program can be performed or edited from the beginning of this sequence number. The skipped block due to index does not affect to the CNC state. (The coordinate value, M, S, T code, G code, etc. of the coordinate value in the skipped block does not affect to the coordinate value of CNC and the modal value.)

If some one block from index program begins performing, in this case, the machine state, CNC state should be checked. It can be operated when the corresponding M, S, T code and the setting of the coordinate system, etc. are consistent.

The index of word, generally, uses the specified address word or numbers during the index for the program editing.

The steps of sequence number, word and line number during the index program:

1. Selection method: <EDIT> or <AUTO> method.
2. Find the object program in【LIST】.

3. Enter to the object program by INPUT.

4. Input the word or sequence number to be indexed, and find it by the ↑ or ↓ button.

5. If the line number in the program should be searched, you can input the desired line number and then confirm it by SEARCH button.

Note 1: The index function is automatically cancelled when the sequence number or word indexes to the end of the program.

Note 2: The sequence number, word and line number can be indexed in the【AUTO】and【EDIT】methods; however, it can only be performed at the background editing interface in【AUTO】mode.

10.1.1.4 Positioning method of cursor

Select the editing method, press the PROGRAM, display program screen.

a) Move the cursor one line upward by ↑, if the line of the cursor locates is more than the end line of the previous one, the cursor then moves to the end of the last line.

b) Move the cursor one line downward by ↓, if the line of the cursor locates is more than the end line of the next one, the cursor then moves to the end of the next line.

c) Move the cursor one line rightward by →, if the cursor at the end line can be moved at the start of the next line.

d) Move the cursor one line rightward by ←, if the cursor at the end start can be moved at the end of the last line.

e) Scroll the screen upward by , cusor the moves to the last screen.

f) Scroll the screen downward by , cusor the moves to the next screen.

g) Cursor moves to the beginning of its line by HOME.

h) Cursor returns to the start of the program by SHIFT + HOME.

i) Cursor moves to the end line of its line by END.

j) Cursor moves to the end of the program by SHIFT + END.

10.1.1.5 Insertion, Deletion and Modification of Word

Select the <EDIT>mode, press the PROGRAM, and then display the program screen, lastly position the cursor at the position to be edited.

1. The insertion of a word

After the data is input, press the INSERT, the system may insert the inputted content at the left fo the cursor;

2. The deletion of a word

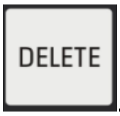
Position the cursor to the place to be deleted, press the DELETE, the system may delete the content where the cursor locates.

3. The modification of a word

Move the cursor to the place to be modified, and then input the modified content, then press the ALTER, the system replaces the positioned content of the cursor inputted one.

10.1.1.6 Deletion of Single Block

Select the <EDIT> mode, press the PROGRAM, and then enter the program screen, then move the cursor to the initial line of the block to be deleted, lastly delete the cursor block by 

+ 

.

Note: The 

can be input to delete (The cursor should be placed at the first line) the block regardless whether the block is with or without the sequence number,

10.1.1.7 Deletion of Multi-Block

Delete to the block of the specified sequence number from the beginning of the current displayed word.

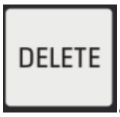
N100 X100.0 M03 S2000; ..... N2233 S02 ; N 2300 M30 ;

Cursor current position

Delete this area

Fig. 10-1-1-7-1

In the <**EDIT**> mode, enter the program display screen by 

, position the cursor at the object start position (Refer to the above-mentioned character N100) to be deleted; and the input the last complete character in the multi-block to be cleared; for example, the **S02** (Refer to the Fig. **10-1-1-7-1),** and then press the 

. The program between the cursor and address mark can be deleted.

Note 1: Up to 100 thousand lines of the block deletion.

Note 2: If there are several same completed characters should be deleted in program, delete the program between the completed character and cursor character with the search sequence downward.

Note 3: When multiple blocks are deleted with N+ sequence number, the N+ sequence number start position of destination deletion should be located at the initial line of this block.

10.1.1.8 Deletion of Multiple Code Word

Delete to the specified code word from current displayed one.

VOLUME II OPERATION

N100 X100.0 M03 S2000; G01 X50.0 Y100.0 N2233 S02 ;

Cursor current position

Delete this area

Fig. 10-1-1-8-1

In the <**EDIT**> mode, enter the program display screen by 

, position the cursor at the object start position (Refer to the above-mentioned character N100) to be deleted; and the input the last complete character in the multi-block to be cleared; for example, the **Y100.0** (Refer to the Fig. **10-1-1-8-1),** and then press the 

. The program between the cursor and address mark can be deleted.

Note 1: If the N+ sequence number lies among the blocks, the system regards that it is the code word treatement.

10.1.2 Deletion of Single Block

When some one program in the memory should be deleted, refer to the following steps:

a) Select <EDIT>operation method;

b) Enter the program display page, there are two methods to delete the program:

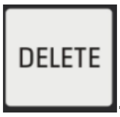
1. Input the address 

; input the program name (input the numerical button 

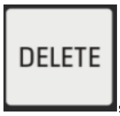
, 

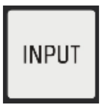
, 

or 

, in this case, the O0002 program is regarded as an example); the program in the memory is deleted by 

.

2. Select the【LIST】interface in program interface, select the program name to be deleted by cursor, the system state column prompts "Confirm deleting the current file?" by 

, and then the state column shows "Deletion Successful" by 

again, lastly, the selected program by cursor can be deleted.

Note 1: If only one program file is performed, under the editing mode program (list) interface, the program name will become O00001 and the program content is deleted regardless of the program name is O00001 or not; when the multiple program files are performed, the program content and name of O00001 will be deleted together.

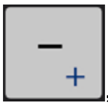
10.1.3 Deletion of Overall Programs

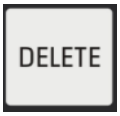
The overall programs in the memory should be deleted, refer to the following steps:

a) Select the <EDIT>operation mode;

b) Enter the program display screen;

c) Input the address ;

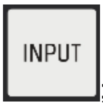
d) Input the address , , , ,  and  in turn;

e) The overall programs in the memory are deleted by .

10.1.4 Program Copy

The current program copies and saves as a new program name:

a) Select the <EDIT>mode

b) Enter the program display page; selet the desired program to be copied by cursor in【LIST】 interface, enter the program display interface by ;

c) Press address button , and then input a new program name;

d) The file is copied by【COPY】software, and then enter a new program editing interface.

e) A copied new program name can be viewed by returning to the【LIST】.

The copy of the program can also be performed at the program editing page (Refer to the Fig. 10-1-1):

1. Press the address button , and then input a new program number;

2. The file is copied by【COPY】softkey, and then enter the new program editing interface.

3. A copied new program name can be viewed by returning to the【LIST】.

10.1.5 Copy and Paste of Block

The operation steps for the copy and paste of blocks are shown below:

a) Cursor moves to the start of the block to be copied.

b) Input the last character of the block to be copied.

c) Press SHIFT + SAVE buttons, the program between the cursor and character input are copied.

d) Move the cursor to the position to be pasted, press the SHIFT + INSERT buttons or 【PASTE】softkey, the paste is then completed.

The copy and paste of the block also can be performed in the program editing page (Refer to the Fig. 10-1-1):

1. Cusor moves to the start of the block to be copied.

2. Input the last character of the block to be copied.

3. Press the【COPY】softkey, the program copy is completed between the cursor and character input.

4. Cursor moves to the desired paste position, the paste is then completed by【PASTE】softkey.

Note 1: If there are several same completed characters should be copied in program, copy the program between the completed character and cursor character with the search sequence downward.

Note 2: If the copy method is performed by N+ sequence number in program, the program of line is copied between the cursor start and N+ sequence number. The N+ sequence number should be located at the beginning of the block, the copy is unsuccessful in other places.

Note 3: Up to 100 thousand lines of the block copy.

10.1.6 Cut and Paste of Block

The operation steps of block cut:

a) Enter the program editing page (Refer to Fig. 10-1-1).

b) Cusor moves to the start of the block to be cut.

c) Input the last character of the block to be cut.

d) Press【CUT】softkey, program is cut to the paste

e) Cursor moves to the position to be pasted, the paste is then completed by pressing the【PASTE】softkey.

Note 1: If there are several same completed characters should be cut in program, cut the program between the completed character and cursor character with the search sequence downward.

Note 2: If the cut method is performed by N+ sequence number in program, the program of line is cut between the cursor start and N+ sequence number.

Note 3: When the program name shares a same block with the program content in the editing method program interface, the character followed with the program name can be performed the copy operation for the system instead of cutting operation.

10.1.7 Replacement of Block

The operation steps of block replacement:

a) Enter the program editing page (Refer to the Fig. 10-1-1).

b) Cursor moves to the character to be replaced.

c) Input the replaced content.

d) Press the【REPLACEMENT】softkey, the system replaces the content positioned by cursor and the overall same contents in block as the one from input.

Note: This operation is only performed for character instead of executing the integrated block.

10.1.8 Rename of Program

The current program name changes into another name:

a) Select <EDIT>operation method;

b) Enter the program display interface (Cursor specifies the program name);

c) Input the address 

, and then input a new program name;

d) Press 

, the file name is then completed.

10.1.9 Program Restart

This function is used for the program operations when the accident of the automatical motion generates, such as the tool broren, power-off, ESP, rest, etc. The system returns to the program breakpoint and performs continuely based upon the reatart function after the accident is eliminated.

The operation steps of the program restart:

1. Resolve the machine tool accident. For example, tool-change, offset alternation and mechincial zero turen, etc.

2. In the <AUTO>mode, press the 

button on the panel.

3. Enter the program interface by the 

on the operation panel, and then enter the sub-menu by the【PROGRAM】software below the LCD screen, page to the last one of the

VOLUME II OPERATION

sub-menu by pressing【▶】twice, enter the program restart interface by pressing the【RESTART】. Record the different codes between the current modal and preload one. (Refer to the Fig. **10-1-9-1**).

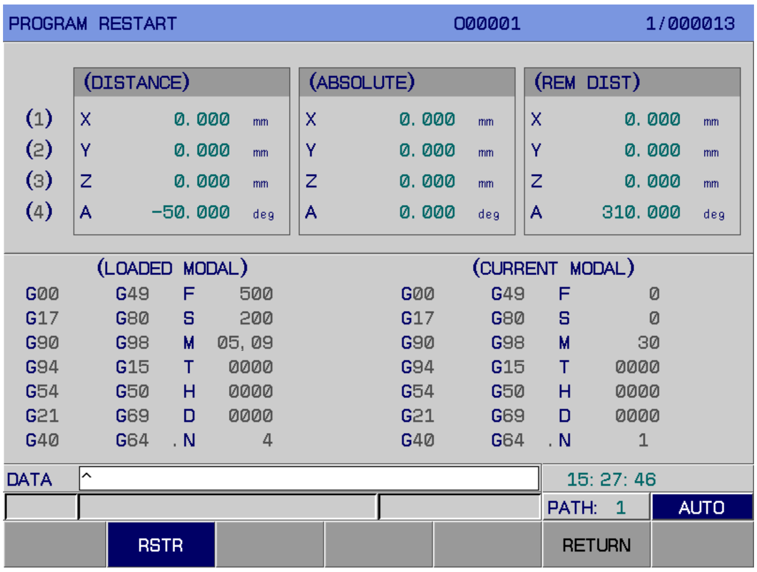


Fig. 10-1-9-1

4. Shift to <MDI>mode, enter the CUR/MOD interface by pressing the【CUR/MOD】softkey, input the corresponding modal code and M code based upon the preload modal value in the Fig. 10-1-9-1.

5. In the mode of returning the <AUTO>, press the 

on the panel, and then press the 

, the program moves to the start (That is, the intervention point of the previous block) of the interruption block according to the sequence before the coordinates (1) (2) and (3) based upon the dry run speed, and the machining is restarted again.

Explanations:

1. The (1) (2) and (3) before the coordinate syste is movement sequence of which each axis moves to the restart position of the program, and its sequence is determined by data parameter **P376**.

2. The single block is switched on when the coordinate system restarts the position movement, the tool may stops after completing a axis direction movement. The intervention can not be performed by shifting to the MDI mode during the execution.

3. The movement method along Z axis can be controlled by bit 0 of parameter No.: 49. (0：G00, 1：G01)

Note 1: The operation can be performed at an arbitary position, therefore, detect whether the tool will impact with the workpiece or other object when moving to the program restart position, if does, the program restart can only be performed after the tool moves to the places without any abstruction.

Note 2: The block of the program restart maybe not be interrupted in the halfway, the operation can be be restarted from any block; its method is identical with the above-mentioned. It is only different that the N line number of the preload modal value is directly defined by the direction button "↓" in the "MDI" mode at the 4th step, and then confirm it by "INPUT" button. You can input the corresponding modal code and M code after entering the CUR/MOD interface again.

Note 3: Never attempt to perform the restart when the restart block indexes to the execution period of program restart; otherwise, the program start should be performed again at the 1st step.

Note 4: The reference position return should be performed before performing the restart after the power is turned on, if the machine tool does not install the absolute position detector (absolute encoder).

Note 5: The program restart function does not support the form with the sub-program;

Note 6: The program restart function does not support the programs with the rotation, mirror image, scaling or polar coordinate modal.

Note 7: The program restart function does not support the canned cycles programs;

Note 8: The program restart function does not support the DNC on-line machining program.

Note 9: The program restart function does not support macro programs (Including Type A, B).

10.2 Program Administration

10.2.1 Index of Program List

Press the 

buttin entering the program list display interface by controlling the【LIST】softkey in the program interface (Refer to the Fig. **10-2-1-1):**

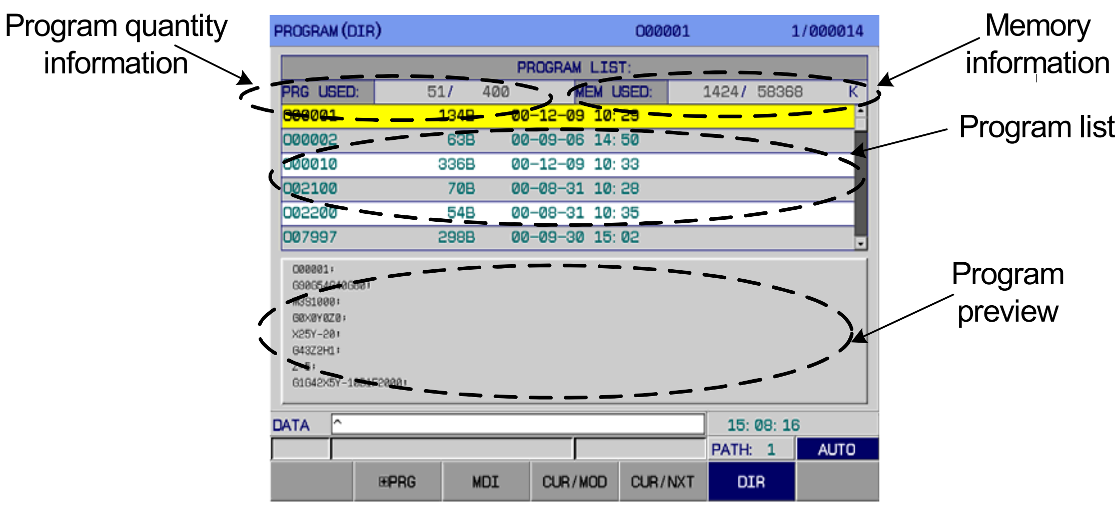


Fig. 10-2-1-1

1) Open the program

Open a specified program: O + series number + input (or EOB) or series number + input (or

EOB)

A new program will be established if the input series number is null in the editing mode.

2) Delete the program: 1. Editing method Delete the specified program by DEL.

2. Edit mode O + series number + DEL or Series number + DEL.

10.2.2 Quatity of Storage Program

Up to 400 can be stored at the system program, the sotred quantity can be view the Section 10.2.1 for the *Program Number Information on the Program List Display Interface*.

10.2.3 Storage Capacity

Refer to the *Program Number Information on the Program List Display Interface* in Section 10.2.1 for the concrete storage capacity.

10.2.4 Check of Program List

Up to 6 CNC program names can be displayed at the program list display page once, if it is more than 6, the display can not be performed within a page; in this case, you can use the page button to display it. In succession, the LCD will display the CNC program name at the next page. LCD will repeatedly show the overall CNC program names if you control the page button again and again.

10.2.5 Locking of Program

This system sets a program switch to avoid that the user program is being modified or deleted by others. After the program is edited, the program can be locked by closing its switch, and the user can not perform the program compilation any more, refer to the Section **3.4.1** for details.

CHAPTER ELEVEN SYSTEM COMMUNICATION

The system can be communicated with the PC terminal or U disk by its interface to carry out the data transmission or DNC on-line machining.

11.1 Serial Port Communication

Serial port communication preparation work:

1. The computer port (COM port) is connected with the RS232 of the system by serial cable.
2. Open the PC terminal GSK Com serial port communication software.

Note: GSK Com serial port communication software is Windows interface, which is suitable for the Win98, WinMe, WinXP and Win2000.

3. The setting of the GSK Com serial port communication software:

(1) Select the available GSK218M (The transmission data error may occur if other system types are selected);

(2) Click the "Series port" menu, set the Baud rate in the "Series Setting" dialog frame, and the Baud rate selects 115200 (It is corresponding with the default value of the data parameter P002) when the data is transmitted; the Baud rate selects 38400 (It is corresponding with the default value of the data parameter P001) when DNC is on-line machining.

11.1.1 Program Start

Directly operate the Comm218M.exe program. The interface is shown below after the program is started.

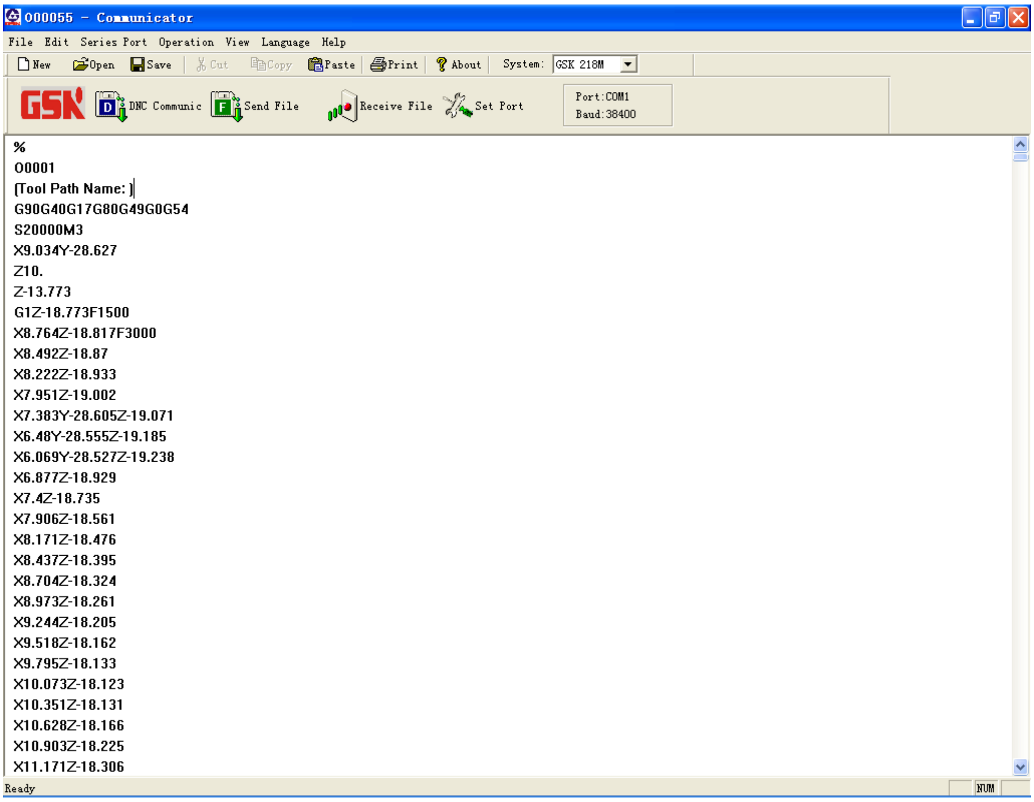


Fig. 11-1-1-1

APPEDIX

11.1.2 Function Introduction

1. File menu

The file menu contains some functions, such as the New, Open and Save the program file, Print and Print setting, the file list for current opening file, etc.

2. Edit menu

Edit menu contains of cut, copy, paste, retraction, reach, replacement etc.

3. Series port menu

It is used for opening and setting of the series port.

4. Transmission method menu

It contains of the DNC transmission method, file delivery transmission method, file reception transmission method.

5. Menu check

The display and hiding of tool and state bars.

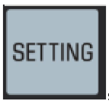
6. Help menu

The software information for this software.

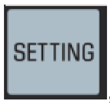
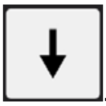
11.1.3 Series Port Data Transmission

The operation steps are shown below:

1) Select <MDI>operation mode;

2) Enter the setting page by , set the I/O channel as 0 or 1.

3) Enter the setting (password) page by【PASSWORD】softkey, input the corresponding level authority password. Refer to the Setting and Modification of Password Authority in Section 3.4.5.

4) Enter the setting (data treatment) page by , move the cursor to the detstination position by  or .

A. Data output (CNC→PC)

1. System prompts “Waiting for the transmitting….” by pressing the【DATA OUTPUT】softkey.

2. Click  button (Alternatively, select the “Accept the file” in the “Transmission mode” draw-down menu on the GSK Com series port communication software, and then the “Accept the file” dialog frame is shown, refer to the Fig. 11-1-3-1.

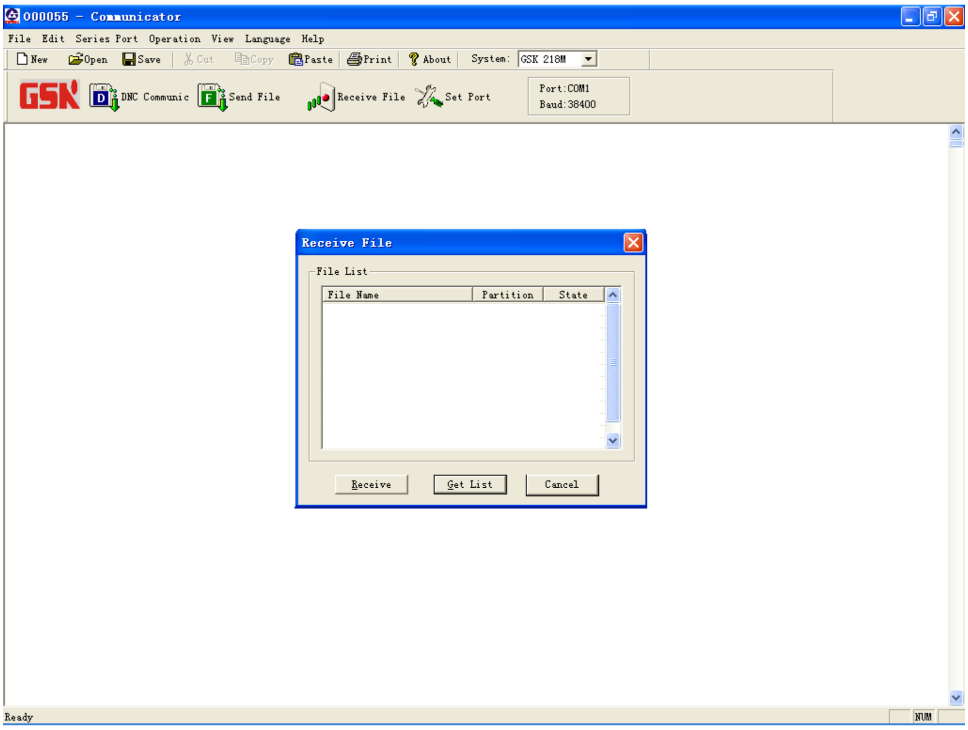


Fig. 11-1-3-1

3. Click the Get List button in acceptance file dialog frame, and then gain the file list at the CNC port, refer to the Fig. 11-1-3-2.

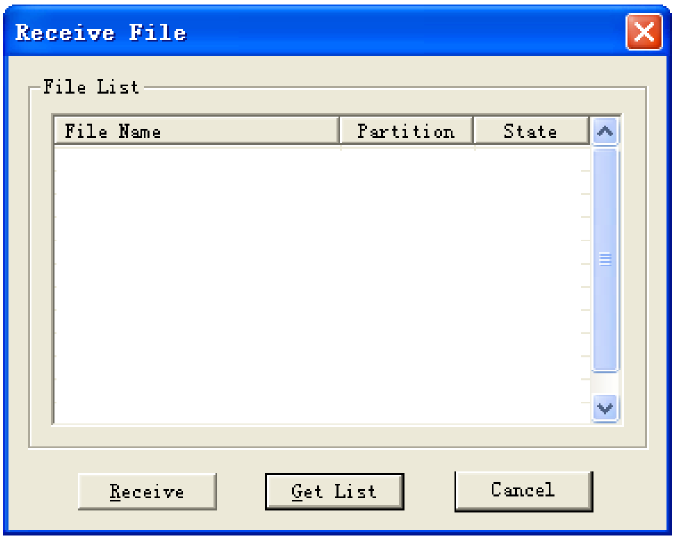


Fig. 11-1-3-2

4. Select the desired file to be accepted (Multiple files can be accepted), and then press the Receive button, the file begins to acceptance, refer to the Fig. 11-1-3-3.

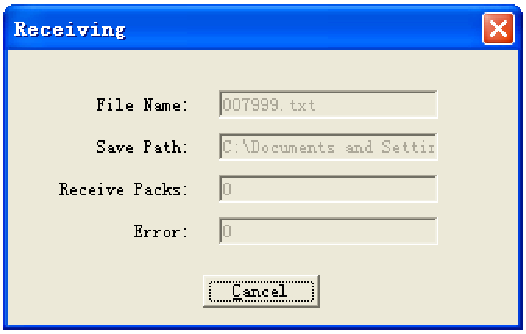


Fig. 11-1-3-3

5. After the file is performed the acceptance, the dialog state bar shows “Accepted”. Refer to the Fig. 11-1-3-4.

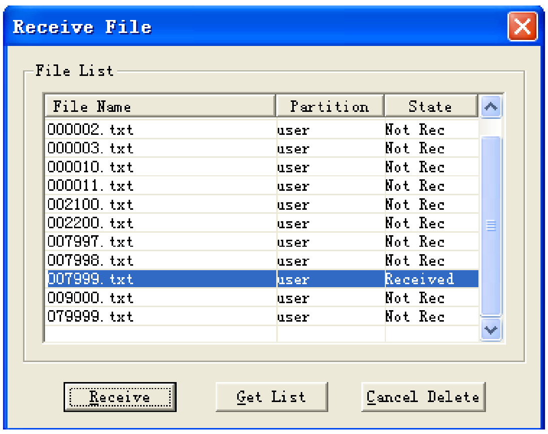


Fig. 11-1-3-4

APPENDIX

B. Data input (PC→CNC)

1. System prompts “Waiting for the transmitting….” by pressing the【DATA INPUT】softkey.

2. Click 

button (Alternatively, select the “Deliver the file” in the “Transmission mode” draw-down menu on the GSK Com series port communication software, and then the “Deliver the file” dialog frame is shown, refer to the Fig. 11-1-3-5.

Fig. 11-1-3-5

3. Click the 

button in acceptance file dialog frame, and then the selection dialog frame shows, refer to the Fig.11-1-3-6.

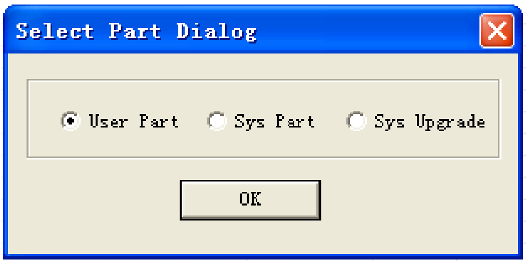


Fig. 11-1-3-6

4. In the selection dialog:

The user sub-area should be selected when the CNC component program and user program are delivered; the system sub-area should be selected when the files suchas the ladder diagram (PLC), parameter (PLC), system parameter value, tool compensation value, pitch compensation value and system macro variable, etc. are transmitted.

5. Select the desired file to be accepted (Multiple files can be delivered), after the sub-area is selected, and then click the 

button, the file delivers, refer to the Fig. 11-1-3-7.

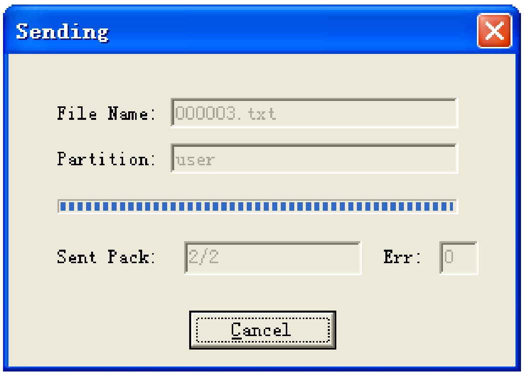


Fig. 11-1-3-7

6. The dialog frame state displays “Delivered” after the file is performed the delivery. Refer to the Fig. 11-1-3-8.

Fig. 11-1-3-8

Note 1: It is necessary to ensure the Baud rate setting is correct before the data transmission, the series port cable connection is reliable.

Note 2: Never attempt to perform the system shifting or page oerate; otherwise, the serious error may occur.

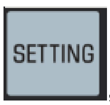
Note 3: LADCHI**.TXT file is disabled after introducing to system, it can be enabled after the power is turned off.

APPEDIX

11.1.4 Series Port DNC ON-Line Machining

Operation steps:

1. CNC port setting:

1) Enter to the setting page by 

, set the I/O channel to 0 or 1.

2) Select the <DNC>operation method; in this case, the system prompts “DNC is already performed,

2. The setting of series port communication software

1) Click the “series port” menu, and then set the baud rate in the “Series port setting” dialog frame, the Baud rate set to 38400.

2) When the system I/O channel sets to 0, the “DNC agreement” in the “Transmission method” drop-down menu should be selected the Xon/Xoff;

When the system I/O channel sets to 1, the “DNC agreement” in the “Transmission method” drop-down menu should be selected the XModem;

3. Open the CNC program file. The program file can be opened by the “Open” button on the file menu or the 

button on tool bar, refer to the Fig. 11-1-4-1 (Further edit the program file to the series port communication software).

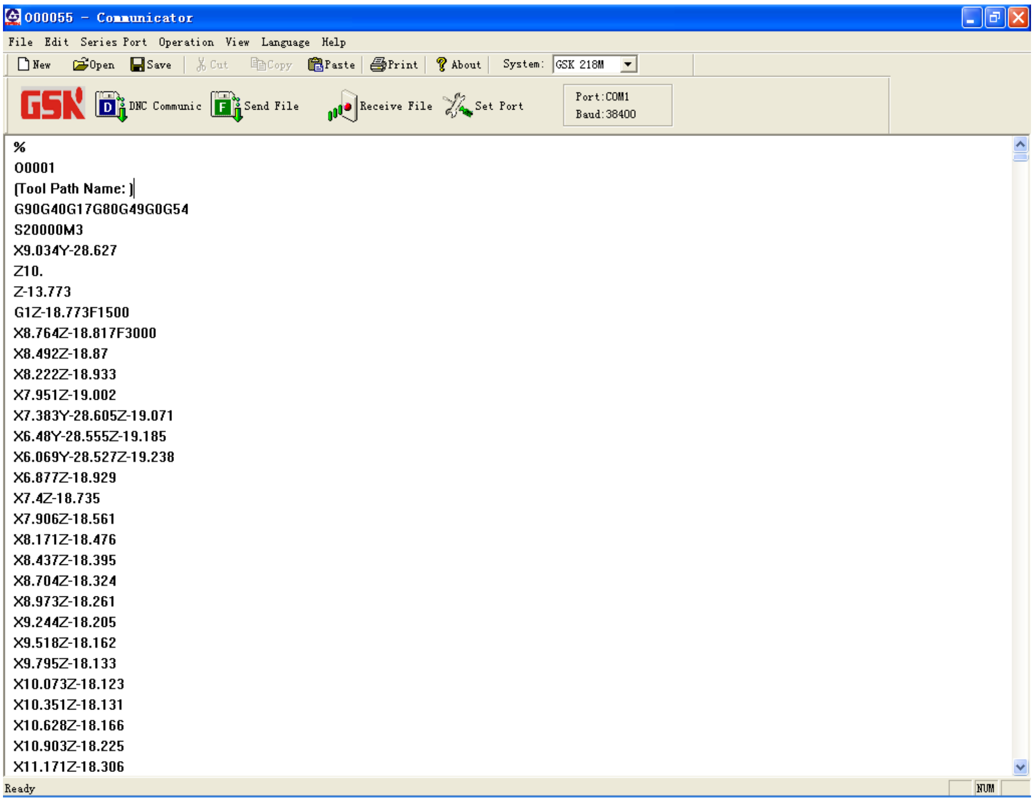


Fig. 11-1-4-1

4. DNC transmission. The data delivers by clicking the 

on tool bar or the “DNC transmission” on the “Transmission mode” drop-down menu. When the system I/O channel sets to 0, PC directly delivers with PC common-use method; in this case, the DNC transmission dialog frame displays the state of file transmission, which includes the delivered file name, delivered byte, line number, and the transmission time and speed (type/second), refer to the Fig. 11-1-4-2. When the system I/O channel sets to 1, PC delivers based upon the data packet; in this case, the dialog frame displays the state of file transmission, which contains the delivered data packet and the times of the retransmission; refer to the Fig. 11-1-4-3:

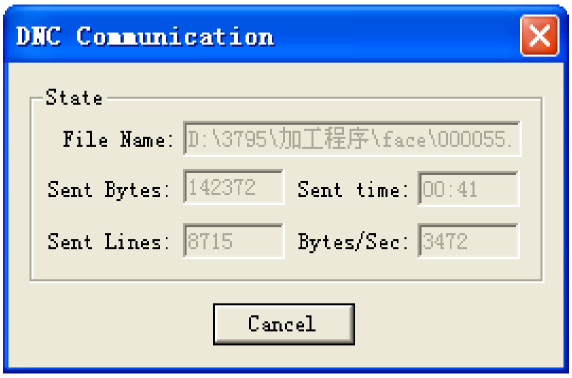


Fig. 11-1-4-2 The system I/O channel sets to 0

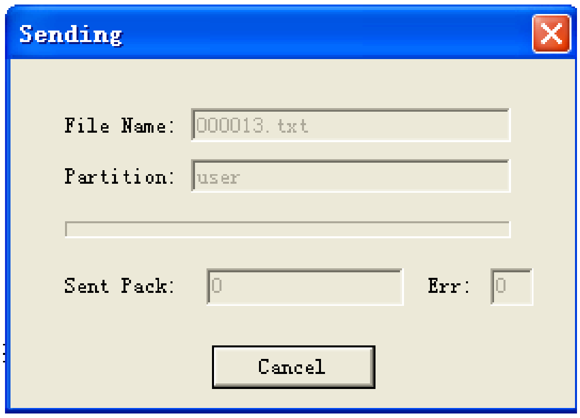


Fig. 11-1-4-3 The system I/O channel sets to 1

Note: 1. The series port communication software can not be performed other operations other than the end transmission during the DNC transmission.

2. M99 regards as M30 in DNC mode.

3. Cancel the operation by 

after the machining is completed.

11.2 USB Communication

11.2.1 Brief & Precaution

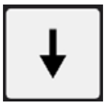
Precautions:

1. Set the I/O channel to 2 in <SETTING> interface.
2. The suffix name of CNC program should be .txt、.nc or .CNC, and stored at the U disk root directory; otherwise, the system will not be read.
3. Never attempt to pull out the U disk when transmitting the communication by USB, to avoid the

product fault or unexpected result.

4. The U disk can be pulled out when the indicator of U disk does not flash after the U disk communication operation is completed, to guarantee that the data transmission is executed.

11.2.2 USB Component Program Operation Steps

In the <MDI MODE>, move the cursor to “CNC component program” by direction button  or  after entering the setting (data treatment) interface. Enter the following operation interface by softkey【DATA OUTPUT】or【DATA INPUT】, refer to the Fig. 11-2-2-1:

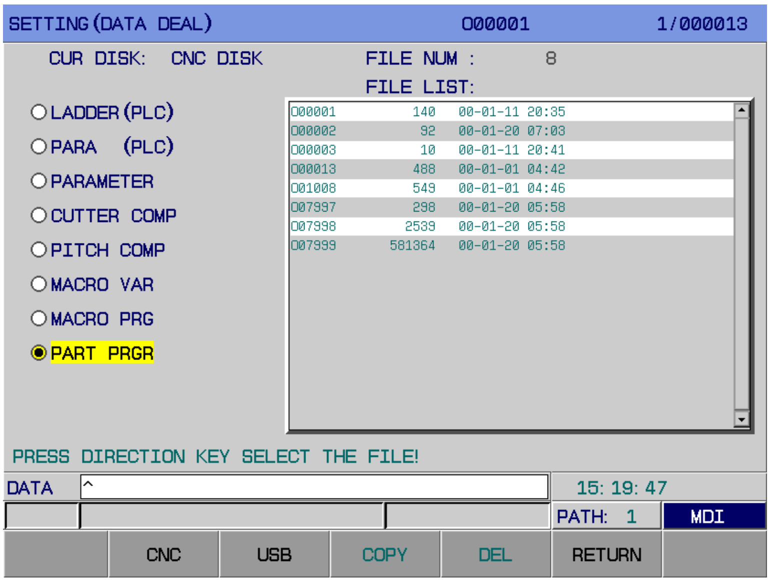
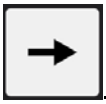
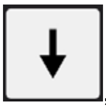


Fig. 11-2-2-1

1. Copy the CNC program file from system disk to U:

a. Shift the cursor to the file list table by direction button .

b. Move the cursor by  or , select the CNC program file in system disk to be copied.

c. The system prompts “Copy to U disk? New file name” by【COPY】softkey; refer to the following figure (Fig. 11-2-2-2)

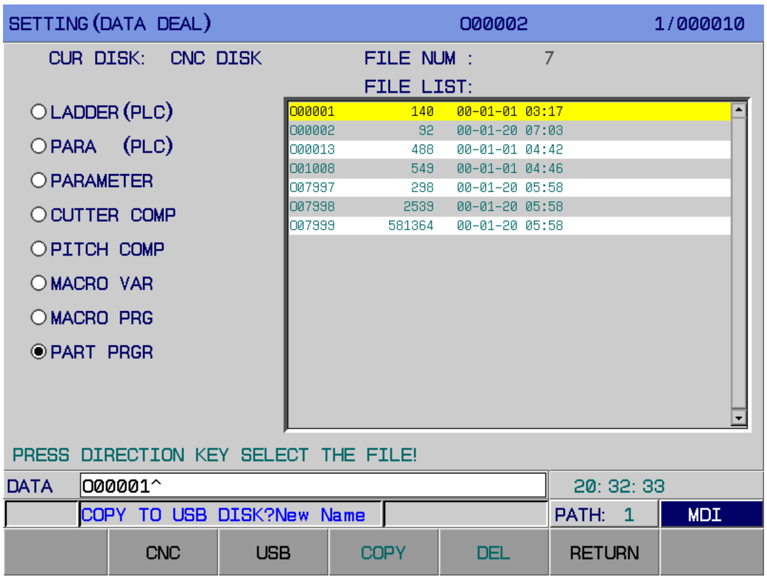


Fig. 11-2-2-2

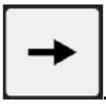
d. If the CNC program file does not need to be renamed, it can be directly copied by <INPUT> button.

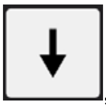
If the CNC program file should be renamed, input the new program number (For example, O10, O100) by <CANCEL>, and the CNC program file then can be copied by <INPUT> again.

If the U disk stores at the program file with a same name, the system may prompt "RENAME AGAIN", therefore, it is necessary to input the new program numbers (for example: O10, O100); and then the CNC program file can be copied by <INPUT> button.

2. Copy the CNC program file to system disk from U disk:

a. Shift to the display interface of the U disk file list by【U DISK】softkey.

b. Shift the cursor to the file list table by direction button .

c. Move the cursor by  or , select the CNC program file in U disk to be copied.

The system prompts "Copy to system disk? New file name" by【COPY】softkey; refer to the following figure (Fig. 11-2-2-3)

APPEDIX

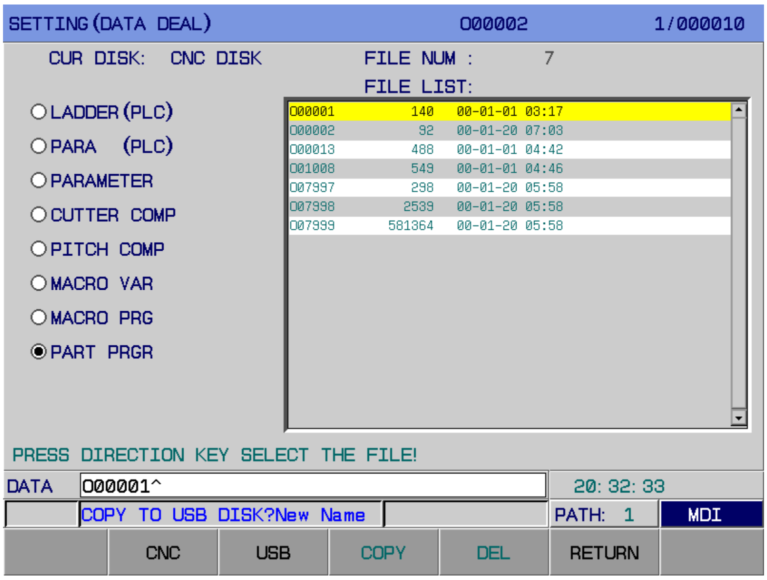


Fig. 11-2-2-3

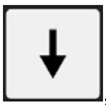
d. If the CNC program file does not need to be renamed, it can be directly copied by <INPUT> button.

If the CNC program file should be renamed, input the new program number (For example, O10, O100) by <CANCEL>, and the CNC program file then can be copied by <INPUT> again.

If the system disk stores at the program file with a same name, the system may prompt “RENAME AGAIN”, therefore, it is necessary to input the new program numbers (for example: O10, O100); and then the CNC program file can be copied by <INPUT> button.

Note: The LADCHI**.TXT file is disabled after transferring to the system, it can be enabled after the power is turned off.

3. Delete a file from system disk/U disk:

a. Move the cursor by  or , select the CNC program file in the system disk/U disk to be deleted.

b. “Confirm deleting the current file?” prompts at the botoom of the interface by【DELET】softkey; cancle the file deletion by <CANCEL>; the file is deleted by <INPUT>button.

11.2.3 USB DNC Machine Operation Steps

1. In the <Setting> interface, set the I/O channel to 2. Refer to the Section 3.4.1 in *OPERATION* for details.

2. Insert the U disk.

3. The system shifts to DNC mode by <DNC>button, in this case, prompt shows at the bottom of the screen: “Select the machining file in USB list”; enter the program interface by <PROGRAM>,

APPEDIX

and then【LIST】, the USB program list table shows; select the program to be machined by moving the cursor, open this program by <INPUT>, then press the <CYCLE START>, and then perform the DNC machining.

Note: In the USB program list page, when the character numbers of program name is less than or equals to 6, the start of program can be previewed; when the character numbers of program name is more than 6, the start of program can not be previewed. When the character numbers of program name is more than or equals to 8, the system is only displayed as abbreviation and can not be previewed the start of program.

11.2.4 Retreat from U Disk Operation Interface

1. Pull out the U disk when the indicator of the U disk does not flash.

Retreat from【SETTING (DATA TREATMENT)】interface by【RETURN】softkey.

APPENDIX

APPENDIX ONE GSK218MC SERIES PARAMETER LIST

Parameter Explanation

The parameter can be dividied into the following types based upon the types of the data:

Two data types and its enabled range of the data value

Data type	Enabled data range	Remark
Bit-type	0 or 1	The default value provided by system can be modified its setting based upon the user's requirements.
Data-type	It determines based upon the data range.	The default range and value provided by system can be modidified its setting based upon the user's requirements.

1. Each data consists of 8-digit for the bit parameter, and each bit owns different meanings.
2. In the above-mentioned table, generally, the data value range of each data type is the enabled range. Actually, the concrete parameter value range is different. Refer to the detailed explanations for each parameter.

[Example]

(1) The meaning of bit parameter

Data No. **BIT7** **BIT6** **BIT5** **BIT4** **BIT3** **BIT2** **BIT1** **BIT0**

(2) The meaning of data parameter

0	2	1	

Data No. Data

Note 1: The null position in parameter explanation and the parameter number on the screen but without recording in the parameter table are backup for the extend in the future, so it is necessary to set as 0.

Note 2: The 0 or 1 in the parameter does not specify the concrete meaning; 1: YES; 0: NO.

Note 3: INI sets to 0; parameter that sets the unit linear axis is mm, mm/min when the metric input is performed; the basis unit of the rotation axis is deg, deg/min.

INI sets to 1; parameter that sets the unit linear axis is inch, inch/min when the inch input is performed; the basis unit of the rotation axis is deg, deg/min.

1. Bit Parameter

System parameter No.

0	0	0		MODE	SVCD	SEQ	MSP	CPB	INI	INM	PBUS

PBUS =1: The transmission method of the drive is bus type.

=0: The transmission method of the drive is pulse type.

INM =1: The least movement unit of the linear axis is inch method

=0: The least movement unit of the linear axis is metric method

INM sets to 0, and when the metric input executes: the basis unit of the linear axis is mm, mm/min; the basis unit of rotation axis is deg, deg/min.

INM sets to 1, and when the inch input executes: the basis unit of the linear axis is inch, inch/min; the basis unit of rotation axis is deg, deg/min.

INI =1: Inch input

=0: Metric input

INM sets to 0, and when the metric input executes: the basis unit of the linear axis is mm, mm/min; the basis unit of rotation axis is deg, deg/min.

INM sets to 1, and when the inch input executes: the basis unit of the linear axis is inch, inch/min; the basis unit of rotation axis is deg, deg/min.

CPB =1: Pulse and Ethernet are simultaneously used.

=0: Pulse and Ethernet are not simultaneously used.

MSP =1: Use the dual-spindle control.

=0: Do not use the dual-spindle control.

SEQ =1: Automatically insert the sequence number.

=0: Do not automatically insert the sequence number.

SVCD =1: Use the bus servo card.

=0: Do not use the bus servo card.

MODE =1: High-speed & high-accuracy mode, it can not be modify the #15.0 and #17.0 but supporting the 4-axis 3-linkage.

=0: Common mode, when the high-speed & high-accuracy mode sets to the common one, #15.0 sets to 1 by default.

Standard setting: 1 0 0 0 0 0 0 1

System parameter number

0	0	1		RAS5	RAS4	RAS3	RAS2	RAS1	SPT	SBUS	RASA

RASA =1: Use the absolute grating scale.

=0: Do not use the absolute grating scale.

SBUS =1: Spindle drive is bus control method.

=0: Spindle drive is non-bus control method.

SPT =1: I/O point control.

=0: Frequency-conversion or others

RAS1 =1: Use the grating scale by setting the 1st axis.

=0: Do not use the grating scale by setting the 1st axis.

RAS2 =1: Use the grating scale by setting the 2nd axis.

=0: Do not use the grating scale by setting the 2nd axis.

RAS3 =1: Use the grating scale by setting the 3rd axis.

=0: Do not use the grating scale by setting the 3rd axis.

RAS4 =1: Use the grating scale by setting the 4th axis.

=0: Use the grating scale by setting the 4th axis.

RAS5 =1: Use the grating scale by setting the 5th axis.

=0: Do not use the grating scale by setting the 5th axis.

Standard setting：0 0 0 0 0 0 0 0

System parameter number

0	0	2					DEC5	DEC4	DEC3	DEC2	DEC1

DEC1 =1: Deleration is performed when the deceleration signal is 1 and the 1st axis reference position returns.

=0: Deleration is performed when the deceleration signal is 0 and the 1st axis reference position returns.

DEC2 =1: Deleration is performed when the deceleration signal is 1 and the 2nd axis reference position returns.

=0: Deleration is performed when the deceleration signal is 0 and the 2nd axis reference position returns.

DEC3 =1: Deleration is performed when the deceleration signal is 1 and the 3rd axis reference position returns.

=0: Deleration is performed when the deceleration signal is 0 and the 3rd axis reference position returns.

DEC4 =1: Deleration is performed when the deceleration signal is 1 and the 4th axis reference position returns.

=0: Deleration is performed when the deceleration signal is 0 and the 4th axis reference position returns.

=1: 第5轴参考点返回时减速信号为1时减速

DEC5 =1: Deleration is performed when the deceleration signal is 1 and the 5th axis reference position returns.

=0: Deleration is performed when the deceleration signal is 0 and the 5th axis reference position returns.

Standard setting：0 0 0 0 0 0 0 0

System parameter number

0	0	3					DIR5	DIR4	DIR3	DIR2	DIR1

DIR1 =1: The 1st axis feed direction is reverse.
=0: The 1st axis feed direction is positive.

DIR2 =1: The 2nd axis feed direction is reverse.
=0: The 2nd axis feed direction is positive.

DIR3 =1: The 3rd axis feed direction is reverse.
=0: The 3rd axis feed direction is positive.

DIR4 =1: The 4th axis feed direction is reverse.
=0: The 4th axis feed direction is positive.

DIR5 =1: The 5th axis feed direction is reverse.
=0: The 5th axis feed direction is positive.

Standard setting：0 0 0 0 0 0 0 0

System parameter number

0	0	4		SK0						STME	TMES

TMES =1: Tool-setter is already installed.
=0: Tool-setter does not install.

STME =1: Tool length measurement value can be written into the reference offset.
=0: Tool length measurement value can not be written into the reference offset.

SK0 =1: SKIP is regarded as signal input when it is set to 0.
=0: SKIP is regarded as signal input when it is set to 1.

Standard setting：0 0 0 0 0 0 0 0

System parameter number

0	0	5		DOUS			HSRZ			ISC	

ISC =1: The least movement unit 0.0001mm&deg,0.00001inch.
=0: The least movement unit 0.001mm&deg,0.0001inch.

HSRZ =1: High-speed zero return is enabled.
=0: High-speed zero return is disabled.

DOUS =1: Dual-drive tool uses the grating position.
=0: Dual-drive tool does not use the grating position.

Standard setting：0 0 0 0 0 0 0 0

System parameter number

0	0	6		MAOB	ZPLS	SIOD	SJZ	AZR	JAX	ZMOD	ZRN

APPEDIX

ZRN =1: Reference point does not establish, specify the code other than G28 in Auto operation, system alarms.

=0: Reference point does not establish, specify the code other than G28 in Auto operation, system does not alarm.

ZMOD =1: Zero return mode selection: Before the block.

=0: Zero return mode selection: After the block.

JAX =1: The control axis at the same time of the manual reference point return: single axis.

=0: The control axis at the same time of the manual reference point return: multiplication axes.

AZR =1: The G28 command when the reference point does not establish: Alarm

=0: The G28 command when the reference point does not establish: Use the block

SJZ =1: Reference point memories.

=0: Reference point does not memory.

SIOD =1: Mechanical zero return deceleration signal performs via PLC logic calculation.

=0 Mechanical zero return deceleration signal directly reads.

ZPLS =1: Zero return method selection: with one-turn signal.

=0: Zero return method selection: without one-turn signal.

MAOB =1: Zero return method selection without one-turn signal: B method.

=0: Zero return method selection without one-turn signal: A method.

Standard setting：1 1 1 0 0 0 0 1

System parameter number

0	0	7					ZMI5	ZMI4	ZMI3	ZMI2	ZMI1

ZMI1 =1: Set the 1st axis reference point return direction: -

=0: Set the 1st axis reference point return direction: +

ZMI2 =1: Set the 2nd axis reference point return direction: -

=0: Set the 2nd axis reference point return direction: +

ZMI3 =1: Set the 3rd axis reference point return direction: -

=0: Set the 3rd axis reference point return direction: +

ZMI4 =1: Set the 4th axis reference point return direction: -

=0: Set the 4th axis reference point return direction: +

ZMI5 =1: Set the 5th axis reference point return direction: -

=01: Set the 5th axis reference point return direction: +

Standard setting：0 0 0 0 0 0 0 0

System parameter number

APPEDIX

0	0	8					AXS5	AXS4	AXS3	AXS2	AXS1

AXS1 =1: The 1st axis sets to rotation axis.
=0: The 1st axis sets to linear axis.

AXS2 =1: The 2nd axis sets to rotation axis.
=0: The 2nd axis sets to linear axis.

AXS3 =1: The 3rd axis sets to rotation axis.
=0: The 3rd axis sets to linear axis.

AXS4 =1: The 4th axis sets to rotation axis.
=0: The 4th axis sets to linear axis.

AXS5 =1: The 5th axis sets to rotation axis.
=0: The 5th axis sets to linear axis.

Standard setting：0 0 0 0 0 0 0 0

System parameter number

0	0	9		PD5	PD4	PD3	PD2	PD1		A4TP	RAB

RAB =1: Each axis rotates nearest when it regards as the rotation axis.
=0: Each axis does not rotates nearest when it regards as the rotation axis.

A4TP =1: 4-axis linkage system.
=0: Non-4-axis linkage system.

PD1 =1: The 1st axis pulse direction of the Non-bus servo is negative.
=0: The 1st axis pulse direction of the Non-bus servo is positive.

PD2 =1: The 2nd axis pulse direction of the Non-bus servo is negative.
=0: The 2nd axis pulse direction of the Non-bus servo is positive.

PD3 =1: The 3rd axis pulse direction of the Non-bus servo is negative.
=0: The 3rd axis pulse direction of the Non-bus servo is positive.

PD4 =1: The 4th axis pulse direction of the Non-bus servo is negative.
=0: The 4th axis pulse direction of the Non-bus servo is positive.

PD5 =1: The 5th axis pulse direction of the Non-bus servo is negative.
=0: The 5th axis pulse direction of the Non-bus servo is positive.

Standard setting：0 0 0 0 0 0 0 1

System parameter number

0	1	0		RCUR	MSL			RLC	ZCL	SCBM	

SCBM =1: Perform the stroke detection before moving.
=0: Do not perform the stroke detection before moving.

ZCL =1: Clear the relative coordinate of the reference point return.
=0: Do not clear the relative coordinate of the reference point return.

APPEDIX

RLC =1: Cancel the relative coordinate after resetting.

=0: Do not cancel the relative coordinate after resetting.

MSL =1: The start line is the one which cursor locates when circularly starting the multi-block MDI.

=0: The start line is the first line of the program when circularly starting the multi-block MDI.

RCUR =1: Non-editing mode resetting cursor returns to the start position of the program.

=0: Non-editing mode resetting cursor does not return to the start position of the program.

Standard setting：0 0 0 0 0 0 1 0

System parameter number

0	1	1	BFA	LZR						OUT2

OUT2 =1: Prohibit entering the external area of the 2nd stroke limit.

=0: Prohibit entering the internal area of the 2nd stroke limit.

LZR =1: Perform the stroke detection after the power is turned on till to the manual reference point return.

=0: Do not perform the stroke detection after the power is turned on till to the manual reference point return.

BFA =1: The alarm occurs followed with the overtravel when the overtravel code issues.

=0: The alarm occurs before the overtravel when the overtravel code issues. (System alarm range is previous 5mm of each boundary for the prohibition area)

Standard setting：0 0 0 0 0 0 0 1

System parameter number

0	1	2	RDR	FDR	TDR	RFO			LRP	RPD

RPD =1: Manual rapid is enabled from the power-on till to the reference point return.

=0: Manual rapid is disabled from the power-on till to the reference point return.

LRP =1: Positioning (G00) interpolation type is straight line.

=0: Positioning (G00) interpolation type is non-straight line.

RFO =1: Rapid feed, it stops when the feedrate is Fo.

=0: Rapid feed, it does not stop when the feedrate is Fo.

TDR =1: During the tapping, the dry run is enabled.

=0: During the tapping, the dry run is disabled.

FDR =1: The dry run is enabled during cutting feed.

APPEDIX

	=0: The dry run is disabled during cutting feed.
RDR	=1: The dry run is enabled in the rapid positioning.
	=0: The dry run is disabled in the rapid positioning.

Standard setting：0 0 0 0　0 0 0 0

System parameter number

0	1	3								HPC	NPC

NPC =1: The feed/rev. is enabled when the position encoder does not install.
=0: The feed/rev. is disabled when the position encoder does not install.

HPC =1: System installs a position encoder.
=0: System does not install a position encoder.

Standard setting：0 0 0 0　0 0 1 0

System parameter number

0	1	4								DLF	

DLF =1: Manual zero return positions to the reference point manually at the rapid traverse rate after the refernce point establishes and memories.
=0: Manual zero return positions to the reference point at the rapid traverse rate after the refernce point establishes and memories.

Standard setting：0 0 0 0　0 0 0 0

System parameter number

0	1	5				PIIS		PPCK	ASL	PLAC	STL

STL =1: Select the pre-reading machining method.
=0: Select the non-pre-reading machining method.

PLAC =1: The acceleration/deceleration method after the predictive control interpolation: Exponent type.
=0: The acceleration/deceleration method after the predictive control interpolation: Linear type.

ASL =1: The automatic corner deceleration function of the predictive control: Velocity difference control.
=0: The automatic corner deceleration function of the predictive control: Angle control.

PPCK =1: Predictive control performs the in-position detection.
=0: Predictive control do not perform the in-position detection.

PIIS =1: The overlapping interpolation of acceleration/deceleration block is enabled before the predictive control.

APPEDIX

=0: The overlapping interpolation of acceleration/deceleration block is disabled before the predictive control.

Standard setting：0 0 0 0 0 0 0 0

System parameter number

0	1	6		ALS					FLLS	FBLS	FBOL

FBOL =1: Rapid operation mode: Backward acceleration/deceleration.
=0: Rapid operation mode: Forward acceleration/deceleration.

FBLS =1: The acceleration/deceleration before the rapid traverse: S type.
=0: The acceleration/deceleration before the rapid traverse: Linear type.

FLLS =1: The acceleration/deceleration after the rapid traverse: Exponent type.
=0: The acceleration/deceleration after the rapid traverse: Linear type.

ALS =1: Automatic corner feed function is enabled.
=1: Automatic corner feed function is disabled.

Standard setting：0 0 0 0 0 0 1 0

System parameter number

0	1	7		CPCT	CALT	WLOE		HLOE	CLLE	CBLS	CBOL

CBOL =1: Cutting feed mode: Backward acceleration/deceleration.
=0: Cutting feed mode: Forward acceleration/deceleration.

CBLS =1: The acceleration/deceleration before the cutting feed: S type.
=0: The acceleration/deceleration before the cutting feed: Linear type.

CLLE =1: The acceleration/deceleration after the cutting feed: Exponent type.
=1: The acceleration/deceleration after the cutting feed: Linear type.

HLOE =1: JOG operation selection: Exponent type.
=0: JOG operation selection: Linear type.

WLOE =1: MPG operation selection: Exponent type.
=0: MPG operation selection: Linear type.

CALT =1: Cutting feed acceleration clamping.
=0: Cutting feed acceleration releasing.

CPCT =1: Cutting feed controls the in-position accuracy.
=0: Cutting feed does not control the in-position accuracy.

Standard setting：1 0 1 0 0 0 0 1

System parameter number

0	1	8		RVCS	RBK						RVIT

RVIT =1: The next block is performed after the compensation is performed when the reverse interval is more than the interval allowable tolerance D-value.

=0: The next block is performed after the compensation does not perform when the reverse interval is more than the interval allowable tolerance D-value.

RBK =1: The cutting feed/rapid movement is separately performed the interval compensatin.

=0: The cutting feed/rapid movement does not separately perform the interval compensatin.

RVCS =1: Reverse interval compensation mode: Speed-up/speed-down.

=0: Reverse interval compensation mode: Fixed frequency.

Standard setting：0 0 0 0 0 0 0 0

System parameter number

0	1	9				ALMS	ALM5	ALM4	ALM3	ALM2	ALM1

ALM1 =1: The alarm is generated when the 1st axis drive alarm signal is set to 1.

=0: The alarm is generated when the 1st axis drive alarm signal is set to 0.

ALM2 =1: The alarm is generated when the 2nd axis drive alarm signal is set to 1.

=0: The alarm is generated when the 2nd axis drive alarm signal is set to 0.

ALM3 =1: The alarm is generated when the 3rd axis drive alarm signal is set to 1.

=0: The alarm is generated when the 3rd axis drive alarm signal is set to 0.

ALM4 =1: The alarm is generated when the 4th axis drive alarm signal is set to 1.

=0: The alarm is generated when the 4th axis drive alarm signal is set to 0.

ALM5 =1: The alarm is generated when the 5th axis drive alarm signal is set to 1.

=0: The alarm is generated when the 5th axis drive alarm signal is set to 0.

ALMS =1: The alarm is generated when the spindle drive alarm signal is set to 1.

=0: The alarm is generated when the spindle drive alarm signal is set to 0.

Standard setting：0 0 1 0 0 0 0 0

System parameter number

0	2	0		UHSM	APC	MAPC		ZRS			ITL

ITL =1: The interlocking of the overall axes are enabled.

=0: The interlocking of the overall axes are disabled.

ZRS =1: The zero point signal is subject on the machine coordinate.

=0: The zero point signal is subject on the absolute encoder.

MAPC =1: Select the absolute encoder: Multi-coil.

=0: Select the absolute encoder: Single-coil.

APC =1: Use an absolute encoder.

=0: Do not use an absolute encoder.

UHSM =1: Directly set the machine zero by manual.

=0: The machine zero can not be directly set by manual.

Standard setting：1 0 0 0 0 0 0 0

System parameter number

0	2	1					APZ5	APZ4	APZ3	APZ2	APZ1

APZ1 =1: The current machine position of the 1st axis sets to zero.
=0: The current machine position of the 1st axis does not set to zero.

APZ2 =1: The current machine position of the 2nd axis sets to zero.
=0: The current machine position of the 2nd axis does not set to zero.

APZ3 =1: The current machine position of the 3rd axis sets to zero.
=0: The current machine position of the 3rd axis does not set to zero.

APZ4 =1: The current machine position of the 4th axis sets to zero.
=0: The current machine position of the 4th axis does not set to zero.

APZ5 =1: The current machine position of the 5th axis sets to zero.
=0: The current machine position of the 5th axis does not set to zero.

Standard setting：0 0 0 0 0 0 0 0

System parameter number

0	2	2		DAL						

DAL =1: The absolute position display considers the tool length compensation.
=0: The absolute position display does not consider the tool length compensation.

Standard setting：0 0 0 0 0 0 0

System parameter number

0	2	3		POSM						

POSM =1: Program monitoring page displays the modal.
=0: Program monitoring page does not display the modal.

Standard setting：0 1 0 0 0 0 0 0

System parameter number

0	2	4		NPA						

NPA =1: Shift to the alarm screen when alarm issues.
=0: Do not shift to the alarm screen when alarm issues.

Standard setting：0 0 0 0 0 0 0 0

System parameter number

APPEDIX

0	2	5		ALM	DGN	GRA	SET		SYS	PRG	POS

POS =1: In the position interface, shift the screen by pressing the "Position" button again.

=0: In the position interface, do not shift the screen by pressing the "Position" button again.

PRG =1: In the position interface, shift the screen by pressing the "Program" button again.

=0: In the position interface, do not shift the screen by pressing the "Program" button again.

SYS =1: In the position interface, shift the screen by pressing the "System" button again.

=0: In the position interface, do not shift the screen by pressing the "System" button again.

SET =1: In the position interface, shift the screen by pressing the "Setting" button again.

=0: In the position interface, do not shift the screen by pressing the "Setting" button again.

GRA =1: In the position interface, shift the screen by pressing the "Graph" button again.

=0: In the position interface, do not shift the screen by pressing the "Graph" button again.

DGN =1: In the position interface, shift the screen by pressing the "Diagnosis" button again.

=0: In the position interface, do not shift the screen by pressing the "Diagnosis" button again.

ALM =1: In the position interface, shift the screen by pressing the "Alarm" button again.

=0: In the position interface, do not shift the screen by pressing the "Alarm" button again.

Standard setting：1 1 1 1　0 1 1 1

System parameter number

0	2	6		HELP	PLC			SMDT	SMDI	SPET	PETP

PETP =1: Automatically skip to the program page by "Editing" button.

=0: Do not automatically skip to the program page by "Editing" button.

SPET =1: In the Editing mode, automatically skip to the program interface by <PROGRAM>.

	=0: In the Editing mode, do not automatically skip to the program interface by <PROGRAM>.
SMDI	=1: In the MDI mode, automatically skip to the MDI interface by <PROGRAM>.
	=0: In the MDI mode, do not automatically skip to the MDI interface by <PROGRAM>.
SMDT	=1: In the MDI mode, automatically skip th the CUR/MOD interface selection.
	=0: In the MDI mode, automatically skip th the MDI interface selection.
PLC	=1: In the PLC interface, shift the page by pressing the “Program control” button again.
	=0: In the PLC interface, do not shift the page by pressing the “Program control” button again.
HELP	=1: In the Help interface, shift the screen by pressing the “Help” button again.
	=0: In the Help interface, do not shift the screen by pressing the “Help” button again.

Standard setting：1 1 0 0 0 0 0 1

System parameter number

0	2	7					NE9				NE8

NE8	=1: Prohibit the sub-program editing of the program numbers 80000 – 89999.
	=0: Allow the sub-program editing of the program numbers 80000 – 89999.
NE9	=1: Prohibit the sub-program editing of the program numbers 90000 - 99999.
	=0: Allow the sub-program editing of the program numbers 90000 - 99999.

Standard setting：0 0 0 1 0 0 0 1

System parameter number

0	2	8		MCL			MKP				

MKP	=1: Clear the compiled program when performing the M02, M30 or % in the MDI mode.
	=1: Keep the compiled program when performing the M02, M30 or % in the MDI mode.
MCL	=1: Delete the compiled program by resetting button in MDI mode.
	=0: Retain the compiled program by resetting button in MDI mode.

Standard setting：0 0 0 1 0 0 0 0

System parameter number

0	2	9					IWZ	WZO	MCV	GOF	WOF

WOF	=1: Prohibit inputting the tool wear offset value by MDI keyboard.

APPEDIX

=0: Input the tool wear offset value by MDI keyboard.

GOF =1: Prohibit inputting the tool geometry offset value by MDI keyboard.

=0: Input the geometry offset value by MDI keyboard.

MCV =1: Prohibit inputting the macro program varible by MDI keyboard.

=0: Input the macro program variable by MDI keyboard.

WZO =1: Prohibit inputting the workpiece origin offset value by MDI keyboard.

=0: Input the workpiece origin offset value by MDI keyboard

IWZ =1: Prohibit inputting the workpiece origin offset value by MDI keyboard in the dwell.

=0: Input the workpiece origin offset value by MDI keyboard in the dwell.

Standard setting：0 0 0 0　0 0 0 0

System parameter number

0	3	1				G13	G91		G19	G18	G01

G01 =1: It is the G01 mode when the power is turned on or the state is cleared.

=0: It is the G00 mode when the power is turned on or the state is cleared.

G18 =1: The plane selection is G18 mode when the power is turned on or the state is cleared.

=0: The plane selection is G17 mode when the power is turned on or the state is cleared.

G19 =1: It is the G19 mode, when G19=1, it is better set the G18 to 0.

=0: It is determined by bit 1 of parameter No.:31.

G19	G18	G17, G18 and G19 methods
0	0	G17 method (X-Y plane)
0	1	G18 method (Z-X plane)
1	0	G19 method (Y-Z plane)

G91 =1: It sets to the G91 mode when the power is turned on or the state is cleared.

=0: It sets to the G90 mode when the power is turned on or the state is cleared.

G13 =1: It sets to the G13 mode when the power is turned on or the state is cleared.

=0: It sets to the G12 mode when the power is turned on or the state is cleared.

Standard setting：0 0 1 0　0 0 0 0

System parameter number

0	3	2			AD2						

AD2 =1: Specify two or more same addresses in a same block, the system then alarms.

APPEDIX

=0: Specify two or more same addresses in a same block, the system does not alarms.

Standard setting：0 1 0 0 0 0 0 0

System parameter number

0	3	3		M3B			M30		M02		

M02 =1: Return to the start of the block when performing to the M02.
=0: Do not return to the start of the block when performing to the M02.

M30 =1: Return to the start of the block when performing to the M30.
=0: Return to the start of the block when performing to the M30.

M3B =1: Up to 3 M codes can be specified in a program.
=0: Only one M code can be specified in a program.

Standard setting：1 0 0 1 0 0 0 0

System parameter number

0	3	4		CFH							DWL

DWL =1: In the feed/rev. mode, G04 is the dwell of per revelution.
=0: In the feed/rev. mode, G04 is not the dwell of per revelution.

CFH =1: Clear the F, H and D codes when resetting or ESP.
=0: Remain the F, H and D codes when resetting or ESP.

Standard setting：0 0 0 0 0 0 0 0

System parameter number

0	3	5		C07	C06	C05	C04	C03	C02	C01	

C01 =1: Clear the G codes in group 01 when resetting or ESP.
=0: Reserve the G codes in group 01 when resetting or ESP.

C02 =1: Clear the G codes in group 02 when resetting or ESP.
=0: Reserve the G codes in group 02 when resetting or ESP.

C03 =1: Clear the G codes in group 03 when resetting or ESP.
=0: Reserve the G codes in group 03 when resetting or ESP.

C04 =1: Clear the G codes in group 04 when resetting or ESP.
=0: Reserve the G codes in group 04 when resetting or ESP.

C05 =1: Clear the G codes in group 05 when resetting or ESP.
=0: Reserve the G codes in group 05 when resetting or ESP.

C06 =1: Clear the G codes in group 06 when resetting or ESP.
=0: Reserve the G codes in group 06 when resetting or ESP.

C07 =1: Clear the G codes in group 07 when resetting or ESP.

=0: Reserve the G codes in group 07 when resetting or ESP.

Standard setting：1 0 0 0 0 0 0 0

System parameter number

0	3	6		C15	C14	C13	C12	C11	C10	C09	C08

C08 =1: Clear the G codes in group 08 when resetting or ESP.
=0: Reserve the G codes in group 08 when resetting or ESP.

C09 =1: Clear the G codes in group 09 when resetting or ESP.
=0: Reserve the G codes in group 09 when resetting or ESP.

C10 =1: Clear the G codes in group10 when resetting or ESP.
=0: Reserve the G codes in group 10 when resetting or ESP.

C11 =1: Clear the G codes in group 11 when resetting or ESP.
=0: Reserve the G codes in group 11 when resetting or ESP.

C12 =1: Clear the G codes in group 12 when resetting or ESP.
=0: Reserve the G codes in group 12 when resetting or ESP.

C13 =1: Clear the G codes in group 13 when resetting or ESP.
=0: Reserve the G codes in group 13when resetting or ESP.

C14 =1: Clear the G codes in group 14 when resetting or ESP.
=0: Reserve the G codes in group 14 when resetting or ESP.

C15 =1: Clear the G codes in group 15 when resetting or ESP.
=0: Reserve the G codes in group 15 when resetting or ESP.

Standard setting：0 0 0 0 0 0 0 1

System parameter number

0	3	7						SOC	RSC	BDP	SCRW

SCRW =1: Perform the pitch compensation.
=0: Do not perform the pitch compensation.

BDP =1: Use the bi-directional pitch error compensation.
=0: Do not use the bi-directional pitch error compensation.

RSC =1: Calculating the reference coordinate of G96 spindle speed is regarded as current point when the G0 is positioned at the rapid traverse rate.
=0: Calculating the reference coordinate of G96 spindle speed is regarded as en point when the G0 is positioned at the rapid traverse rate.

SOC =1: G96 spindle speed clamps after the spindle override.
=0: G96 spindle speed clamps before the spindle override.

Standard setting：0 0 0 0 0 0 0 0

System parameter number

APPEDIX

0	3	8		PG2	PG1		FLRE	FLR			SAR

SAR =1: Detect the spindle speed arrival signal.

=0: Do not detect the spindle speed arrival signal.

FLR =1: In the spindle speed fluctuation check, the unit both the allowable rate (q) and fluctuation rate (r) is 0.1%.

=0: In the spindle speed fluctuation check, the unit both the allowable rate (q) and fluctuation rate (r) is 1%.

FLRE =1: Spindle speed fluctuation check is enabled.

=1: Spindle speed fluctuation check is disabled.

PG2, PG1: The gear ratio between the spindle and position encoder. 00 is 1:1; 01 is 2:1; 10 is 4:1 and 11 is 8:1.

Standard setting：0 0 0 0 0 0 0 0

System parameter number

0	3	9									TLC

TLC =1: Select the type of the tool length compensation: Type B.

=0: Select the type of the tool length compensation: Type A.

Standard setting：0 0 0 0 0 0 0 1

System parameter number

0	4	0		ODI					CCN		SUP

SUP =1: The start type in the cutter compensation: Type B.

=0: The start type in the cutter compensation: Type A.

CCN =1: G28, G30 commands are moved to the intermediate point, and then cancel the radius compensation.

=0: G28, G30 commands are moved to the intermediate point, and then retain the radius compensation.

ODI =1: Set the cutter compensation value based upon its diameter value.

=0: Set the cutter compensation value based upon its radius value.

Standard setting：1 0 0 0 0 1 0 0

System parameter number

0	4	1			CNI	G39		PUIT			

PUIT =1: The input and display of the data parameter are determined by bit parameter NO.0#2 INI.

=0: The input and display of the data parameter are metric system.

APPEDIX

G39 =1: In the radius compensation, the corner arc function is enabled.

=0: In the radius compensation, the corner arc function is disabled.

CNI =1: Radius compensation performs the interference inspection.

=0: Radius compensation does not perform the interference inspection.

Standard setting：0 1 1 0 0 0 0 0

System parameter number

0	**4**	**2**			**RD2**	**RD1**				

RD1 =1: Set the tool-retraction direction of G76, G87: Negative.

=0: Set the tool-retraction direction of G76, G87: Positive.

RD2 =1: Set the tool-retraction axis of the G76, G87: The 2nd axis.

=0: Set the tool-retraction axis of the G76, G87: The 1st axis.

Standard setting：0 0 0 0 0 0 0 0

System parameter number

0	**4**	**3**							**QZA**	

QZA =1: In the peck drilling (G73, G83), the alarm will generate if the cutting value does not specify.

=0: In the peck drilling (G73, G83), the alarm will not generate even if the cutting value does not specify.

Standard setting：0 0 0 0 0 0 1 0

System parameter number

0	**4**	**4**			**PCP**	**DOV**			**VGR**	

VGR =1: The gear ratio between the spindle and position encoder can be set arbitrarily.

=0: The gear ratio between the spindle and position encoder can not be set freely.

DOV =1: The override is enabled when the rigid tapping is performed the tool-retraction.

=1: The override is disabled when the rigid tapping is performed the tool-retraction.

PCP =1: The tapping is the high-speed peck tapping cycle.

=0: The tapping is the standard peck tapping cycle.

Standard setting：0 0 0 0 0 0 0 0

System parameter number

0	**4**	**5**				**OVS**	**OVU**	**TDR**		**NIZ**

NIZ =1: Perform the rigid tapping smooth treatment.

=0: Do not perform the rigid tapping smooth treatment.

TDR =1: Rigid tapping in-feed. Use the identical time constante when tool-retraction is performed.

=0: Rigid tapping in-feed. Do not use the identical time constante when tool-retraction is performed.

OVU =1: The unit of the rigid tapping tool-retraction override is 10%.

=0: The unit of the rigid tapping tool-retraction override is 1%.

OVS =1: In the rigid tapping, the feedrate selection and override cancellation signal are enabled.

=0: In the rigid tapping, the feedrate selection and override cancellation signal are disabled.

Standard setting：0 0 0 0 0 0 0 0

System parameter number

0	4	6				ORI				SSOG	

SSOG =1: At the beginning of the tapping, the control method of spindle is servo.

=0: At the beginning of the tapping, the control method of spindle is follow.

ORI =1: At the beginning of the tapping, the spindle dwells.

=0: At the beginning of the tapping, the spindle does not dwell.

Standard setting：0 0 0 0 0 0 0 0

System parameter number

0	4	7			XSC	SCL3	SCL2	SCL1			RIN

RIN =1: The rotation angle of the coordinate rotation: G90/G91 command.

=0: The rotation angle of the coordinate rotation: Absolute command.

SCL1 =1: The scaling of the 1st axis is enabled.

=0: The scaling of the 1st axis is disabled.

SCL2 =1: The scaling of the 2nd axis is enabled.

=0: The scaling of the 2nd axis is disabled.

SCL3 =1: The scaling of the 3rd axis is enabled.

=0: The scaling of the 3rd axis is disabled.

XSC =1: The specification method of the scaling override along each axis is I, J or K.

=0: The specification method of the scaling override along each axis is P code.

Standard setting：0 1 1 1 1 0 0 1

System parameter number

0	4	8									MDL

MDL =1: The unidirectional position G code is set to modal code.

=0 : The unidirectional position G code does not set to modal code.

Standard setting：0 0 0 0 0 0 0 0

System parameter number

0	4	9									RPST

RPST =1: Z axis moves with the G01 mode during the restart of program.

=0: Z axis moves at the dry run speed with the G00 mode during the restart of program.

Standard setting：0 0 0 0 0 0 0 0

System parameter number

0	5	0			SIM		G90			REL	

REL =1: The relative display setting of the index worktable: Within 360°.

=0 : The relative display setting of the index worktable: Without 360°.

G90 =1: Index command: Absolute command.

=0: Index command: G90/G91 command.

SIM =1: The alarm occurs when the index code and other controllable axis codes are share with a same block.

=0: The alarm does not occur when the index code and other controllable axis codes are share with a same block.

Standard setting：0 1 0 0 0 0 0 0

System parameter number

0	5	1		MDLY		SBM					

SBM =1: The single block can be used in the macro program command statement.

=0: The single block can not be used in the macro program command statement.

MDLY =1: Without delay in the macro program command statement.

=1: Delays in the macro program command statement.

Standard setting：0 0 0 0 0 0 0 0

System parameter number

0	5	2		CLV	CCV						

CCV =1: Macro program common variables #100 - #199 will be cleared after

APPEDIX

resetting.

=0: Macro program common variables #100 - #199 will not be cleared after resetting.

CLV =1: Macro program local variables #1 - #50 will be cleared after resetting.

=0: Macro program local variables #1 - #50 will not be cleared after resetting.

Standard setting：0 0 0 0 0 0 0 0

System parameter number

0	**5**	**3**		**PLCV**				**LAD3**	**LDA2**	**LAD1**	**LAD0**

LAD0~LAD3 are the binary combination parameter. Use the No.0 ladder digram when it is set to 0; Use the No. 0~15 ladder diagrams when it set to 1~15.

PLCV =1: Read and display the PLC software version number.

=0: Do not read and display the PLC software version number.

Standard setting：1 0 0 0 0 0 0 1

System parameter number

0	**5**	**4**		**OPRG**							

OPRG =1: One-touch input/output is enabled to the component program when performing the debugging or its above authority.

=0: One-touch input/output is disabled to the component program when performing the debugging or its above authority.

Standard setting：0 0 0 0 0 0 0 0

System parameter number

0	**5**	**5**									**CANT**

CANT =1: Single machining time is automatically cleared.

=0: Single machining time is not automatically cleared.

Standard setting：0 0 0 0 0 0 0 0

System parameter number

0	**5**	**6**		**HNGD**				**HISR**			**HPF**

HPF =1: MPG movement amount selects the absolutely operation.

=0: MPG movement amount does not select the absolutely operation.

HISR =1: Use the MPG/single-step interruption function.

=0: Do not use the MPG/single-step interruption function.

HNGD =1: The movement along each axis is identical with the MPG revolving direction.

APPEDIX

=0: The movement along each axis is different with the MPG revolving direction.

Standard setting：1 0 0 0 0 0 0 1

System parameter number

0	5	7					PLW5	PLW4	PLW3	PLW2	PLW1

PLW1 =1: The 1st axis pulse width changes along with the speed.
=0: The 1st axis pulse width fixes at 1ms.

PLW2 =1: The 2nd axis pulse width changes along with the speed.
=0: The 2nd axis pulse width fixes at 1ms.

PLW3 =1: The 3rd axis pulse width changes along with the speed.
=0: The 3rd axis pulse width fixes at 1ms.

PLW4 =1: The 4th axis pulse width changes along with the speed.
=0: The 4th axis pulse width fixes at 1ms.

PLW5 =1: The 5th axis pulse width changes along with the speed.
=0: The 5th axis pulse width fixes at 1ms.

Standard setting：0 0 0 0 0 0 0 0

System parameter number

0	5	9			LEDT	LOPT				RHPG	

RHPG =1: Use the electric MPG drive function.
=0: Do not use the electric MPG drive function.

LOPT =1: Operation panel locking singal is enabled.
=0: Operation panel locking singal is disabled.

LEDT =1: External program locking signal is enabled.
=0: External program locking signal is disabled.

Standard setting：0 0 0 0 0 0 0 0

System parameter number

0	6	0				SCL				EPW	

EPW =1: The Max. quantity of the position switches are 16.
=0: The Max. quantity of the position switches are 10.

SCL =1: Use the scaling.
=0: Do not use the scaling.

Standard setting：0 0 1 0 0 0 0 0

System parameter number

0	6	1		FALM	LALM	EALM	SALM	AALM			SSC

APPEDIX

SSC =1: Use the constant peripheral speed control.
=0: Do not use the constant peripheral speed control.

AALM =1: Ignor external user alarm.
=1: Do not ignore external user alarm.

SALM =1: Ignore the spindle drive alarm.
=0: Do not ignore the spindle drive alarm.

EALM =1: Ignore the ESP alarm.
=0: Do not ignore ESP alarm.

LALM =1: Ignore hard-limit alarm.
=0: Do not ignore hard-limit alarm.

FALM =1: Ignore feed axis drive alarm.
=0: Do not ignore feed axis drive alarm.

Standard setting：0 0 0 0 0 0 0 0

2. Data Parameter

Parameter No. Parameter definition Default value

0000	I/O channel, input/output equipment (0:Xon/Xoff 1:XModem 2:USB)	2

Setting range：0～2

When CNC is performed the communication with the PC machine by the RS232 interface, set it as the 0 or 1, and it is set to 2 when connecting with the U disk.

0001	Communication channel Baud Rate (DNC)	38400

Setting range：0～115200 （Unit：BPS）

0002	Communication channel Baud Rate (Transmission file)	115200

Setting range：0～115200 （Unit：BPS）

0004	System interporlation period (1, 2, 4, and 8ms)	1

Setting range：1～8

0005	CNC controllable axis number	4

Setting range：3～5

0006	System language selection	0

Setting range：0～3 0: Chinese 1: English 2: Russian 3: Spanish

0008	The dimension of the slave state MDT data package of the Ethernet bus	12

Setting range：0～20

0009	The Max. retransmission times of the Ethernet bus	10

Setting range：0～30

0010	The 1st axis offset value of the external workpiece origin	0.0000

Setting range：-9999.9999～9999.9999（mm）

APPEDIX

0011	The 2nd axis offset value of the external workpiece origin	0.0000

Setting range：-9999.9999～9999.9999（mm)

0012	The 3rd axis offset value of the external workpiece origin	0.0000

Setting range：-9999.9999～9999.9999（mm)

0013	The 4th axis offset value of the external workpiece origin	0.0000

Setting range：-9999.9999～9999.9999（mm)

0014	The 5th axis offset value of the external workpiece origin	0.0000

Setting range：-9999.9999～9999.9999（mm)

0015	The 1st axis workpiece origin offset value of G54	0.0000

Setting range：-9999.9999～9999.9999（mm）

0016	The 2nd axis workpiece origin offset value of G54	0.0000

Setting range：-9999.9999～9999.9999（mm）

0017	The 3rd axis workpiece origin offset value of G54	0.0000

Setting range：-9999.9999～9999.9999（mm）

0018	The 4th axis workpiece origin offset value of G54	0.0000

Setting range：-9999.9999～9999.9999（mm）

0019	The 5th axis workpiece origin offset value of G54	0.0000

Setting range：-9999.9999～9999.9999（mm）

0020	The 1st axis workpiece origin offset value of G55	0.0000

Setting range：-9999.9999～9999.9999（mm）

0021	The 2nd axis workpiece origin offset value of G55	0.0000

Setting range：-9999.9999～9999.9999（mm）

0022	The 3rd axis workpiece origin offset value of G55	0.0000

Setting range：-9999.9999～9999.9999（mm）

0023	The 4th axis workpiece origin offset value of G55	0.0000

Setting range：-9999.9999～9999.9999（mm）

0024	The 5th axis workpiece origin offset value of G55	0.0000

Setting range：-9999.9999～9999.9999（mm）

0025	The 1st axis workpiece origin offset value of G56	0.0000

Setting range：-9999.9999～9999.9999（mm）

0026	The 2nd axis workpiece origin offset value of G56	0.0000

Setting range：-9999.9999～9999.9999（mm）

0027	The 3rd axis workpiece origin offset value of G56	0.0000

Setting range：-9999.9999～9999.9999（mm）

0028	The 4th axis workpiece origin offset value of G56	0.0000

Setting range：-9999.9999～9999.9999（mm）

0029	The 5th axis workpiece origin offset value of G56	0.0000

Setting range：-9999.9999～9999.9999（mm）

0030	The 1st axis workpiece origin offset value of G57	0.0000

Setting range：-9999.9999～9999.9999（mm）

0031	The 2nd axis workpiece origin offset value of G57	0.0000

Setting range：-9999.9999～9999.9999（mm）

0032	The 3rd axis workpiece origin offset value of G57	0.0000

Setting range：-9999.9999～9999.9999（mm）

0033	The 4th axis workpiece origin offset value of G57	0.0000

Setting range：-9999.9999～9999.9999（mm）

0034	The 5th axis workpiece origin offset value of G57	0.0000

Setting range：-9999.9999～9999.9999（mm）

0035	The 1st axis workpiece origin offset value of G58	0.0000

Setting range：-9999.9999～9999.9999（mm）

0036	The 2nd axis workpiece origin offset value of G58	0.0000

Setting range：-9999.9999～9999.9999（mm）

0037	The 3rd axis workpiece origin offset value of G58	0.0000

Setting range：-9999.9999～9999.9999（mm）

0038	The 4th axis workpiece origin offset value of G58	0.0000

Setting range：-9999.9999～9999.9999（mm）

0039	The 5th axis workpiece origin offset value of G58	0.0000

Setting range：-9999.9999～9999.9999（mm）

0040	The 1st axis workpiece origin offset value of G59	0.0000

Setting range：-9999.9999～9999.9999（mm）

0041	The 2nd axis workpiece origin offset value of G59	0.0000

Setting range：-9999.9999～9999.9999（mm）

0042	The 3rd axis workpiece origin offset value of G59	0.0000

Setting range：-9999.9999～9999.9999（mm）

0043	The 4th axis workpiece origin offset value of G59	0.0000

Setting range：-9999.9999～9999.9999（mm）

0044	The 5th axis workpiece origin offset value of G59	0.0000

Setting range：-9999.9999～9999.9999（mm）

0045	The 1st axis coordinate value of the 1st reference point on the mechanical coordinate system	0.0000

Setting range：-9999.9999～9999.9999（mm）

0046	The 2nd axis coordinate value of the 1st reference point on the mechanical coordinate system	0.0000

Setting range：-9999.9999～9999.9999（mm）

0047	The 3rd axis coordinate value of the 1st reference point on the mechanical coordinate system	0.0000

Setting range：-9999.9999～9999.9999（mm）

0048	The 4th axis coordinate value of the 1st reference point on the mechanical coordinate system	0.0000

Setting range：-9999.9999～9999.9999（mm）

0049	The 5th axis coordinate value of the 1st reference point on the mechanical coordinate system	0.0000

Setting range：-9999.9999～9999.9999（mm）

0050	The 1st axis coordinate value of the 2nd reference point on the mechanical coordinate system	0.0000

Setting range：-9999.9999～9999.9999（mm）

0051	The 2nd axis coordinate value of the 2nd reference point on the mechanical coordinate system	0.0000

Setting range：-9999.9999～9999.9999（mm）

0052	The 3rd axis coordinate value of the 3rd reference point on the mechanical coordinate system	0.0000

Setting range：-9999.9999～9999.9999（mm）

0053	The 4th axis coordinate value of the 2nd reference point on the mechanical coordinate system	0.0000

Setting range：-9999.9999～9999.9999（mm）

0054	The 2nd axis coordinate value of the 5th reference point on the mechanical coordinate system	0.0000

Setting range：-9999.9999～9999.9999（mm）

0055	The 1st axis coordinate value of the 3rd reference point on the mechanical coordinate system	0.0000

Setting range：-9999.9999～9999.9999（mm）

0056	The 2nd axis coordinate value of the 3rd reference point on the mechanical coordinate system	0.0000

Setting range：-9999.9999～9999.9999（mm)

0057	The 3rd axis coordinate value of the 3rd reference point on the mechanical coordinate system	0.0000

Setting range：-9999.9999～9999.9999（mm）

0058	The 4th axis coordinate value of the 3rd reference point on the mechanical coordinate system	0.0000

Setting range：-9999.9999～9999.9999（mm）

0059	The 5th axis coordinate value of the 3rd reference point on the mechanical coordinate system	0.0000

Setting range：-9999.9999～9999.9999（mm）

0060	The 1st axis coordinate value of the 4th reference point on the mechanical coordinate system	0.0000

Setting range：-9999.9999～9999.9999（mm）

0061	The 2nd axis coordinate value of the 4th reference point on the mechanical coordinate system	0.0000

Setting range：-9999.9999～9999.9999（mm)

0062	The 3rd axis coordinate value of the 4th reference point on the mechanical coordinate system	0.0000

Setting range：-9999.9999～9999.9999（mm)

0063	The 4th axis coordinate value of the 4th reference point on the mechanical coordinate system	0.0000

Setting range：-9999.9999～9999.9999（mm)

0064	The 5th axis coordinate value of the 4th reference point on the mechanical coordinate system	0.0000

Setting range：-9999.9999～9999.9999（mm)

0066	The coordinate value of the stored stroke detection 1 along the reverse boundary of the 1st axis	-9999

Setting range：-9999.9999～9999.9999（mm)

0067	The coordinate value of the stored stroke detection 1 along the positive boundary of the 1st axis	9999

Setting range：-9999.9999～9999.9999（mm)

0068	The coordinate value of the stored stroke detection 1 along the reverse boundary of the 2nd axis	-9999

Setting range：-9999.9999～9999.9999（mm)

0069	The coordinate value of the stored stroke detection 1 along the positive boundary of the 2nd axis	9999

Setting range：-9999.9999～9999.9999（mm)

0070	The coordinate value of the stored stroke detection 1 along the reverse boundary of the 3rd xis	-9999

Setting range：-9999.9999～9999.9999（mm)

0071	The coordinate value of the stored stroke detection 1 along the positive boundary of the 3rd axis	9999

Setting range：-9999.9999～9999.9999（mm)

0072	The coordinate value of the stored stroke detection 1 along the reverse boundary of the 4th axis	-9999

Setting range：-9999.9999～9999.9999（mm)

0073	The coordinate value of the stored stroke detection 1 along the positive boundary of the 4th axis	9999

Setting range：-9999.9999～9999.9999（mm)

0074	The coordinate value of the stored stroke detection 1 along the reverse boundary of the 5th axis	-9999

Setting range：-9999.9999～9999.9999（mm)

0075	The coordinate value of the stored stroke detection 1 along the positive boundary of the 5th axis	9999

Setting range：-9999.9999～9999.9999（mm)

0076	The coordinate value of the stored stroke detection 2 along the reverse boundary of the 1st axis	-9999

Setting range：-9999.9999～9999.9999（mm)

0077	The coordinate value of the stored stroke detection 2 along the positive boundary of the 1st axis	9999

Setting range：-9999.9999～9999.9999（mm)

0078	The coordinate value of the stored stroke detection 2 along the reverse boundary of the 2nd axis	-9999

Setting range：-9999.9999～9999.9999（mm)

0079	The coordinate value of the stored stroke detection 2 along the positive boundary of the 2nd axis	9999

Setting range：-9999.9999～9999.9999（mm)

0080	The coordinate value of the stored stroke detection 2 along the reverse boundary of the 3rd axis	-9999

Setting range：-9999.9999～9999.9999（mm)

0081	The coordinate value of the stored stroke detection 2 along the positive boundary of the 2nd axis	9999

Setting range：-9999.9999～9999.9999（mm)

APPEDIX

0082	The coordinate value of the stored stroke detection 2 along the reverse boundary of the 4th axis	-9999

Setting range：-9999.9999～9999.9999（mm)

0083	The coordinate value of the stored stroke detection 2 along the positive boundary of the 4th axis	9999

Setting range：-9999.9999～9999.9999（mm)

0084	The coordinate value of the stored stroke detection 2 along the reverse boundary of the 5th axis	-9999

Setting range：-9999.9999～9999.9999（mm)

0085	The coordinate value of the stored stroke detection 2 along the positive boundary of the 5th axis	9999

Setting range：-9999.9999～9999.9999（mm)

0086	Dry run speed	5000

Setting range：0～9999 (mm/min)

0087	The cutting feedrate when the power is turned on.	300

Setting range：0～9999 (mm/min)

0088	G0 rapid traverse rate along the 1st axis	5000

Setting range：

Metric: 0～30000 (mm/min)

Inch: 0～30000/ 25.4（inch/min）

Rotation axis: 0～30000（deg/min）

0089	G0 rapid traverse rate along the 2nd axis	5000

Setting range：

Metric: 0～30000 (mm/min)

Inch: 0～30000/ 25.4（inch/min）

Rotation axis: 0～30000（deg/min）

0090	G0 rapid traverse rate along the 3rd axis	5000

Setting range：

Metric: 0～30000 (mm/min)

Inch: 0～30000/ 25.4（inch/min）

Rotation axis: 0～30000（deg/min）

0091	G0 rapid traverse rate along the 4th axis	5000

Setting range：

Metric: 0～30000 (mm/min)

Inch: 0～30000/ 25.4（inch/min）

Rotation axis: 0～30000（deg/min）

0092	G0 rapid traverse rate along the 5th axis	5000

Setting range：

Metric: 0～30000 (mm/min)

Inch: 0～30000/ 25.4（inch/min）

Rotation axis: 0～30000（deg/min）

0093	The Fo speed (General-purpose for overall axes) at the rapid traverse rate along each axis	30

Setting range：0～1000 (mm/min)

0094	The top controllable speed (General-purpose for overall axes) at the rapid position	8000

Setting range：300～30000(mm/min)

0095	The lowest controllable speed (General-purpose for overall axes) at the rapid position	0

Setting range：0～300 (mm/min)

0096	The top controllable speed (General-purpose for overall axes) for the cutting feed	6000

Setting range：300～9999 (mm/min)

0097	The lowest controllable speed (General-purpose for overall axes) for the cutting feed	0

Setting range：0～300 (mm/min)

0098	The consecutive feedrate of the JOG along each axis	2000

APPEDIX

Setting range：0～9999 (mm/min)

0099	(FL) speed (General-purpose for overall axes) when gaining the Z pulse signal	40

Setting range：1～60 (mm/min)

0100	Reference position speed return along the 1st axis	4000

Setting range：0～9999 (mm/min)

0101	Reference position speed return along the 2nd axis	4000

Setting range：0～9999 (mm/min)

0102	Reference position speed return along the 3rd axis	4000

Setting range：0～9999 (mm/min)

0103	Reference position speed return along the 4th axis	4000

Setting range：0～9999 (mm/min)

0104	Reference position speed return along the 5th axis	4000

Setting range：0～9999 (mm/min)

0105	Acceleration/deceleration L-type time constant before the 1st axis at the rapid traverse rate	100

Setting range：3～400 （ms）

0106	Acceleration/deceleration L-type time constant before the 2nd axis at the rapid traverse rate	100

Setting range：3～400 （ms）

0107	Acceleration/deceleration L-type time constant before the 3rd axis at the rapid traverse rate	100

Setting range：3～400 （ms）

0108	Acceleration/deceleration L-type time constant before the 4th axis at the rapid traverse rate	100

Setting range：3～400 （ms）

0109	Acceleration/deceleration L-type time constant before the 5th axis at the rapid traverse rate	100

Setting range：3～400 （ms）

0110	Acceleration/deceleration S-type time constant before the 1st axis at the rapid traverse rate	100

Setting range：3～400 （ms）

0111	Acceleration/deceleration S-type time constant before the 2nd axis at the rapid traverse rate	100

Setting range：3～400 （ms）

0112	Acceleration/deceleration S-type time constant before the 3rd axis at the rapid traverse rate	100

Setting range：3～400 （ms）

0113	Acceleration/deceleration S-type time constant before the 4th axis at the rapid traverse rate	100

Setting range：3～400 （ms）

0114	Acceleration/deceleration S-type time constant before the 5th axis at the rapid traverse rate	100

Setting range：3～400 （ms）

0115	Acceleration/deceleration L-type time constant after the 1st axis at the rapid traverse rate	80

Setting range：0～400 （ms）

0116	Acceleration/deceleration L-type time constant after the 2nd axis at the rapid traverse rate	80

Setting range：0～400 （ms）

0117	Acceleration/deceleration L-type time constant after the 3rd axis at the rapid traverse rate	80

Setting range：0～400 （ms）

0118	Acceleration/deceleration L-type time constant after the 4th axis at the rapid traverse rate	80

Setting range：0～400 （ms）

0119	Acceleration/deceleration L-type time constant after the 5th axis at the rapid traverse rate	80

Setting range：0～400 （ms）

0120	Acceleration/deceleration E-type time constant after the 1st axis at the rapid traverse rate	60

Setting range：0～400 （ms）

0121	Acceleration/deceleration E-type time constant after the 2nd axis at the rapid traverse rate	60

Setting range：0～400 （ms）

0122	Acceleration/deceleration E-type time constant after the 3rd axis at the rapid traverse rate	60

Setting range：0～400 （ms）

0123	Acceleration/deceleration E-type time constant after the 4th axis at the rapid traverse rate	60

Setting range：0～400 （ms）

0124	Acceleration/deceleration E-type time constant after the 5th axis at the rapid traverse rate	60

Setting range：0～400 （ms）

0125	Acceleration/deceleration L-type time constant before the cutting feed	100

Setting range：3～400 （ms）

0126	Acceleration/deceleration S-type time constant before the cutting feed	100

Setting range：3～400 （ms）

0127	Acceleration/deceleration L-type time constant after the cutting feed	80

Setting range：3～400 （ms）

0128	Acceleration/deceleration E-type time constant after the cutting feed	60

Setting range：3～400 （ms）

0129	Acceleration/deceleration FL speed of the exponential type	10

Setting range：0～9999 （mm/min）

0130	Interpolate the Max. incorporative block numbers in adance	0

Setting range：0～10

0131	Cutting feed position accuracy	0.03

Setting range：0.001～0.5（mm）

0132	Circular interpolation control accuracy	0.03

Setting range：0～0.5（mm）

0133	Interpolate outline control accuracy in advance	0.01

Setting range：0.01～0.5 （mm）

0134	The acceleration of the linear acceleration/deceleration before interpolation in the forecasting control method	250

Setting range：0～2000 （mm/s^2）

0135	The acceleration/deceleration constent before S-type in the forecasting control method.	100

Setting range：0～400 （ms）

0136	The linear acceleration/deceleration time constant followed with the acceleration/deceleration in the	80

APPEDIX

	forecasting control.	

Setting range：0～400 （ms）

0137	The exponential acceleration/deceleration time constant followed with the acceleration/deceleration in the forecasting control.	60

Setting range：0～400 （ms）

0138	The exponential type acceleration/deceleration FL speed of the cutting feed in forcasting control mode.	10

Setting range：0～400 （ms）

0139	Outline control accuracy of forcasting control mode.	0.01

Setting range：0～0.5 (mm)

0140	Merger block section in the forcasting control mode.	0

Setting range：0～10

0141	In-position accuracy in the forcasting control mode.	0.05

Setting range：0～0.5 (mm)

0142	The condition for sampling length in the forcasting control mode.	5

Setting range：0～30

0143	The condition for sampling angle in the forcasting control mode.	10

Setting range：0～30

0144	The critical angle between two blocks of the automatic corner deceleration in the forcasting control mode.	5

Setting range：2～178 (Degree)

0145	The lowest feedrate of automatic corner deceleration in the forcasting control mode	120

Setting range：10～1000 (mm/min)

0146	The allowable error of each axis for the deceleration function with the speed difference mode in the forcasting control mode	80

Setting range：60～1000

0147	Cutting accuracy level in the forcasting control mode	2

Setting range：0～8

0148	The acceleration limit out of the circular arc interpolation	1000

Setting range：100～5000 （mm/s^2）

0149	The low speed lower-limit of the acceleration clamping at the circular arc interpolation	200

Setting range：0～2000 (mm/min)

0150	The acceleration clamped time constant of the cutting feed	50

Setting range：0～1000（ms）

0151	The Max. clamped speed in the MPG incomplete operation mode	2000

Setting range：0～3000 (mm/min)

0152	The linear acceleration/deceleration time constant of MPG	120

Setting range：0～400 （ms）

0153	The exponential acceleration/deceleration time constant of MPG	80

Setting range：0～400 （ms）

0154	The acceleration clamping time constant of MPG	100

Setting range：0～400 （ms）

APPEDIX

0155	The top clamping speed of single feed	1000

Setting range：0～3000 （mm/min）

0156	The linear acceleration/deceleration time constant of JOG feed along each axis	100

Setting range：0～400 （ms）

0157	The exponential acceleration/deceleration time constant of JOG feed along each axis	120

Setting range：0～400 （ms）

0158	The acceleration clamping constant of MPG incomplete operation mode	50

Setting range：0～1000 （ms）

0160	The 1st axis command frequency-multiplication coefficient (CMR)	1

Setting range：1～65536

0161	The 2nd axis command frequency-multiplication coefficient (CMR)	1

Setting range：1～65536

0162	The 3rd axis command frequency-multiplication coefficient (CMR)	1

Setting range：1～65536

0163	The 4th axis command frequency-multiplication coefficient (CMR)	1

Setting range：1～65536

0164	The 5th axis command frequency-multiplication coefficient (CMR)	1

Setting range：1～65536

0165	The 1st axis command frequency-division coefficient (CMD)	1

Setting range：1～65536

0166	The 2nd axis command frequency-division coefficient (CMD)	1

Setting range：1～65536

0167	The 3rd axis command frequency-division coefficient (CMD)	1

Setting range：1～65536

0168	The 4th axis command frequency-division coefficient (CMD)	1

Setting range：1～65536

0169	The 5th axis command frequency-division coefficient (CMD)	1

Setting range：1～65536

0170	The 1st axis manual position at the rapid traverse rate	5000

Setting range：0～30000

0171	The 2nd axis manual position at the rapid traverse rate	5000

Setting range：0～30000

0172	The 3rd axis manual position at the rapid traverse rate	5000

Setting range：0～30000

0173	The 4th axis manual position at the rapid traverse rate	5000

Setting range：0～30000

0174	The 5th axis manual position at the rapid traverse rate	5000

Setting range：0～30000

0175	The program name of the 1st axis	0

Setting range：0～8 (0:X 1:Y 2:Z 3:A 4:B 5:C 6:U 7:V 8:W)

0176	The program name of the 2nd axis	1

Setting range：0～8 (0:X 1:Y 2:Z 3:A 4:B 5:C 6:U 7:V 8:W)

0177	The program name of the 3rd axis	2

Setting range：0～8 (0:X 1:Y 2:Z 3:A 4:B 5:C 6:U 7:V 8:W)

0178	The program name of the 4th axis	3

Setting range：0～8 (0:X 1:Y 2:Z 3:A 4:B 5:C 6:U 7:V 8:W)

0179	The program name of the 5th axis	4

Setting range：0～8 (0:X 1:Y 2:Z 3:A 4:B 5:C 6:U 7:V 8:W)

0180	The grid or reference position offset value of the 1st axis	0

Setting range：0～50

0181	The grid or reference position offset value of the 2nd axis	0

Setting range：0～50

0182	The grid or reference position offset value of the 3rd axis	0

Setting range：0～50

0183	The grid or reference position offset value of the 4th axis	0

Setting range：0～50

0184	The compensation condition when the friction	1

	compensation along Z axis of the machine tool is performed (Default: 1.0)	

Setting range：0～50

0185	The friction compensation method along Z axis of machine tool	1

Setting range：0～50

0: Disabled, 1: Up, 2: Down, 3: Up or Down

0186	The friction compensation value along Z axis of machine tool (mm)	0.5

Setting range：0～0.5

0187	Reverse interval compensation condition along Z axis (Default: 1)	1

Setting range：0～50

0188	Reverse interval compensation accumulation distance along Z axis (Default: 0.02)	0.02

Setting range：0～0.5

0189	Reverse interval compensation condition along Z axis (Default: 1)	0.0100

Setting range：0.0001～1.0000 (mm)

Set $\alpha = p(189) \times 0.0001$, after the feed is reversed; single servo period feed value is more than α, and the reverse interval compensation begins.

Ttherefore, when the excircle outline with bigger radius is machined, the least accuracy should be set for guard against that the compensation position does not deviate position of the pass-quadrant. In the machining curve surface, to avoid each tool path is performed the reverse interval compensation at a fixed position, so that a raised ridge occurs, and therefore, the bigger accuracy should be set to evenly distribute the interval compensation within a specified width.

0190	The reverse interval compensation value of the 1st axis	0.0000

Setting range：

Metric: -0.5～0.5 (mm)

Inch: -0.5～0.5/25.4 (inch)

Rotation axis: -0.5～0.5000 (deg)

0191	The reverse interval compensation value of the 2nd axis	0.0000

Setting range：

Metric: -0.5～0.5 (mm)

Inch: -0.5～0.5/25.4 (inch)

Rotation axis: -0.5～0.5 (deg)

0192	The reverse interval compensation value of the 3rd axis	0.0000

Setting range：

Metric: -0.5～0.5 (mm)

Inch: -0.5～0.5/25.4 (inch)

Rotation axis: -0.5～0.5 (deg)

0193	The reverse interval compensation value of the 4th axis	0.0000

Setting range：

Metric: -0.5～0.5 (mm)

Inch: -0.5～0.5/25.4(inch)

Rotation axis: -0.5～0.5(deg)

0194	The reverse interval compensation value of the 5th axis	0.0000

Setting range：

Metric: -0.5～0.5(mm)

Inch: -0.5～0.5/25.4(inch)

Rotation axis: -0.5～0.5 (deg)

0195	The compensation step length of the 1st axis interval based upon the fixed frequency compensation method	0.0030

Setting range：0～0.5 (mm)

0196	The compensation step length of the 2nd axis interval based upon the fixed frequency compensation method	0.0030

Setting range：0～0.5 (mm)

0197	The compensation step length of the 3rd axis interval based upon the fixed frequency compensation method	0.0030

Setting range：0～0.5 (mm)

0198	The compensation step length of the 4th axis interval based upon the fixed frequency compensation method	0.0030

Setting range：0～0.5 (mm)

0199	The compensation step length of the 5th axis interval based upon the fixed frequency compensation method	0.0030

Setting range：0～0.5 (mm)

0200	The time constant of the reverse interval based upon the speed up/down compensation method	20

Setting range：0～400 （ms）

0201	Reverse interval compensation method	0

Setting range：0～2

0: Modal A, 1: Modal B, 2: Modal C

0202	The acceptance width of completion signal of the M, S and T signals.	0

Setting range：0～9999 （ms）

0203	Output time of resetting signal	200

Setting range：50～400 (ms)

0204	The allowable digit of M code	2

Setting range：1～2

0205	The allowable digit of S code	5

Setting range：1～6

0206	The allowable digit of T code	4

Setting range：1～4

0210	The incremental value of the number when the sequence number is automatically inserted.	10

Setting range：0～1000

0211	Prohibt the beginning number of the tool offset value inputted by MDI	0

Setting range：0～9999

0212	Prohibt the number of the tool offset value inputted by MDI	0

Setting range：0～9999

0214	The limit value of circular arc radius error	0.05

Setting range：0.0001～0.1000 （mm）

0216	Pitch error compensation number of the 1st axis reference position	0

Setting range：0～9999

0217	Pitch error compensation number of the 2nd axis reference position	0

Setting range：0～9999

0218	Pitch error compensation number of the 3rd axis reference position	0

Setting range：0～9999

0219	Pitch error compensation number of the 4th axis reference position	0

Setting range：0～9999

0220	Pitch error compensation number of the 5th axis reference position	0

Setting range：0～9999

0226	The pitch error compensation interval of the 1st axis	5

Setting range：0～9999.9999

0227	The pitch error compensation interval of the 2nd axis	5

Setting range：0～9999.9999

0228	The pitch error compensation interval of the 3rd axis	5

Setting range：0～9999.9999

0229	The pitch error compensation interval of the 4th axis	5

Setting range：0～9999.9999

0230	The pitch error compensation interval of the 5th axis	5

Setting range：0～9999.9999

0231	The reverse interval compensation value of the 1st axis at the rapid traverse rate	0

Setting range：-0.5～0.5

0232	The reverse interval compensation value of the 2nd axis at the rapid traverse rate	0

Setting range：-0.5～0.5

0233	The reverse interval compensation value of the 3rd axis at the rapid traverse rate	0

Setting range：-0.5～0.5

0234	The reverse interval compensation value of the 4th axis at the rapid traverse rate	0

Setting range：-0.5～0.5

0235	The reverse interval compensation value of the 5th axis at the rapid traverse rate	0

Setting range：-0.5～0.5

0236	Circular arc pointed angle disposes parameter 1	0

Setting range：0～5

0237	Circular arc pointed angle disposes parameter 2	0

Setting range：0～5

0238	Circular arc pointed angle disposes parameter 3	0

Setting range：0～5

0240	The gain adjustment data of the spindle speed analog output	1

Setting range：0.98～1.02

0241	The compensation value of the offset voltage for the spindle speed analog output	0

Setting range：-0.2～0.2

0242	The spindle speed in the spindle orientation or JOG	50

Setting range：0～9999 (r/min)

0243	The Max. setting value of the frequency-converter	8191

Setting range：4000～8191

0246	The spindle Max. speed corresponding to the gear 1	6000

Setting range：0～99999 (r/min)

0247	The spindle Max. speed corresponding to the gear 2	6000

Setting range：0～99999 (r/min)

0248	The spindle Max. speed corresponding to the gear 3	6000

Setting range：0～99999 (r/min)

0250	The motor speed in the spindle gear shifting	50

Setting range：0～1000 (r/min)

0251	The top motor speed in the spindle gear shifting	6000

Setting range：0～99999 (r/min)

0254	The axis regards as the count reference in the surface speed control	0

Setting range：0～4

0255	The lowest speed in the constant surface speed control (G96)	100

Setting range：0～9999 (r/min)

0257	Spindle upper-limit speed in tapping cycle	2000

Setting range：0～5000 (r/min)

0258	Spindle upper-limit speed	6000

Setting range：0～99999 (r/min)

0261	Spindle encoder linear number	1024

Setting range：0～9999

0262	Spindle override lower-limit value	0.5000

Setting range：0.5～1

0266	The limit value is ignored moving along the external side of the corner in the cutter compensation C	0

Setting range：0～9999.9999

0267	The top value of the cutter wear compensation value	400.0000

Setting range：0～999.9999 (mm)

0268	The Max. error value of the cutter compensation C	0.0010

Setting range：0.0001~0.0100

0269	The cutter radius coefficient with helix in groove cycle	1.5000

Setting range：0.0100～3.0000

0270	The clearance of G73 in high-speed peck cycle	2.0000

Setting range：0～999.9999 (mm)

0271	The clearance value of G83 in canned cycle	2.0000

Setting range：0～999.9999 (mm)

0281	The least dwell time at the bottom of a hole	250

Setting range：0～1000 (ms)

0282	The top dwell time at the bottom of a hole	9999

Setting range：1000～9999 (ms)

0283	The override value in the rigid tapping tool-retraction	100

Setting range：0～100

Note: When the override value of the bit 4 of parameter N.:44 =1 is enabled.

When the bit of parameter No.: 45 equals to 1, the data unit to be set is regarded as 10%, up to 1000% override value can be set.

0284	The retraction value or clearance value in the peck tapping cycle	0

Setting range：0～100 (mm)

0286	The gear's number at the side of the spindle (the 1st gear)	1

Setting range：1～999

0287	The gear's number at the side of the spindle (the 2nd gear)	1

Setting range：1～999

0288	The gear's number at the side of the spindle (the 3rd	1

APPEDIX

	gear)	

Setting range：1～999

0290	The gear's number at the side of the position encoder side (the 1st gear)	1

Setting range：1～999

0291	The gear's number at the side of the position encoder side (the 2nd gear)	1

Setting range：1～999

0292	The gear's number at the side of the position encoder side (the 3rd gear)	1

Setting range：1～999

0294	The top speed of the spindle in rigid tapping (the 1st gear)	6000

Setting range：0～9999 (r/min)

0295	The top speed of the spindle in rigid tapping (the 2nd gear)	6000

Setting range：0～9999 (r/min)

0296	The top speed of the spindle in rigid tapping (the 3rd gear)	6000

Setting range：0～9999 (r/min)

0298	The linear acceleration/deceleration time constant of the spindle and tapping axis (the 1st gear)	200

Setting range：0～9999 （ms）

0299	The linear acceleration/deceleration time constant of the spindle and tapping axis (the 2nd gear)	200

Setting range：0～9999 （ms）

0300	The linear acceleration/deceleration time constant of	200

APPEDIX

	the spindle and tapping axis (the 3rd gear)	

Setting range：0～9999 （ms）

0302	The time constant both the spindle and tapping axis in tool-retraction (the 1st gear)	200

Setting range：0～9999 （ms）

0303	The time constant both the spindle and tapping axis in tool-retraction (the 2nd gear)	200

Setting range：0～9999 （ms）

0304	The time constant both the spindle and tapping axis in tool-retraction (the 3rd gear)	200

Setting range：0～9999 （ms）

0310	It is regarded as the speed allowance rate (q) when the spindle reaches to the command speed	5

Setting range： 0～1000

0311	Spindle fluctuation rate (r) without issuing the detection alarm of the spindle speed alteration	5

Setting range： 0～1000

0312	Fluctuation amplitude (i) of the spindle speed without the detection alarm of spindle speed alternation	10

Setting range： 0～9999

0313	Spindle speed fluctuation detection time (p)ms from the command speed change to the beginning.	1000

Setting range： 0～99999

0320	The clearance value of spindle in rigid tapping (the 1st gear)	0

Setting range：0～99.9999

0321	The clearance value of spindle in rigid tapping (the	0

APPEDIX

	2^{nd} gear)	

Setting range：0～99.9999

0322	The clearance value of spindle in rigid tapping (the 3^{rd} gear)	0

Setting range：0～99.9999

0323	Spindle command multiplication coefficient (CMR) (the 1^{st} gear)	512

Setting range：0～9999

0324	Spindle command multiplication coefficient (CMR) (the 2^{nd} gear)	512

Setting range：0～9999

0325	Spindle command multiplication coefficient (CMR) (the 3^{rd} gear)	512

Setting range：0～9999

0326	Spindle command frequency-division coefficient (CMD) (the 1^{st} gear)	125

Setting range：0～9999

0327	Spindle command frequency-division coefficient (CMD) (the 2^{nd} gear)	125

Setting range：0～9999

0328	Spindle command frequency-division coefficient (CMD) (the 3^{rd} gear)	125

Setting range：0～9999

0329	The used rotation angle in the coordinate rotation when the rotation angle command does not perform.	0

Setting range：0～9999.9999

0330	The used scaling override when using without the	1

	scaling override command	

Setting range：0.0001～9999.9999

0331	The 1st axis scaling override	1

Setting range：0.0001～9999.9999

0332	The 2nd axis scaling override	1

Setting range：0.0001～9999.9999

0333	The 3rd axis scaling override	1

Setting range：0.0001～9999.9999

0334	The dwell time in the unidirection orientation	0

Setting range：0～10(S)

0335	The unidirection orientation and overtravel value along the 1st axis	0

Setting range：-99.9999～99.9999

0336	The unidirection orientation and overtravel value along the 2nd axis	0

Setting range：-99.9999～99.9999

0337	The unidirection orientation and overtravel value along the 3rd axis	0

Setting range：-99.9999～99.9999

0338	The unidirection orientation and overtravel value along the 4th axis	0

Setting range：-99.9999～99.9999

0339	The unidirection orientation and overtravel value along the 5th axis	0

Setting range：-99.9999～99.9999

0341	The buffering area dimension of ARM interpolation point	36

Setting range：0～99999

0342	The low speed of the 1st axis zero return	200

Setting range：0～1000

0343	The low speed of the 2nd axis zero return	200

Setting range：0～1000

0344	The low speed of the 3rd axis zero return	200

Setting range：0～1000

0345	The low speed of the 4th axis zero return	200

Setting range：0～1000

0346	The low speed of the 5th axis zero return	200

Setting range：0～1000

0347	The absolute position of the 1st axis reference position when using the absolute rotation encoder.	65000

Setting range：0～131071

0348	The absolute position of the 2nd axis reference position when using the absolute rotation encoder.	65000

Setting range：0～131071

0349	The absolute position of the 3rd axis reference position when using the absolute rotation encoder.	65000

Setting range：0～131071

0350	The absolute position of the 4th axis reference position when using the absolute rotation encoder.	65000

Setting range：0～131071

0351	The absolute position of the 5th axis reference	65000

	position when using the absolute rotation encoder.	

Setting range：0～131071

0352	Acceleration/deceleration time constant with high-speed in zero return	60

Setting range：3～400

0353	Acceleration/deceleration time constant with low-speed in zero return	100

Setting range：3～400

0354	Failure times of DSP start	0

Setting range：0～999999

0355	Successful times of system start	0

Setting range：0～999999

0356	The machined components	0

Setting range：0～9999

0357	The total components to be machined	0

Setting range：0～9999

0358	The accumulation value (Hours) of power-on time	0

Setting range：0～99999

0359	The accumulation value (Days) of power-on time	0

Setting range：0～99999

0360	The accumulation value of cutting time (Hours)	0

Setting range：0～99999

0361	Backup the connection state (Unchangeable) without MDT package of the Ethernet	0

Setting range：0～20

0362	Backup DSP scan counter (Unchangeable)	78

Setting range：0～1000

0363	Backup the invalid MDT package counter (Unchangeable) of the Ethernet	0

Setting range：0～99999

0364	Backup the consecutive times (Unchangeable) without MDT package of the Ehternet	0

Setting range：0～99999

0365	Backup the connection state (Unchangeable) without MDT package of the Ehternet	0

Setting range：0～99999

0371	The reverse position tolerance along the 1st axis	0.0150

Setting range：0～99.9999 (mm)

0372	The reverse position tolerance along the 2nd axis	0.0150

Setting range：0～99.9999 (mm)

0373	The reverse position tolerance along the 3rd axis	0.0150

Setting range：0～99.9999 (mm)

0374	The reverse position tolerance along the 4th axis	0.0150

Setting range：0～99.9999 (mm)

0375	The reverse position tolerance along the 5th axis	0.0150

Setting range：0～99.9999 (mm)

When the reverse interval compensation values (P0190---P0193) set by one axis is more than the inversion position tolerances (P0371---P0374)set by this axis, one single-unit end speed before the reverse interval compensation beginning of this axis reduces to the lowest speed, so that the other axes within the interval compensation period moves a lesser position to ensure that the compound path is lesser deviated from the true path.

0376	Each axis sequence moves to the program restart position	12345

Setting range：0～99999

0380	Set the synchronized axis with the 4th one 0: no synchronization with any axis 1: the 1st axis 2: the 2nd axis 3: the 3rd axis	0

Setting range：0～3

0381	The top allowable error among the synchronization axes	200

Setting range：0～10000

0382	Set the D-value of dual-drive reference position	0.0000

Setting range：0.0000～2000.0000

0387	The position value of tool-setter in G53 along the 1st axis	0

Setting range：-999.9999～999.9999

0388	The position value of tool-setter in G53 along the 2nd axis	0

Setting range：-999.9999～999.9999

0389	The position value of tool-setter in G53 along the 3rd axis	0

Setting range：-999.9999～999.9999

0390	The estimated length from the current tool point to the fixture	0

Setting range： 0.0000～999.9999

0391	The diameter of automatic prober	2

Setting range： 0.5000～999.9999

0392	The movement distance of the servo optimization	50

Setting range: 0～100

0393	The movement rate of the servo optimization	2000

Setting range: 0～5000

0394	Coordinate system back up along 1st axis	0

Setting range: -9999.9999～9999.9999

0395	Coordinate system back up along 2nd axis	0

Setting range: -9999.9999～9999.9999

0396	Coordinate system back up along 3rd axis	0

Setting range: -9999.9999～9999.9999

0397	Coordinate system back up along 4th axis	0

Setting range: -9999.9999～9999.9999

0398	Coordinate system back up along 5th axis	0

Setting range: -9999.9999～9999.9999

0399	Interpolation step-length times	1.5

Setting range: 1.0000～10.0000

0400	Shape matched parameter	20

Setting range: 0.0020～99.0000

The shap matched parameter (#400) is performed the shape optimization based upon the initial spline and the analysis shape error, and control the error within the allowable range.

The greater the parameter is, the greater the shape error is. The smaller the parameter is, the smaller the shape error is.

0401	Shape matched limit	15

Setting range: 1.0000～999.000

The shape matched limit parameter (#401) can be restricted error fluctuation on shape due to the curvature optimization when the speed mating calculation is performed.

0402	Speed matched parameter	1

Setting range：0.0020～99.0000

Speed matched parameter (#402) is the curvature on curve which shows the radial-shape along the normals of each point on the curve to optimize the curvature and smooth the speed.
The bigger the parameter is, the lower the curvature optimization is, and the greater the machining speed is, the shorter the machining time is.
The smaller the parameter is, the higher the curvature optimization is, the longer the machining time is.

0403	The fitting section number of small-line-block	7

Setting range：0.0020～999.0000

This parameter (#403) determines the cutter position points of the fitting spline curve, which should be controlled within a certain range.
#403 = 1~10 the greater the parameter is, the more the calculation value is, the less the shape error is.
the smaller the parameter is, the smaller the calculation value is, the greater the shape error is.

0404	Spline coefficient n1	30

Setting range：1.0000～199.0000

0405	Spline coefficient n2	30

Setting range：1.0000～199.0000

0406	Spline coefficient n3	30

Setting range：1.0000～199.0000

Fit a piece of initial spline curves for 3 times based upon the spline coefficients n1,n2,n3 (#404, #405, #406); the greater the spline coefficients are, the greater the curve errors are, however the speed is smooth and the machine operates stably. The least the coefficients are, the least the curve errors are, but the speed does not smooth and the machine viberates. The spline coefficient n3 (#406) is reversed.

0407	System internal parameter 1	0.6000

Setting range：0.0020～99.0000

0408	System internal parameter 2	0.6000

Setting range：0.0020～99.0000

0409	Pre-read the smooth treatment control	2.0000

Setting range：0.0000～30.0000

The parameter of the pre-reading treatment control is pre-read the machining shape in advance, automatically calculate the tendency of the integral shape, so reduce machining scars due to the program error generated from the CAM.

0: Close the pre-reading smooth treatment function

1: Perform the smooth treatment based upon the length

2: Perform the smooth treatment based upon the integrated relationships between length and angle

0410	Accuracy smooth balance coefficient	10.0000

Setting range：0.0000～10.0000

If you want to carry out the high-accuracy control, the setting value can only be set for the parameter accuracy smooth balance coefficient (#410), this parameter can be controlled the machining level. Total 11 levels (0-10);

#410 = 0: High accuracy control, strictly control the in-position accuracy, regardless of the smooth, it is enable to program for machining the meticulous edge type (for example: word).

=1-10: Return to the high-speed accuracy contro. The lower the level is, the better the accuracy is; the higher the lever is, the better the smooth is.

Adjust this parameter to reach an ideal effect.

0411	Spline shape control coefficient	10.0000

Setting range：0.0000～10.0000

0412	Fitting accuracy control for small-line-block	-1.0000

Setting range：-10.0000～50.0000

0413	Roundness smooth control coefficient n1	3.0000

Setting range：0.0000～50.0000

0414	Roundness smooth control coefficient n2	0.0000

Setting range：0.0000～50.0000

0444	The Max. allowable error between the machine coordinate and absolute encoder position of each axis	50

Setting range：0～500

0445	Axis 1 configures griding accuracy	0.0010

Setting range：0～10

0446	Axis 2 configures griding accuracy	0.0010

Setting range：0～10

0447	Axis 3 configures griding accuracy	0.0010

Setting range：0～10

0448	Axis 4 configures griding accuracy	0.0010

Setting range：0～10

0449	Axis 5 configures griding accuracy	0.0010

Setting range：0～10

0450	Machine stroke detection: the absolute position of the 1st axis encoder along negative direction boundary	0.0000

Setting range：-99999.999～99999.999

0451	Machine stroke detection: the absolute position of the 1st axis encoder along positive direction boundary	0.0000

Setting range：-99999.999～99999.999

0452	Machine stroke detection: the absolute position of the 2nd axis encoder along negative direction boundary	0.0000

Setting range：-99999.999～99999.999

0453	Machine stroke detection: the absolute position of	0.0000

	the 2nd axis encoder along positive direction boundary	

Setting range：-99999.999～99999.999

0454	Machine stroke detection: the absolute position of the 3rd axis encoder along negative direction boundary	0.0000

Setting range：-99999.999～99999.999

0455	Machine stroke detection: the absolute position of the 3rd axis encoder along positive direction boundary	0.0000

Setting range：-99999.999～99999.999

0456	Machine stroke detection: the absolute position of the 4th axis encoder along negative direction boundary	0.0000

Setting range：-99999.999～99999.999

0457	Machine stroke detection: the absolute position of the 4th axis encoder along positive direction boundary	0.0000

Setting range：-99999.999～99999.999

0458	Machine stroke detection: the absolute position of the 5th axis encoder along negative direction boundary	0.0000

Setting range：-99999.999～99999.999

0459	Machine stroke detection: the absolute position of the 5th axis encoder along positive direction boundary	0.0000

Setting range：-99999.999～99999.999

Appendix Two Alarm List

Alarm No.	Content	Remark
0000	The parameter must to be cut the power once that is altered.	
0001	Fail to open the file	
0002	The MDI data exceeds its range.	
0003	The copied or renamed program No. exists.	
0004	A number or the sign "_" was input without an address at the beginning of a block. Modify the program	
0005	The address was not followed by the appropriate data but was followed by another address or EOB code. Modify the program.	
0006	Sign "-" input error (Sign "-" was input followed with an address which it can not be used the "-" or two or more "-" signs were input). Modify the program	
0007	Sign "." input error (Sign "." was input in an address which it can not be used the "." or two or more "." signs were input). Modify the program	
0008	Program file excessive big. It is better to transfer by DNC.	
0009	Illegal address input. Modify the program	
0010	An unusable G code is used or a G code without this function is commanded. Modify the program.	
0011	Feedrate was not commanded to a cutting feed or the feedrate was inadequate. Modify the program.	
0012	Inadequate disc space. Fail to create a new file or add file content.	
0013	Fail to create a new program due to the program file numbers are reached to the upper-limit.	
0014	It can not specify the G95, without supporting from spindle	
0015	The allowed simultaneously controlled axes are exceeded.	
0016	The current pitch error compensation exceeds its range.	
0017	Inadequate authority modification, input the corresponding password to the password interface.	
0018	Do not modify the null and local variable. G10 is only altered the parameter for the user level.	
0019	Without scaling function, if you want to use it, it is better to open it by altering the bit parameter 60.5.	
0020	In circular interpolation (G02 or G03), the distance between start and arc	

APPEDIX

	center is different from the one between the end and arc center, the D-value exceeds the value specified in the data parameter 214.	
0021	An axis not included in the selected plane (by using G17,G18,G19) was commanded in circular interpolation. Modify the program.	
0022	In the circular interpolation, neither R (Specifying an arc radius) nor I, J and K (Specifying the distance from a start point to the center) is specified.	
0023	In the arc interpolation, I, J, K and R are simultaneously specified.	
0024	Helical interpolation, helical angle is set to 0; the system does not be treated. Modify the program.	
0025	G12 can not be shared a same block with G command.	
0026	The file format of which the system does not support. The file is enormous or the line length of the file exceeds 1024 bytes.	
0027	Length cutter compensation command can not be shared a same block with G92. Modify the program.	
0028	In the plane selection command, two or more axes on the same diretion are commanded. Modify the program.	
0029	The compensation value specified by D/H code is excessive large. Modify the program	
0030	The tool length compensation number or the cutter compensation number specified by D/H is excessive large. Additionally, the workpiece coordinate system number specified by P code is also too large. Modify the program	
0031	When G10 sets the offset amount, workpiece coordinate system, external workpiece coordinate system and additional workpiece coordinate system, the specified P value is excessive big or does not be specified.	
0032	In setting an offset amount by G10 or in writing an offset amount by system variables, the offset amount was excessive or does not be specified. Modify the program.	
0033	A intersection point can not be determined in a cutter compensation C or a chamfering. Modify the program.	
0034	The cutter compensation can not be set up or cancelled in the arc command. Modify the program.	
0036	Skip cutting (G31) was specified in cutter compensation mode. Modify the program.	
0037	The plane selected by using the G17, G18 or G19 is changed in cutter compensation C mode. Modify the program	
0038	Overcutting will occur in cutter compensation C because the arc start point or end coincides with the arc center. Modify the program.	
0039	The cutter nose positioning in the cutter compensation C is incorrect.	

0040	The workpiece coordinate system can not be converted in cutter compensation C. The coordinate system conversion can only be performed after the cutter compensation is cancelled.	
0041	Overcutting will occur and interference generates in cutter compensation C. Modify the program.	
0042	In the cutter compensation, more than 10 blocks without movement command but the dwell command are consecutively performed. Modify the program.	
0043	The authority can be changed at the password interface if the authority is absent.	
0044	One of G27, G28, G29 or G30 is commanded in canned cycle mode. Modify the program	
0045	In the canned cycle G73/G83, a depth for each cutting (Q) is not specified or Q value is regarded as 0. Modify the program.	
0046	The commands other than P2, P3 and P4 are specified in the 2nd, 3rd and 4th reference position return command.	
0047	Firstly perform the machine zero before executing the commands such as the G28, G30 and G53, etc.	
0048	In the canned cycle, Z plane should be higher than the R plane.	
0049	In the canned cycle, Z plane should be lower than the R plane.	
0050	The position should be moved when changing the canned cycle method.	
0051	Improper movement operation or move distance is specified in block followed with the chamfering round angle or chamfering oblique angle. Modify the program.	
0052	The canned cycle in milling groove can not be used the image function. Modify the program.	
0053	Incorrect chamfering oblique angle or chamfering round angle command format. Modify the program.	
0054	DNC transmission error	
0055	Fail to complete the chamfering movement	
0056	M99 can not be shared a same block with the macro program command (G65). Modify the program.	
0057	Fail to write the file. It is necessary to restart after the power is turned off.	
0058	In an arbitrary chamfering oblique angle or chamfering round angle block, a specified axis is not within the selected plane. Modify the program.	
0059	In an external program number search, a specified program number was not found or a specified program is being edited in background. Check the program number and external signal. Or discontinue the background editing.	

0060	A specified sequence number was not found in the sequence number search. Check the sequence number.	
0061	The 1st axis is out of the reference position	
0062	The 2nd axis is out of the reference position	
0063	The 3rd axis is out of the reference position	
0064	The 4th axis is out of the reference position	
0065	The 5th axis is out of the reference position	
0066	It is necessary to cancel the canned cycle mode before performing the parameter input (G10).	
0067	The setting format does not be supported by G10.	
0068	Fail to open the parameter switch	
0069	The U-disc operation interface should be closed when machining.	
0070	The memory area is insufficient. Delete any unnecessary program, then retry.	
0071	The address to be searched was not found. Or the program with specified program number was not found in program index. Check the data.	
0072	The stored program numbers are exceeded 400. Delete unnecessary programs.	
0073	The program number being specified is used. Change the program number or delete the unnecessary programs.	
0074	The program number is other than 1 to 99999. Change the program number.	
0075	An attempt was made to register a program whose number was protected.	
0076	Address P (Program number) was not commanded in the block which includes an M98. Modify the program.	
0077	The program was called in five folds. Modify the program.	
0078	The program number specified by address P in the block which includes G98 and G65 was not found or the macro program called by M06 does not exist.	
0079	The system is expired, contact the supplier.	
0080	Unreasonable MDI data, the Max. speed is less than the Min. speed; or the lowest speed is more than the top speed.	
0081	Macro program cannot be called the subprogram. Modify the program.	
0084	Button overtime or short-circuit occurs.	
0085	When the data is input to memory by using the series port, overflow occurs. The setting or input/output equipment of Baud rate is incorrect.	
0086	In the canned cycle modal, the system cannot be shifted the panel.	
0087	Alarm No. 0087~0091 are the start points of the reference position return along each axis of which the reference position return can not be performed due to the alarm numbers are too close to the reference position or it is so slowly. The reference position should be departed from the start as far as	

	possible. Alternatively, the reference position should be specified the fast rate for the reference position return.	
0092	G27 command can not return to the reference position.	
0093	Unmatch motor type	
0098	A program restart is performed without the reference position return after Power ON or ESP, and G28 was found in program.	
0100	On the PARAMETER (SETTING) screen, PWE (parameter writing enabled) is set to 1. Set it to 0, then reset the system.	
0101	Power-off memory data disorder, ensure that the position is correct.	
0102	The motor type parameter between the system and drive are inconsistent.	
0103	Bus communication error. Check the reliability of the reticle.	
0104	Machine tool zero setting overtime	
0105	Drive data gain overtime	
0106	The drive is inconsistent with the gear ratio of the system servo parameter.	
0107	The parameters between the drive and system servo are inconsistent.	
0108	Insert the U disk	
0110	The position data exceeds the allowance range. Perform the zero return.	
0111	The calculation is out of the allowable range (-10^{47} to -10^{-29}, 0 and 10^{-29} to 10^{47}).	
0112	Division by zero was specified (including tan90°)	
0113	A function command which cannot be used in custom macro is specified. Modify the program.	
0114	G39 format error. Modify the program.	
0115	The value regarded as the variable can not be specified. Alternatively, the O, N regarded as variable are specified in the user marco program. Modify the program.	
0116	The left side of assignment statement is a variable whose an assignment is inhibited. Modify the program.	
0117	This parameter does not support the G10 on-line modification. Alter the program.	
0118	The nesting of bracket exceeds the upper limit (quintuple). Modify the program.	
0119	M00,M01,M02,M30,M98,M99,M06 commandes can not be shared with a same block with other M codes.	
0120	Partion setting is being recovered.	
0121	The machine tool coordinate and the encoder feedback value are exceed the setting value of the offset.	
0122	The nesting layers of the marco program calling are more than 5. modify the	

	program.	
0123	The transmission and cycle statement can not be used in the MDI current modal and DNC mode.	
0124	Program illegal end, without M30 or M02 or M99 command, or without the end symbol. Modify the program.	
0125	Format error in macro. Modify the program.	
0126	Program cycle failed. Modify the program.	
0127	NC and user macro command statement are existed together. Modify the program.	
0128	The sequence number specified in the branch command was not 0-99999. Or, it cannot be found. Modify the program.	
0129	An address of <Argument assignment> is incorrect. Modify the program.	
0130	An axis control command was given by PLC to an axis controlled by CNC. Or an axis control command was given by CNC to an axis controlled by PLC. Modify the program.	
0131	Five or more alarms have generated in external alarm message. Check the ladder diagram.	
0132	No alarm in external alarm message. Check the PLC.	
0133	The axis command does not be supported by system. Modify the program.	
0135	The index angle of the index worktable does not the multiple of the angle unit. Modify the program.	
0136	In the index worktable, another axis was instructed with the B axis together. Modify the program.	
0137	Skip the sequence number to be transferred by command within the cycle. Modify the program.	
0138	Cycle statement dismatches or skip command enters into the cycle. Modify the program.	
0139	PLC axis control selection error. Modify the program	
0140	The sequence number of the macro command skip is absent.	
0141	MDI current modal and DNC mode does not support the macro command skip.	
0142	The proportion scaling override other than from 1 to 999999 is specified. Scaling magnification is specified in other than 1 – 999999. Correct the scaling setting.	
0143	The scaling results, move distance, coordinate value and circular radius exceed the Max. command value. Correct the program or scaling magnification.	
0144	The coordinate rotation plane should be idencial with the arc or cutter	

	compensation C plane. Modify the program.	
0145	G28 command is already specified when the reference position does not establish yet. Modify the program or alter the bit 3 of parameter No.4 (AZR).	
0148	Automatic corner deceleration rate is out of the settable value of judgement angle. Modify the parameter.	
0160	Acr can only be used by R programming in the polar coordinate mode.	
0161	In the polar coordinate mode, the relevant commands, such as the reference point, plane selection or direction can not be performed.	
0163	In the rotation mode, the relevant G commands, such as the reference point or coordinate system can not be performed.	
0164	In the scaling mode, the relevant G commands such as the reference point or coordinate system can not be performed.	
0165	It is better to speicify the rotation, scaling or G10 command with a single block.	
0166	There is no specification axis when reference point return.	
0167	Intermediate point coordinate excessive big	
0168	The Min. dwell time at the bottom of a hole should be less than the Max. one at the bottom of a hole.	
0170	Cutter compensation does not cancel when entering or retracting eh subprogram.	
0172	P does not the integer, alternatively, P is less than or equals to 0 in the block for calling the subprogram.	
0173	The subprogram calling should be less than 9999 times.	
0175	The canned cycle can only be performed on the G17 plane.	
0176	The spindle speed does not specify before the rigid tapping begins.	
0177	The IO control below the G76 command does not support the orientation function of the spindle. Modify the program or parameter.	
0178	The spindle speed does not specify before the canned cycle begins.	
0181	Illegal M codes	
0182	Spindle speed excessive big or small	
0183	Illegal T codes	
0184	The selected tool is out of the range.	
0185	L is excessive small, the alarm reasons are shown below: 1) L is less than the tool radius in the rectangle groove fine-milling. 2) L is less than 0 in the groove rough-milling.	
0186	L is excessive big, the alarm reasons are shown below: 1) L is more than the cutter diameter in the circle groove rough-milling. 2) L is more than the cutter diameter in the rectangle groove rough-milling.	

	3) L is more than the I witin the rectangle groove in the finish-milling. 4) L is more than the J witin the rectangle groove in the finish-milling.	
0187	The cutter diameter is excessive big, the alarm reasons are shown below: 1) Cutter diameter is more than I in the interior circle groove rough-milling. 2) Cutter radius is more than the I-J in the interior circle groove fine-milling. 3) Cutter radius is more than J in the exterior circle groove fine-milling. 4) Cutter diameter is more than I within the rectangle groove in the fine/rough-milling. 5) Cutter diameter is more than J within the rectangle groove in the fine/rough-milling. 6) Cutter radius is more than U within the rectangle groove in the fine/rough-milling. 7) The cutter coefficient or the D is excessive big in helix. Modify the value of the data parameter No.269 or the radius compensation value.	
0188	U is excessive big, the alarm reasons are shown below: 1) Twice as much as U is more than I in the rectangle groove complex cycle. 2) Twice as much as U is more than J in the rectangle groove complex cycle.	
0189	U is excessive small, which should be more than or equals to cutter radius.	
0190	V is excessive small or does not define, which should be more than 0.	
0191	W is excessive small or does not define, which should be more than 0.	
0192	Qis excessive small or does not define, which should be more than 0.	
0193	I does not define or regards to 0.	
0194	J does not define or regards to 0.	
0195	D does not define or regards to 0.	
0198	In the constant surface cutting speed control, it is incorrect to specify an axis. (Refer to the parameter No.254). The command P of the specified axis is with the illegal data. Modify the program.	
0199	Undefined macro command. Modify the program.	
0200	In the rigid tap, an S value is out of the range or is not specified. The Max. value for S value can be specified by parameter. Change the setting in the parameter or modify the program.	
0201	In the rigid tapping, no F value is specified. Modify the program.	
0202	In the rigid tapping, spindle distribution value is too large.	
0203	In the rigid tapping, the position for a M code (M29) or a S command in program is incorrect. Modify the program.	
0204	M29 should be specified in the G80 modal. Modify the program.	
0205	Rigid tapping signal is not 1 when G84 (or G74) is executed through the M code (M29) is specified. Check the ladder diagram to find the reason.	

0206	Plane changeover was instructed in the rigid mode. Correct the program.	
0207	The specified distance was too short or too long in rigid tapping.	
0208	This command can not be performed in G10 modal, it is better to cancel the G10 modal firstly.	
0209	The scaling/rotation/polar coordinate modal does not support the program restart.	
0210	The program restart file name is inconsistent. Select the correct file name.	
0212	Specify a chamfering or chamfering R, alternatively, there is an extral axis on plane. Modify the program.	
0213	Tool-change marco program does not support the G31 skip. Modify the program.	
0214	Tool-change macro program does support the skip operation.	
0215	Tool-change macro program does not support the dynamic coordinate system modification and cutter compensation.	
0216	The scaling/rotation/polar coordinate does not support the G31 skip. Modify the program.	
0217	The scaling/rotation/polar coordinate does not support skip operation.	
0218	The scaling/rotation/polar coordinate does not support the coordinate system dynamic modification and cutter compensation.	
0219	Tool magazine does not use (fail to open the parameter). Tool-change command M06 can not be used.	
0220	The scaling/rotation/polar coordinate does not support metic/inch input shifting.	
0221	Tool-chang macro program does not support the metic/inch input shifting.	
0224	Reference position return has not been performed before the automatic operation starts.	
0231	Any of the following errors occurred in the specified format at the programmable-parameter input: 1) Address N or R was not entered; 2) Parameter number was not specified; 3) Address P in the bit-parameter input L50 does not specify; 4) N, P and R exceed its range. Modify the program.	
0232	Three or more axes were specified as the helical interpolation axes.	
0233	Other operations are being used the equipment connected with the RS-232-C interface.	
0235	The record end symbol (%) was specified.	
0236	The parameter for specifying program restart is not set correctly.	
0237	The command that should be specified the decimal point does not specify the	

APPEDIX

	decimal point.	
0238	The same address appears more than once in a block. Alternatively, a block contains two ore more G codes belonging to the same group.	
0239	An illegal G code is specified at the pre-read control method; an index axis is specified in the pre-treat control method; the Max. cutting feed parameter sets to 0; the parameter of acceleration/deceleration before interpolation sets to 0. Correct the setting parameter.	
0241	MPG pulse abnormality	
0242	Bus connection error	
0250	Axis name repeated. Modify the parameters No.: 175~179	
0251	ESP alarm, after the alarm cancels, it is better to return the zero.	
0252	Program illegal end (CNC transmission velocity is slow, reduce the feedrate)	
0261	DSP interpolation axis pulse command speed is excessive big, it is better to perform the zero return by resetting.	
0262	DSP alarm DSP does not start; restart it again.	
0263	DSP alarm DSP parameter setting error	
0264	DSP alarm Delivery data are excessive big; restart it again.	
0265	DSP alarm Disconnect with the bus, or fail to initialize the bus.	
0266	The speed of DSP interpolation axis exceeds 200m/min.; Zero return is performed again after controlling the reset button.	
0267	DSP initialization mark (5555) abnormality; Zero return is performed again after controlling the reset button.	
0268	The output value of the DSP unit period pulse is excessive big; Zero return is performed again after controlling the reset button.	
0269	DSP internal alarm; Zero return is performed again after controlling the reset button.	
0270	That the DSP evenly divides the interpolation point length is excessive small.	
0271	That the DSP accepts the interpolation data is excessive small; zero return is performed again after pressing the ESP.	
0272	DSP receives the unidendified G code.	
0273	DSP hardware data interchange abnormality (Command type)	
0274	DSP hardware data interchange abnormality (Data type)	
0275	The interpolation velocity multimplication is 0 in the high speed mode.	
0280	Each axis should be set to 0 firstly when using the tool-setting function.	
0281	Shift to the [SETTING] [TOOL-SETTING MIDDLE] interface when using the tool-setting function.	
0282	Check whether the tool setter is installed or set the bit parameter 1.6 to 1.	
0283	Z axis exceeds the safety position, check the tool setter or tool length setting.	

APPEDIX

0286	Automatic tool length measure error, perform it again.	
0401	Drive alarm 01: Servo motor speed exceeds the setting value.	
0402	Drive alarm 02: Main circuit power voltage excessive high	
0403	Drive alarm 03: Main circuit power voltage excessive low	
0404	Drive alarm 04: The numerical value of the position error counter exceeds the setting value	
0405	Drive alarm 05: Motor's temperature is excessive high.	
0406	Drive alarm 06: Speed regulator is saturation for a long time.	
0407	Drive alarm 07: CCW, CW drive prohibition input are OFF.	
0408	Drive alarm 08: The absolute value of the numerical value of the position error counter exceeds 230.	
0409	Drive alarm 09: Encoder signal error	
0410	Drive alarm 10: Control power ±15V is lower.	
0411	Drive alarm 11: IPM intelligent module fault	
0412	Drive alarm 12: Motor's current excessive big	
0413	Drive alarm 13: Servo drive and motor overloading (Instantaneously heat)	
0414	Drive alarm 14: Brake circuit fault	
0415	Drive alarm 15: Encoder counter abnormality	
0420	Drive alarm 20: EEPROM error	
0430	Drive alarm 30: Encoder Z pulse error	
0431	Drive alarm 31: Encoder UVW signal error or unmatch with the encoder.	
0432	Drive alarm 32: UVW signal exist full-high-level or full-low-level	
0433	Drive alarm 33: Communication interruption	
0434	Drive alarm 34: Encoder speed abnormality	
0435	Drive alarm 35: Encoder state abnormality	
0436	Drive alarm 36: Encoder count abnormality	
0437	Drive alarm 37: Encoder single-loop count overflow	
0438	Drive alarm 38: Encoder multi-loop count overflow	
0439	Drive alarm 39: Encoder battery alarm	
0440	Drive alarm 40: Encoder battery power-shortage	
0441	Drive alarm 41: Unmatched motor type	
0442	Drive alarm 42: Absolute position data abnormality alarm	
0443	Drive alarm 43: Encoder EPPROM verification alarm	
0449	Fail to initialize the Ethernet. Check the hardware	
0450	Drive OFF, check whether the hardware is correctly connected.	
0451	The 1st axis drive alarm	
0452	The 2nd axis drive alarm	

0453	The 3rd axis drive alarm	
0454	The 4th axis drive alarm	
0455	The 5th axis drive alarm	
0456	Spindle drive alarm	
0500	The 1st axis soft limit overtravel along -: (Manual or MPG releasing along +)	
0501	The 1st axis soft limit overtravel along -: (Manual or MPG releasing along -)	
0502	The 2nd axis soft limit overtravel along -: (Manual or MPG releasing along +)	
0503	The 2nd axis soft limit overtravel along -: (Manual or MPG releasing along -)	
0504	The 3rd axis soft limit overtravel along -: (Manual or MPG releasing along +)	
0505	The 3rd axis soft limit overtravel along -: (Manual or MPG releasing along -)	
0506	The 4th axis soft limit overtravel along -: (Manual or MPG releasing along +)	
0507	The 4th axis soft limit overtravel along -: (Manual or MPG releasing along -)	
0508	The 5th axis soft limit overtravel along -: (Manual or MPG releasing along +)	
0509	The 5th axis soft limit overtravel along -: (Manual or MPG releasing along -)	
0510	The 1st axis hard limit overtravel along -: (Overtravel releasing, Manual or MPG releasing along +)	
0511	The 1st axis hard limit overtravel along -: (Overtravel releasing, Manual or MPG releasing along -)	
0512	The 2nd axis hard limit overtravel along -: (Overtravel releasing, Manual or MPG releasing along +)	
0513	The 2nd axis hard limit overtravel along -: (Overtravel releasing, Manual or MPG releasing along -)	
0514	The 3rd axis hard limit overtravel along -: (Overtravel releasing, Manual or MPG releasing along +)	
0515	The 3rd axis hard limit overtravel along -: (Overtravel releasing, Manual or MPG releasing along -)	
0516	The 4th axis hard limit overtravel along -: (Overtravel releasing, Manual or MPG releasing along +)	
0517	The 4th axis hard limit overtravel along -: (Overtravel releasing, Manual or MPG releasing along -)	
0518	The 5th axis hard limit overtravel along -: (Overtravel releasing, Manual or MPG releasing along +)	
0519	The 5th axis hard limit overtravel along -: (Overtravel releasing, Manual or MPG releasing along -)	
0600	Disconnect the operation keyboard. Check the operation keyboard connection cable	
1001	The address of relay or coil does not set.	
1002	The function code of input code is absent.	

1003	Function command COM does not use correctly, its corresponding relationship between COM and COME is incorrect, alternatively, the function command is used between them.	
1004	User ladder diagram exceeds the Max. allowable lines or steps. (Troubleshooting) reduce the compiled NET numbers.	
1005	Function commands END1 and END2 are absent; or error in the END1 and END2; or END1 and END2 are incorrect.	
1006	Illegal output in the network. Check the output format.	
1007	PLC is without communication due to the hardware fault or the system interruption error. It is necessary to contact the system equipment manufacturer.	
1008	Function code does not correctly connected.	
1009	Fail to connect the network horizontal.	
1010	The network is being compiled is lost because the the power is turned off while the ladder diagram is edited.	
1011	The address or data is inconsistent with the format of this function command. Input again.	
1012	The address or data does not correctly input. Input again.	
1013	Illegal characters are specified or the data are exceeded its range.	
1014	CTR address repeated. Select the unused CTR address again.	
1015	Function command JMP does not used correctly, the corresponding relationships between JMP and LBL are incorrect. Alternatively, the JMP function command is used again between JMP and LBL.	
1016	Incompleted network structure. Change the ladder diagram	
1017	The network structure does not support at present. Change the ladder diagram.	
1019	TMR address repeated. Select the unused TMR address again.	
1020	The parameter is absent in function command. Input the legal parameter	
1021	PLC performance is overtime, the system will then stop the PLC automatically. Check the ladder diagram logic and eliminate the dead-cycle or excessive repeated calling.	
1022	Function command name is absent. Correctly input the function command name.	
1023	The address or constant of the function command exceeds its range.	
1024	The unnecessary relay or coil exists. Delete the unwanted connection	
1025	Function command correct output	
1026	The line numbers of network connection are exceeded its support range. Change the ladder diagram.	

1027	The same output address is used in another position. Reselect the unused the output address.	
1028	Ladder diagram file format error	
1029	The ladder diagram being used is lost.	
1030	There is incorrect vertical line in the network. Delete the vertical line.	
1031	User data area is already full, reduce the data table capacity of the COD codes.	
1032	The 1st level of the ladder diagram is excessive big, which can not be immediately performed it. Reduce the 1st level ladder diagram.	
1033	Function command SFT exceeds the Max. allowable use number. Reduce the use amount.	
1034	Function command DIFU/DIFD address repeated. Reselect the address.	
1039	The command or network are not within the performable range. Clear it.	
1040	Fail to correcet use the function command CALL or SP; the corresponding relationships between the CALL and SP or the SP and SPE are incorrect. Alternatively, either the SP function command is used again between the SP and SPEC, or the SP is set before using the END2.	
1041	The horizontal breakover cable is parallel with the node network.	
1042	Fail to load the PLC system parameter file.	