This User Manual describes all items concerning the operations of the GSK218MC Machining Center CNC System in detail as much as possible. However, it is impractical to give particular descriptions of all unnecessary or unallowable operations for this series system product due to text limit, product specific applications and other causes. Therefore, operations not specified herein shall be considered “impossible” or “unallowable”.

This User Manual is applicable for the following models of CNC systems manufactured by GSK CNC Equipment Co., Ltd.:

Series	Product type	Structure	LED dimension	Description
GSK218MC	GSK218MC	Integral	10.4	The LED dimension is 10.4 inch by default
	GSK218MC-H	Horizontal	8.4	The LED dimension is 8.4 inch by default
	GSK218MC-H2	Horizontal	10.4	The LED dimension is 10.4 inch by default
	GSK218MC-V	Vertical	10.4	The LED dimension is 10.4 inch by default

GSK218MC, GSK218MC-H and GSK218MC-V have communication interfaces of RS232, USB and Network, which are all provided on the front of their hosts.

Foreword

Dear customers:

It's our pleasure for your patronage and purchase of this GSK218MC Series Machining Center CNC System made by GSK CNC Equipment Co., Ltd.

This manual is the part of the "Programming and Operation Manual" (Software version: V1.5) of the User Manual for GSK218MC Series Machining Center CNC System, which introduces its programming and operation.

In order to guarantee safe, normal and effective operation, the user must carefully read this User Manual before installing and using this product.

Safety Warnings

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

Special prompt: **The system power supply installed on (inside) the cabinet is the one specially provided for GSK CNC system.**

It is prohibited to use this power supply for other purposes.

Otherwise, tremendous hazard may occur!

Safety precautions

■ Transportation and storage

- **Do not pile up the product packaging cartons for more than six layers.**
- **Do not climb, stand on the product packaging carton, and do not place heavy objects on it.**
- **Do not drag or move the product using the cables connected to it.**
- **It is prohibited to collide or scratch the panel and display screen.**
- **Keep the product packaging carton away from damp, sunshine and rain.**

■ Unpacking inspection

- **Confirm whether the product is the one you purchased after unpacking.**
- **Check whether the product is damaged during transportation.**
- **Check the list to see if components are complete or damaged.**
- **In the event of inconsistent product type, missing accessories or damage from transportation, please contact us in time.**

■ Wiring

- **The wiring and inspection personnel must be qualified and skilled professionals.**
- **The product must be reliably grounded with grounding resistance ≤ 0.1Ω. Do not use the neutral line (null line) to replace the ground wire.**
- **The wiring must be correct and firm to avoid product malfunction or unexpected results.**
- **The surge absorption diode must be connected to the product in the prescribed direction. Otherwise, it may damage the product.**
- **Before inserting the plug/unplugging or opening the product cabinet, the power supply must be disconnected.**

■ Overhaul

- **The power supply must be disconnected before maintaining or changing components.**

- **Check the malfunction in the event of short circuit or overloading. The product can only be restarted after troubleshooting.**
- **Do not frequently power on/off the product. After power off, wait for at least 1min before power-on again.**

DECLARATION

- **Despite the description of various contents herein as much as possible, it is unnecessary to cover all the matters that can be or cannot be done due to so many possibilities concerned. Therefore, matters not covered herein can be deemed as "Not Allowed".**

Warning

- **Before installation, programming and operation, the user must carefully read this User Manual and User Instructions from the Machine Manufacturer, and strictly operate the product based on requirements in this Manual and the User Instructions. Otherwise, it may cause damage to the product or machine, work piece scrapping and even personal injury.**

Notes

- **The product functions and technical indexes (such as precision, velocity) described in this User Manual are only for this product. For the CNC machine installed with this product, the actual function configuration and technical performance should be determined by the design of the machine manufacturer, and are subject to the User Instructions of the machine manufacturer.**

This User Manual is subject to change without prior notice.

Volume I Programming Description

Introduce technical specifications, product type spectrum, command code and program format of GSK218M series

Volume II Operating Description

Introduce all items concerning the operations of the GSK218MC Machining Center CNC System

Appendix

Introduce all items concerning the ex-factory standard parameters and alarm table of the GSK218MC Machining Center CNC System

Safety responsibility

Manufacturer's safety responsibility

- The manufacturer should be responsible for the cleared and/or controlled dangers in the design and structure of the CNC system and accessories.

- The manufacturer should be responsible for safety of the CNC system and accessories.

- The manufacturer should be responsible for usage messages and recommendations provided for the user.

User's safety responsibility

- The user should get familiar and master the safety operation of the CNC system through study and training.

- The user should be responsible for the danger caused by adding, changing or modifying the CNC system and accessories without authorization.

- The user should be responsible for the danger caused by operation, adjustment, maintenance, installation and storage in violation of the User Manual.

The User Manual shall be kept by the final user.

Thanks for your support during the usage of GSK product!

CONTENTS

VOLUME I PROGRAMMING DESCRIPTION

APPENDIX

VOLUME I PROGRAMMING DESCRIPTION

Chapter I Overview

1.1 Product Introduction

GSK218MC Machining Center CNC System, the upgraded version of GSK218M, uses the high-speed spline interpolation algorithm, and features greatly improved machining speed, precision, surface gloss and quality.

Newly designed HMI is more attractive, friendly and easy to use. This series of system supports GSK-Link Ethernet bus communication function for easier connection, supports statement macro programs (macro B) for simplified programming, and can be adapted to CNC milling machines, machining centers, high-speed engraving and milling machines, CNC grinding machines, CNC gear hobbing machines and many other CNC machines.

- High speed and high precision, complex surface machining effective speed 8m/min, best machining speed 4m/min
- The maximum positioning speed is 60m/min, and maximum feed speed is 15m/min
- The number of pre-processing segments is up to 1,000, featuring look-ahead function, fast speed, high precision and good finish
- The installation structure can be united, horizontal and vertical, each using 8.4/10.4 inch HD color LCD
- Newly designed HMI is more attractive, friendly and easy to use
- Supports multiple languages including Chinese and English
- Supports PLC online monitoring, editing, compiling, and signal tracking
- Supports various types of tool magazines such as carousel type, disc, and servo
- Supports statement macro programs (macro B), simplifying programming
- Plentiful help and prompt messages, easy to learn, use, and debug
- Supports communication interfaces of RS232, USB, and Network, enables file transfer, DNC processing, and USB online processing
- Supports GSK-Link Ethernet bus function, easy connection, good expandability, supports 17-bit

absolute type encoder, high precision, non-return-to-zero, full closed loop control (optional)

1.2 Technical Specifications

Movement control	Control axis and linkage axis: The standard configuration is four-axis three-linkage, with four-axis four-linkage optional. Each axis can be set to a linear or rotary axis
	Interpolation method: positioning (G00), linear (G01), circular (G02, G03), helical interpolation
	Position command range: Metric: -99,999.999mm ~ 99,999.999mm, minimum command unit: 0.0001mm Imperial: -9,999.9999inch ~ 9,999.9999inch, minimum command unit: 0.00001inch
	Electronic gear: Command multiplier coefficient 1~65,536, command division coefficient 1~65,536
	Fast movement speed: Up to 60m/min, fast speed override: F0, 25%, 50%, 100%, four-stage real-time adjustment
	Cutting feed speed: Up to 15m/min (G94) or 500.00mm/r (G95) Feed override: 0~200%, 21-stage real-time adjustment, band control available
	MPG feed: 0.001, 0.01, 0.1mm three-gear; single-step feed: 0.001, 0.01, 0.1, 1mm four-gear
Acceleration/deceleration	Forward acceleration/deceleration: Linear acceleration/deceleration or S-type acceleration/deceleration is optional, and acceleration/deceleration time constant can be set
	Backward acceleration/deceleration: Linear acceleration/deceleration or exponential-type acceleration/deceleration is optional, and acceleration/deceleration time constant can be set
	In manual, MPG, and single-step mode, backward acceleration/deceleration is applied. Forward/backward acceleration/deceleration can be selected for fast positioning, cutting feed
Auxiliary functions	Specifying with address M and 2 numbers, M function can be customized
	System internal M commands (cannot be redefined): End of program M02, M30; stop of program M00; optional stop M01; magazine call M06; subprogram call M98; subprogram end M99
	M commands already defined by the standard PLC: M03, M04, M05, M08, M09, M10, M11, M16, M17, M18, M19, M20, M21, M22, M23, M24, M26, M27, M28, M29, M35, M36, M44, M45, M50, M51
Tool functions	●Tool selected by T and 4 numbers ●Offset value for 256 sets of tools ●Length compensation ●Wear compensation ●C-type radius tool compensation
Spindle function	●S2 digits (I/O gear control)/S5 digits (analog output) ●Maximum spindle speed limit ●Constant linear speed function
	Spindle encoder: Number of encoder lines can be set (100~5,000p/r) Encoder to spindle transmission ratio: (1~255): (1~255)
	Spindle override: 50%~120%, eight-stage real-time adjustment, band control available
	Tapping cycle, soft tapping and rigid tapping, tapping backing
Automatic compensation	●Pitch error compensation: The compensation interval and compensation origin can be set, and the compensation range: -999 ~ +999 pulse equivalent
	●Backlash compensation: The machine tool backlash can be compensated at a fixed frequency or a higher/lower speed
	●Tool length compensation: A- or B-type length compensation function can be selected by parameters
	●Tool radius compensation: C-type tool compensation function, maximum compensation value is ±999.999mm or ±99.9999inch
Reliability and safety	Status signal: ●Emergency stop ●Over-travel ●Stored travel limit ●NC readiness signal ●Servo readiness signal ●MST function completed signal ●Automatic operation start light signal ●Automatic operation signal ●Feed hold light signal
	Self-diagnosis function: ●Signal ●System ●Position control ●Servo ●Communication ●Spindle

	NC alarm: ●Program ●Operation ●Over-travel ●Servo ●Connection ●PLC ●Memory (ROM and RAM)
Operation functions	●Edit ●Automatic ●MDI ●Zero return ●Manual ●Single step ●MPG ●DNC ●Single segment ●Skip ●Dry run ●Auxiliary lock ●Program restart ●MPG interrupt ●Single step interrupt ●Manual intervention ●Machine lock ●Interlock ●Feed hold ●Cycle start ●Emergency stop ●External reset signal ●External power ON/OFF
Display	●GSK 218MC and GSK 218MC-V systems adopt 10.4-inch color LCD with resolution of 800×600. ●GSK 218MC-H system adopts 8.4-inch color LCD with resolution of 800×600. ●Two interfaces in Chinese and English are available, which can be selected by parameters
	●Location information ●User program ●System setting ●PLC ●Diagnosis information ●System parameters ●Graphics ●Alarm information ●Help
	●Actual feed speed, spindle speed ●Real-time waveform diagnosis ●System run time and other NC commands and status information
Program editing	Program capacity: 216M, able to store up to 400 programs
	●Program preview Program editing ●Background editing;
PLC function	PLC processing speed: 3us/step; up to 8,000 steps; 10 basic commands, 35 function commands; online editing ladder diagram
	I/O unit input/output: 48/48, expandable
Communication functions	Supports communication interfaces of RS-232 Serial Port, USB, and Network, enables file transfer, DNC processing (Serial Port, Network) and USB online processing
Compatible drives	GSK DA98 series, GS series, GR series, GE series digital AC servo drive, etc.

1.3 Product Model Definition

GSK218 | M | (C) | (a) — (H) | (1) | (1)

1 2 3 4 5 6

S/N	Code description	
①	**Product model subject attribute part:** GSK218 series	
②	**Function (machining object) configuration:** Indicated in capital letters M-milling machine	
③	**Continuation of series:** Indicated in capital letters. None: Indicates the initial version	
④	**Continuation of sub-series (or improved version):** Indicated in small letters **a, b, c...**. None: Indicates the initial version	
⑤	**Structure type or special machine type**	**Structure type:** Indicated in capital letters U, H, V, and B, respectively. **U-united, H-horizontal, V-vertical, B-box.** **Special machine type:** Indicated in capital letter P.
⑥	**LCD size (structure) or special machine code**	**LCD size:** Indicated **in one-digit Arabic numeral 1~9**. **1 indicates 8.4-inch, 2 10.4-inch, 3~9 ...,** **Special machine code:** Indicated **in two-digit Arabic numeral 01~99**.

Example

- **GSK218MC-H:** Indicates 218MC series, horizontal structure, 8.4-inch LCD (default size)
- **GSK218MC-H2:** Indicates 218MC series, horizontal structure, 10.4-inch LCD
- **GSK218MC-P01:** Indicates 218MC series, No.01 special machine

1.4 Bus Function Description

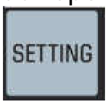
From the system software version **V1.4**, the system adds Ethernet bus communication mode.

The functions described in this manual are applicable to the bus transmission mode and the pulse transmission mode. Special descriptions are given for the new functions of the system in the bus transmission mode.

When Ethernet bus communication mode or pulse communication mode is desired, please follow the instructions below:

Method I:

1. Enter the <**Input**> operation mode;

2. Press the key 

to enter the <Settings> page, press the soft key [Password] to enter the Password page, and enter the password of the appropriate level. For details, please refer to "3.4.5 Password permission setting and modification" in the "Chapter II Operating Instructions" of this manual;

3. In the <Settings> interface, set the parameter switch to "1";

4. Press the key 

, and then press [⊞ Bus Configuration] to enter the page for the settings (Figure 1-4-1):

   1) Move the cursor to the item "Whether bus or not";

   2) Type "1" to select the bus, or type "0" to select the pulse;

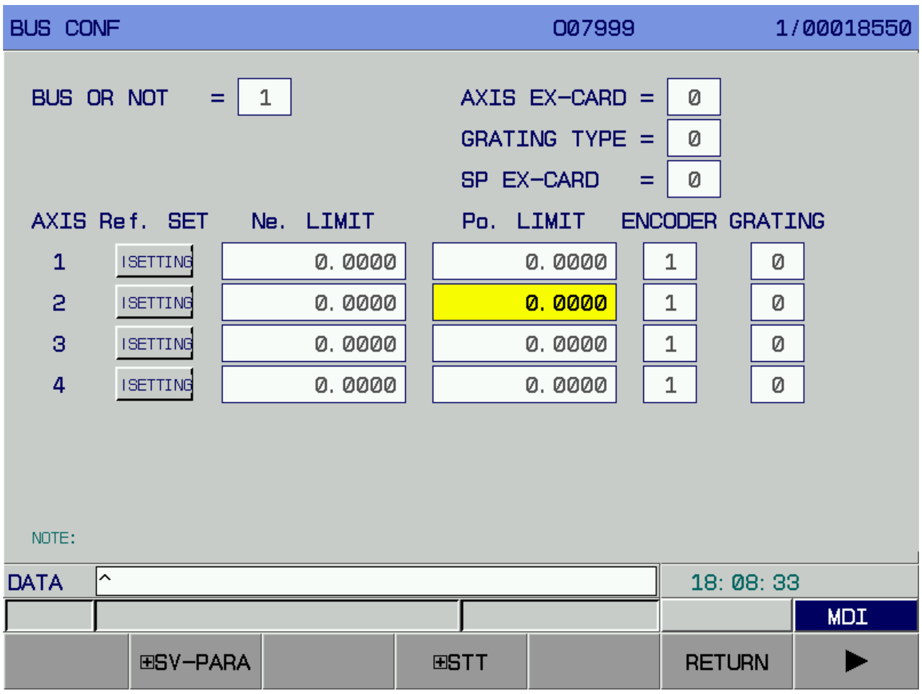


Figure 1-4-1

Method II:

Directly setting the position parameter **No: 0#0=1**, so the drive unit transmission mode is bus; or setting the position parameter **No: 0#0=0**, so the drive unit transmission mode is pulse.

Note: To modify this parameter, all power supplies must be cut off. Please re-power after modification.

Chapter II Programming Basics

2.1 Control Axis

Table 2-1-1

Item	GSK218MC
Number of basic control axes	3 (X, Y, Z)
Number of extended control axes (total)	Up to 5

As required by the structure design of some machine tools, it is sometimes necessary to have an additional axis, such as for a rotary table and a revolving table. This axis can be designed as a linear axis or as a rotary axis. GSK218MC can set each axis to a linear axis or a rotary axis through the position parameter **No.8#0~No.8#4**.

2.2 Axis Name

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

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

Note: If the axis name input is occupied, the system will automatically initialize it to X, Y, Z, A, B.

2.3 Axis Display

When the additional axis is set to a rotary axis, the unit of the rotary axis is displayed as deg. If it is set to a linear axis, it is displayed the same as the basic three axes (X, Y, Z) and the unit is mm. The following is the axis display when the 4th axis is set to a linear axis and the 5th axis to a rotary axis.

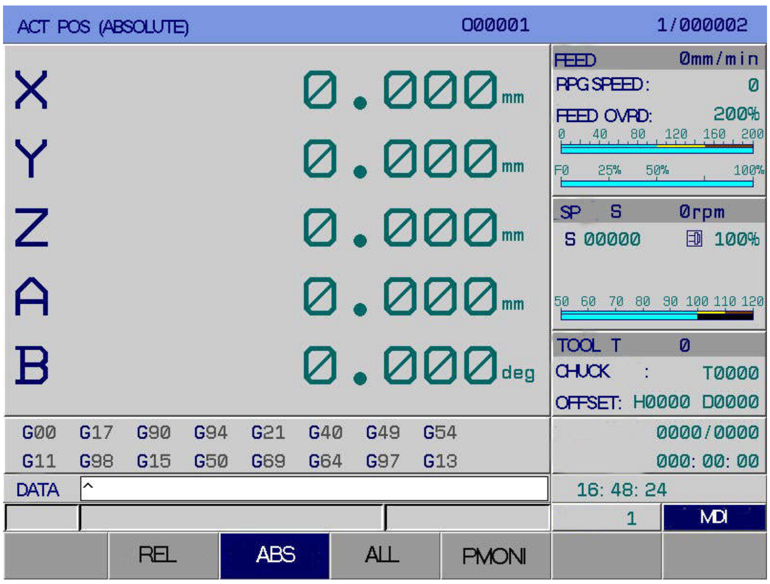


Fig. 2-3-1

2.4 Coordinate System

2.4.1 Machine Coordinate System

The specific point on the machine that is used as the machining reference is called the machine zero. The machine manufacturer sets the machine zero for each machine. The coordinate system set with the machine zero as the origin is called the machine coordinate system. After power is turned on, a manual return to the machine zero is performed to establish the machine coordinate system. Once the machine coordinate system is set, it remains unchanged until the power is turned off, the system is restarted, or the emergency stop is pressed.

The system adopts the right-hand Cartesian coordinate system. The vertical movement in the spindle direction is Z-axis movement. Viewing from the spindle to the workpiece, the spindle box is in Z-axis negative movement approaching workpiece and in Z-axis positive movement leaving workpiece. The remaining directions are determined by the right-hand Cartesian coordinate system.

2.4.2 Reference Point

On **CNC** machines, there is a special position where tool change usually takes places or the coordinate system is set, called the reference point. It is a fixed point in the machine coordinate system set by the machine manufacturer. With the reference point return function the tool can be easily moved to this position. Generally, the reference point of the CNC milling machine system coincides with the machine zero, and the machining center reference point is usually the tool change point.

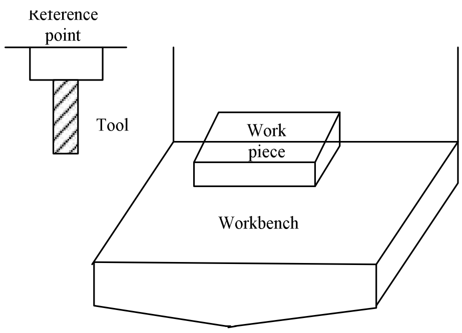


Fig. 2-4-2-1

To move the tool to the reference point, there are two ways:

1. Manually return to the reference point (see “Chapter IX Zeroing Operation”);
2. Automatically return to the reference point.

2.4.3 Workpiece Coordinate System

The coordinate system used when machining a workpiece is called workpiece coordinate system (also called part coordinate system). The workpiece coordinate system is preset by the **CNC** (setting the workpiece coordinate system).

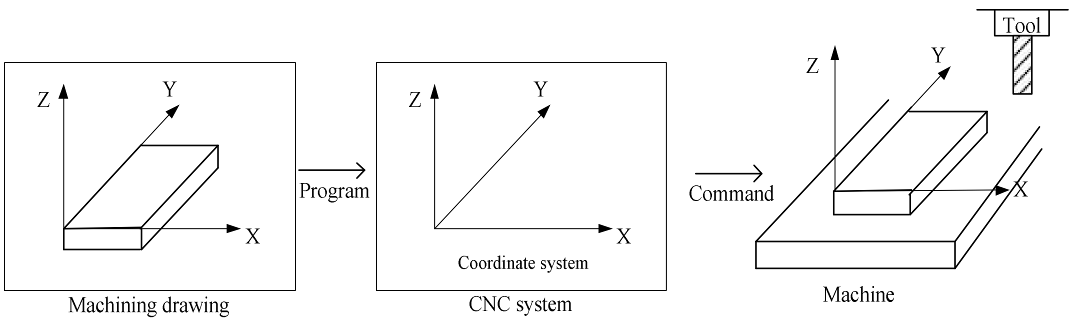


Fig. 2-4-3-1

The tool cuts the workpiece into the shape as indicated on the drawing on the **CNC** command workpiece coordinate system according to the command program of the programmed coordinate system on the machining drawing. The relative relationship between the machine coordinate system and the workpiece coordinate system must be determined. The method of determining the relative relationship between those two coordinate systems is called alignment. Different methods can be used depending on the shape of parts and the number of parts to be machined.

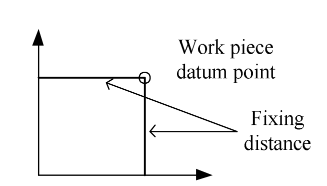
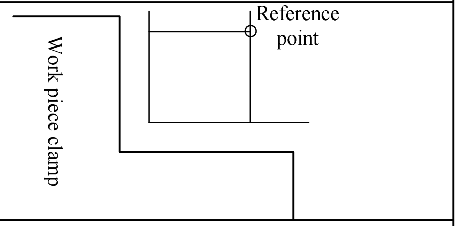
<table>
<tr><td>I) Use the part datum point</td><td>II) When the part is directly mounted on the clamp</td></tr>
<tr><td>
</td><td>
</td></tr>
<tr><td>Move the tool center and align it with the part datum point. In this position, use the CNC command to set the work piece coordinate system. At this moment, the work piece coordinate system is coincided with the programming coordinate system.</td><td>As the tool center cannot be directly positioned on the work piece datum point, it is necessary to position the tool to a position with known distance to the datum point (which can be the reference point), and use the known distance to set the work piece coordinate system of the CNC command. (Such as G92)</td></tr>
</table>

Fig. 2-4-3-2

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

There are two ways to set the workpiece coordinate system:

1. G92. For details, see 4.2.11 of this chapter.
2. G54 to G59. For details, see 4.2.8 of this chapter.

2.4.4 Absolute Coordinate Programming and Relative Coordinate Programming

There are two methods of defining the axis movement amount: absolute value and relative value. The absolute value definition is a method of programming with the coordinates of the end position of the axis movement, called absolute coordinate programming. The relative value definition is a method of directly programming with the axis movement amount, called relative coordinate programming (also called incremental coordinate programming).

1) Absolute coordinates

The coordinates of the target position in the specified workpiece coordinate system, that is, the coordinate position to which the tool is moved.

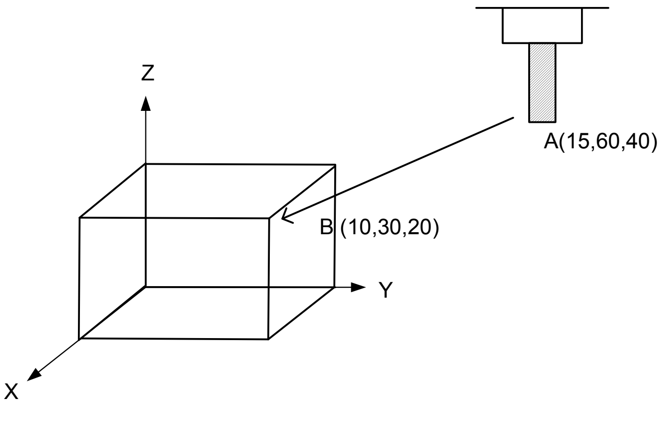


Fig.2-4-4-1

The tool moves from point A to point B. The coordinates of point B are used in the G54 workpiece coordinate system. The code is as follows:

G90 G54 X10 Y30 Z20;

2) Incremental coordinates

The coordinates of the target position relative to the current position using the current position as the coordinate origin.

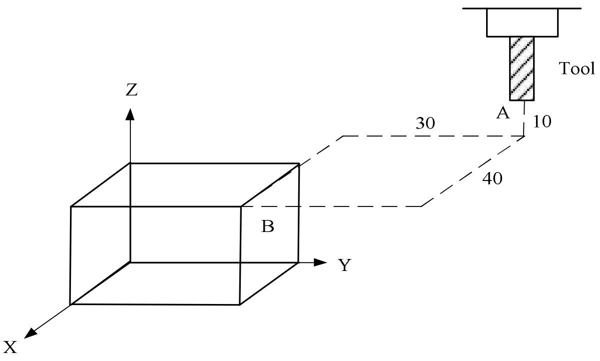


Fig. 2-4-4-2

The tool moves fast from point A to point B. The code is as follows:

G0 G91 X40 Y-30 Z-10;

2.5 Modal and Non-Modal

Modal means that the value of an address remains valid as soon as it is set until it is reset. Another meaning of modal is that after a function word is set, if the same function is used in a later program segment, it is not necessary to input the field.

- For example, the following programs:

G0 X100 Y100; (Fast positioning to X100 Y100)
X20 Y30; (Fast positioning to X20 Y30, G0 is modal specified and is not required to input)
G1 X50 Y50 F300; (Linear interpolation to X50 Y50, feed speed 300mm/min G0→G1)
X100; (Linear interpolation to X100 Y50, feed speed 300mm/min, G1, Y50, F300 are modal specified and are not required to input)
G0 X0 Y0; (Fast positioning to X0 Y0)

The initial state refers to the default modal after the system is powered on. See Table 4-1-2 for details.

- For example, the following programs:

O00001
X100 Y100; (Fast positioning to X100 Y100, G0 is the initial state of the system)
G1 X0 Y0 F100; (Linear interpolation to X0 Y0, feed per minute at a speed of 100mm/min)

Non-modal means that the value of the corresponding address is valid only in the program segment where this code is written, and must be reassigned if it is used in the later program segment. For example, Group 00 Code G in Table 4-1-2.

See Table 2-5-1 for the modal and non-modal descriptions of the function words.

Table 2-5-1 Modal and non-modal of function codes

Modal	Modal G function	A set of G functions that can end each other and, once executed, remain valid until they are ended by the G function in the same group.
	Modal M function	A set of M functions that can end each other and remain valid until they are ended by another function in the same group

Non-modal	Non-modal G function	Valid only in the specified program segment, and ended at the end of the program segment
	Non-modal M function	Valid only in the program segment where this code is written

Chapter III Composition of Parts Program

3.1 Program Component

A program is composed of multiple program segments which are further composed of words. Each program segment is separated by a program segment end code (LF for ISO, CR for EIA). In this manual, the character “;” is used to indicate the program segment end code.

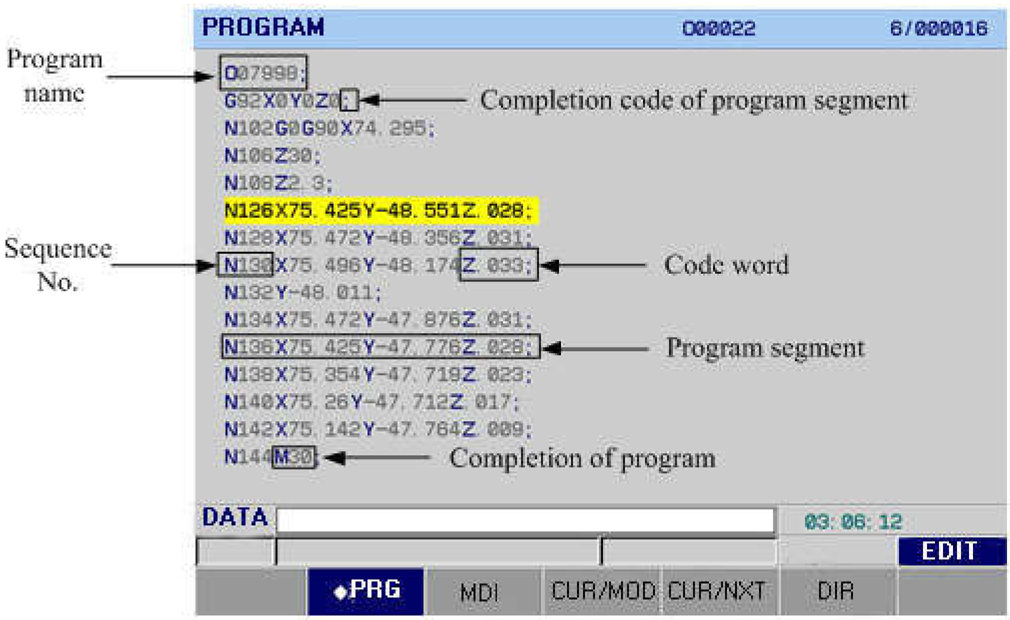


Figure 3-1-1 Structure of program

The collection of code series that control the CNC machine tool to complete part machining is called a program. After the written program is input to the CNC system, the system controls the tool to move along a straight line or an arc by code, or rotates and stops the spindle. In the program, the codes shall be written in the sequence the machine tool actually moves. The structure of program is shown in Figure 3-1-1.

3.1.1 Program Name

Multiple programs can be stored in the system’s memory. In order to distinguish those programs, the program name starts with address O followed by five numbers, as shown in Figure 3-1-1-1.

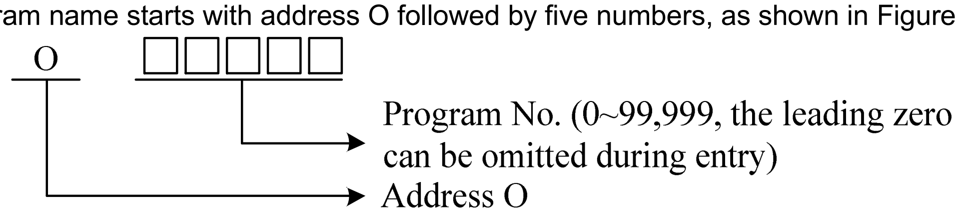


Figure 3-1-1-1 Composition of program name

3.1.2 Sequence Number and Program Segment

A program is composed of multiple codes, and a code unit is called a program segment (see Figure 3-1-1). The program segments are separated by a program segment end code (see Figure 3-1-1). In this manual, the character “;” is used to indicate the program segment end code.

A program segment can start with a sequence number consisting of address N and following four numbers (see Figure 3-1-1), with the leading zero code omitted. The order of the sequence number is arbitrary (set whether to insert the sequence number by the position parameter NO:0 # 5, or directly in the Settings interface. For details, see 3.4.1 of the “Chapter II Operating Instructions”), and the interval size may also vary (the interval size is set by data parameter P210). A sequence number may be provided to all program segments, or to the important ones. However, in the general machining order, the sequence number shall be from small to large. A sequence number is provided to the important program segments for convenience (e.g.: When changing a tool, or when the table indexing moves to a new machining face).

Note: When the **N** code is in the same segment as G10, it is not treated as a line number.

3.1.3 Code Word

The code word (Figure 3-1-3-1) is the element that makes up a program segment. It consists of the address and following numbers (which are sometimes preceded by symbols such as +, -).

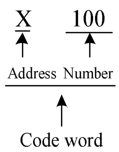


Figure 3-1-3-1 Composition of code word

The address is one of the English letters (A to Z). It defines the meaning of the following numbers. In this system, the address that can be used and its meaning and value range are shown in Table 3-1-3-1.

Depending on the preparation functions, sometimes an address has a different meaning.

If two or more identical addresses appear in the same code, whether to alarm is set by the position parameter N0:32#6.

Table 3-1-3-1

Address	Value range	Function meaning
A, B, C	Set by data parameters P175~179	Axis name address
D	0～255	The radius offset number, D0 defaults to 0 which the user cannot set or modify.
E		Not used
F	0.001～99999.999（mm/min）	Feed per min
	0.001～500(mm/r)	Feed per revolution
G	00～99	Preparation functions
H	01～99	Operators in G65
	0～255	The length offset number (H0 defaults to 0 which the user cannot set or modify.)
I	-99999999～99999999 (mm)	The arc center relative to origin is in the X-axis vector

Address	Value range	Function meaning
		(circular/helical interpolation, scaling)
	I shall be greater than the radius of the current tool	G22/G23 radius of groove inside circle
	(Tool radius + J) < I ≤ 99,999.999mm, the absolute value is taken if it is negative	G24/G25 and G26/G32 finish-milling circle radius
	I shall be greater than {(data parameter **P269** set value * tool radius) + tool radius}*2. The helical entry radius shall be less than {(I/2) - tool radius}	G33/G34 rectangular groove width in the X-axis direction
	0< I ≤99,999.999mm, the absolute value is taken if it is negative	G35/G36, G37/G38 rectangular groove width in the X-axis direction
J	-99999999～99999999 (mm)	The arc center relative to origin is in the Y-axis vector (circular/helical interpolation, scaling)
	0≤ J ≤99,999.999mm, the absolute value is taken if it is negative	G24/G25 and G26/G32 distance between finish-milling origin and finish-milling circle center
	J shall be greater than {(data parameter P269 set value * tool radius) + tool radius}*2. The helical entry radius shall be less than {(J/2) - tool radius}	G33/G34 rectangular groove width in the Y-axis direction
	0< J ≤99,999.999mm, the absolute value is taken if it is negative	G35/G36, G37/G38 rectangular groove width in the Y-axis direction
K	-99999999～99999999 (mm)	The arc center relative to origin is in the Z-axis vector (circular/helical interpolation, scaling)
	1～99999	Number of fixed cycle repeats
L	1～99998	Number of repeated subprogram calls
	Shall be smaller than the tool diameter and greater than 0	G22/G23 cutting width increment in XY plane of groove cycle inside circle
	Shall be smaller than the tool diameter and greater than 0, the absolute value is taken if it is negative	G33/G34 cutting width increment in the specified plane
	Tool radius ≤ L ≤ 99,999,999mm, the absolute value is taken if it is negative	G37/G38 distance between finish-milling origin and rectangular side X-axis direction
M	Set by data parameter P204	Auxiliary function output, program execution flow, subprogram call
N	0～99999	Sequence No.
	0～999	Parameter serial number (G10 online modification)
O	0～99999	Program name
P	0～99999.9999(ms)	Pause time
	1～99999	Called subprogram number
	-9999.9999～9999.9999	Scaling
	Data parameters P296~282	Pause time at hole bottom in fixed cycle or pause time at point R during backing
Q	-99999.999～99999.999 (mm)	Cutting depth or hole bottom offset in fixed cycle
R	-99999999～99999999 (mm)	Arc radius/angular displacement/corner value
	-99999.999～99999.999 (mm)	R plane in fixed cycle
S	Set by data parameter P205	Spindle speed specified

Address	Value range	Function meaning
	00～04	Multi-speed spindle output
T	Set by data parameter P206	Tool functions
U	Set by data parameters P175~179	Axis name address
	The value range of U is |U| greater than or equal to D/2, less than or equal to the smaller of I/2 and J/2	Corner arc radius in fixed cycle
V	Set by data parameters P175~179	Axis name address
	Greater than 0	Distance from the unmachined surface at the time of fast entry
W	Set by data parameters P175~179	Axis name address
	Shall be greater than 0 (if the first cutting depth exceeds the groove bottom, the machining will be done directly at the groove bottom)	The distance from R plane down to first cutting depth in the Z-axis direction in fixed cycle
X	Set by data parameters P175~179	Axis name address
	-99999.999～99999.999 (mm)	X-axis coordinate address
	0～9999.999 (S)	Specify pause time
Y	Set by data parameters P175~179	Axis name address
	-99999.999～99999.999 (mm)	Y-axis coordinate address
Z	Set by data parameters P175~179	Axis name address
	-99999.999～99999.999 (mm)	Z-axis coordinate address

Please note that all of the limits shown in Table 3-1-3-1 are for CNC devices and not for machine tools. Therefore, in programming, please refer also to the machine manufacturer's user manual in addition to this manual so as to write programs based on the understanding of programming limits.

Note: Each code word shall not be longer than 79 characters.

3.2 General Structure of Program

Programs are divided into main programs and subprograms. Usually, the CNC moves as instructed by the main programs. If codes of calling subprograms are present in main programs, the CNC moves as instructed by the subprograms. If codes of returning to main programs are present in subprograms, the CNC returns to the program segment following the segment where the main programs call the subprograms and continues execution. The sequence of program actions is shown in Figure 3-2-1.

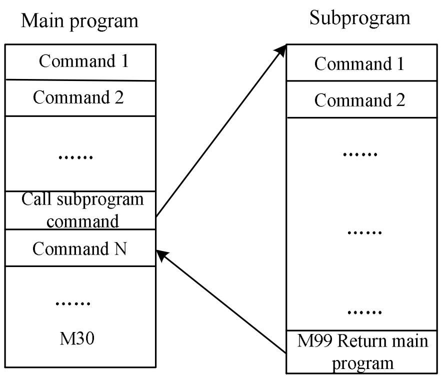


Fig. 3-2-1

Main programs and subprograms share the same composition.

When a fixed sequence exists and repeats in a program, it can be stored as a subprogram in memory without having to write it repeatedly to simplify the program. The subprogram can be called in the automatic mode, generally called with M98 in the main program, and the called subprogram can also call another subprogram. The subprogram called from the main program is called one-fold subprogram, and four levels of subprograms can be called (see Figure 3-2-2). At the last segment of the subprogram the main program is returned with the M99 code, and the program following the subprogram segment is called to continue execution. (If at the last segment the subprogram is ended with the M02 or M30 code with the same function as M99 to return to the main program, the program following the subprogram segment is called to continue execution.)

When the main program ends with M99, the program is executed repeatedly.

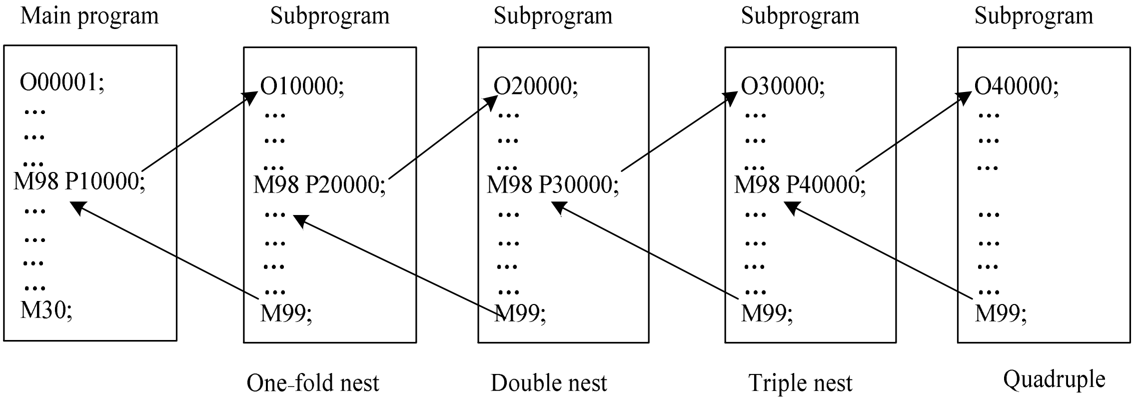


Figure 3-2-2 Four levels of subprogram nesting

A subprogram call code can be used to call the same subprogram continuously and repeatedly, up to 9,999 times.

3.2.1 Subprogram Writing

A subprogram is written in the following format:

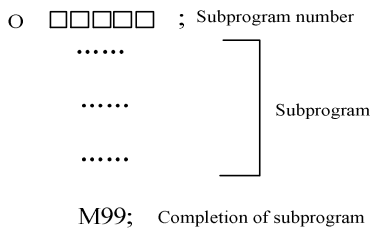


Fig. 3-2-1-1

A subprogram starts with address O, followed by subprogram number, and ends with M99 code (the M99 is written as shown above).

3.2.2 Subprogram Call

The subprogram is called by the main program or subprogram call code for execution. The format of a code for calling a subprogram is as follows:

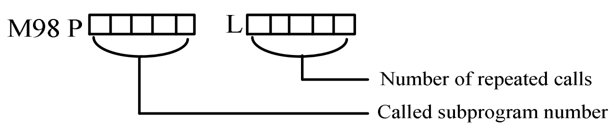


Fig. 3-2-2-1

● If the number of repeats is omitted, it is considered to be 1.

(Example) M98 P1002L5; (It indicates the subprogram with the number 1002 is called 5 times in succession.)

● The sequence in which the subprograms are called from the main programs for execution

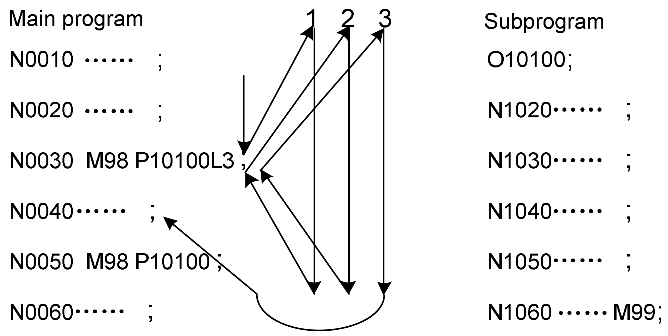


Fig. 3-2-2-2

Calling a subprogram from a subprogram is the same as calling a subprogram from a main program.

Note 1: An alarm is generated when the subprogram number specified by the address P cannot be retrieved.

Note 2: The subprograms No.90000~99999 are the system retained programs. When the user calls such subprograms, the system can execute the subprogram content but will not display it.

Note 3: Subprogram calls can be nested up to four levels.

3.2.3 End of Program

A program starts with the program name and ends with M02, M30 or M99 (see Figure 3-2-2-2). In the execution of a program, if the program end code M02, M30 or M99 is detected, the program ends if it ends with code M02 or M30 and the system is in reset state; for M30, the position parameter **N0:33#4** can be used to control whether to return to the program header, and for M02, the position parameter **N0:33#2** can be used to control whether to return to the program header. If it ends with code M99, it will return to the program header and the program loops. If M99, M02, and M30 are at the end of a subprogram, it will return to the program calling the subprogram and continue to execute the following program segment.

Chapter IV Preparation Function G Code

4.1 Type of Preparation Function G Code

The preparation function is indicated in G code followed by numbers, specifying the meaning of the program segment where it is located. There are two types of G codes:

Table 4-1-1

Type	Meaning
Non-modal G code	Valid only in the program segment where it is instructed
Modal G code	Always valid before other G codes in the same group

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

G01 X __ ;
Z ___ ; G01 is valid
X __ _; G01 is valid
G00 Z_ _; G00 is valid

When the system position parameter NO: 0#7 is set to 0, it is in normal machining mode, and when NO: 0#7 is set to 1, it is in high-speed high-precision machining mode.

Note 1: F: indicates it is the same as the normal machining mode; T: indicates it is in high speed and high precision machining mode

Note 2: Refer to the system parameter list for specific system parameters.

Table 4-1-2 G code and function

G code	Group	Code form	Is HS HP mode valid	Function
*G00	01	G00 X_Y_Z_	T	Positioning (fast movement)
G01		G01 X_Y_Z_F_	T	Linear interpolation (cutting feed)
G02		G02 / G03 X_Y_ R_ / I_J_ F_;	T	Circular interpolation CW (clockwise)
G03			T	Circular interpolation CCW (counterclockwise)
G04	00	G04 P_ or G04 X_	F	Pause, exact stop
G10		G10 L_N_P_R_	F	Programmable data input
*G11		G11	F	Cancel programmable data input mode
*G12	16	G12 X_Y_Z_ I_J_K_	F	Stored travel detection function connected
G13		G13	F	Stored travel detection function disconnected
*G15	11	G15	F	Cancel polar coordinate code

<table>
<tr><th>G code</th><th>Group</th><th colspan="3">Code form</th><th>Is HS HP mode valid</th><th>Function</th></tr>
<tr><td>G16</td><td></td><td colspan="3">G16</td><td>F</td><td>Polar coordinate code</td></tr>
<tr><td>*G17
G18
G19</td><td>02</td><td colspan="3">They are written in the program segments and used in circular interpolation and tool radius compensation.</td><td>F</td><td>XY plane selection
ZX plane selection
YZ plane selection</td></tr>
<tr><td>G20</td><td rowspan="2">06</td><td colspan="3" rowspan="2">The command must be specified in a separate program segment</td><td rowspan="2">F</td><td>Imperial data input</td></tr>
<tr><td>G21</td><td>Metric data input</td></tr>
<tr><td>G22</td><td rowspan="5">09</td><td colspan="3">G22 X_Y_Z_R_I_L_W_Q_V_D_F_ K</td><td>F</td><td>Groove rough milling inside circle (CCW)</td></tr>
<tr><td>G23</td><td colspan="3">G23 X_Y_Z_R_I_L_W_Q_V_D_F_K</td><td>F</td><td>Groove rough milling inside circle (CW)</td></tr>
<tr><td>G24</td><td colspan="3">G24 X_Y_Z_R_I_J_D_F_K_</td><td>F</td><td>Finish milling cycle inside full circle (CCW)</td></tr>
<tr><td>G25</td><td colspan="3">G25 X_Y_Z_R_I_J_D_F_K_</td><td>F</td><td>Finish milling cycle inside full circle (CW)</td></tr>
<tr><td>G26</td><td colspan="3">G26 X_Y_Z_R_I_J_D_F_K_</td><td>F</td><td>Finish milling cycle outside circle (CCW)</td></tr>
<tr><td>G27</td><td rowspan="5">00</td><td>G27</td><td colspan="2" rowspan="5">X_Y_Z_</td><td>T</td><td>Reference point return detection</td></tr>
<tr><td>G28</td><td>G28</td><td>T</td><td>Reference point return</td></tr>
<tr><td>G29</td><td>G29</td><td>T</td><td>Return from reference point</td></tr>
<tr><td>G30</td><td>G30Pn</td><td>T</td><td>Return to reference points 2, 3 and 4</td></tr>
<tr><td>G31</td><td>G31</td><td>F</td><td>Skip function</td></tr>
<tr><td>G32</td><td rowspan="7">09</td><td colspan="3">G32 X_Y_Z_R_I_J_D_F_K_</td><td>F</td><td>Finish milling cycle outside circle (CW)</td></tr>
<tr><td>G33</td><td colspan="3">G33X_Y_Z_R_I_J_L_W_Q_V_U_D_F_ K</td><td>F</td><td>Rectangular groove rough milling (CCW)</td></tr>
<tr><td>G34</td><td colspan="3">G34X_Y_Z_R_I_J_L_W_Q_V_U_D_F_ K</td><td>F</td><td>Rectangular groove rough milling (CW)</td></tr>
<tr><td>G35</td><td colspan="3">G35 X_Y_Z_R_I_J_L_ U_D_F_K_</td><td>F</td><td>Finish milling cycle inside rectangular groove (CCW)</td></tr>
<tr><td>G36</td><td colspan="3">G36 X_Y_Z_R_I_J_L_ U_D_F_K_</td><td>F</td><td>Finish milling cycle inside rectangular groove (CW)</td></tr>
<tr><td>G37</td><td colspan="3">G37 X_Y_Z_R_I_J_L_ U_D_F_K_</td><td>F</td><td>Finish milling cycle outside rectangle (CCW)</td></tr>
<tr><td>G38</td><td colspan="3">G38 X_Y_Z_R_I_J_L_ U_D_F_K_</td><td>F</td><td>Finish milling cycle outside rectangle (CW)</td></tr>
<tr><td>G39</td><td>00</td><td colspan="3">G39</td><td>F</td><td>Corner offset circular interpolation</td></tr>
<tr><td>*G40</td><td rowspan="3">07</td><td>G17</td><td rowspan="3">G40
G41
G42</td><td>D_X_Y_</td><td>T</td><td>Cancel tool radius compensation</td></tr>
<tr><td>G41</td><td>G18</td><td>D_X_Z_</td><td>T</td><td>Left tool radius compensation</td></tr>
<tr><td>G42</td><td>G19</td><td>D_Y_Z_</td><td>T</td><td>Right tool radius compensation</td></tr>
</table>

G code	Group	Code form		Is HS HP mode valid	Function
G43		G43		T	Positive direction tool length compensation
G44	08	G44	H_Z_	T	Negative direction tool length compensation
*G49		G49		T	Cancel tool length compensation
*G50	12	G50		T	Cancel scaling
G51		G51 X_ Y_ Z_ P_		T	Scaling
G53	00	It is written in programs		T	Selection of machine coordinate system
G54					Workpiece coordinate system 1
G55					Workpiece coordinate system 2
G56	05	They are written in the program segments, usually at the beginning of the programs.		T	Workpiece coordinate system 3
G57					Workpiece coordinate system 4
G58					Workpiece coordinate system 5
G59					Workpiece coordinate system 6
G60	00/01	G60 X_ Y_ Z_		T	Single direction positioning
G61		G61		T	Exact stop mode
G62	14	G62		T	Automatic corner override
G63		G63		T	Tapping method
*G64		G64		T	Cutting method
G65	00	G65 H_P# i Q# j R# k		T	Macro program code
G68	13	G68 X_ Y_ R_		T	Coordinate rotation
*G69		G69		T	Cancel coordinate rotation
G73		G73 X_Y_Z_R_Q_F_;		F	High-speed deep hole machining cycle
G74		G74 X_Y_Z_R_P_F_;		F	Left tapping cycle
G76		G76 X_Y_Z_Q_R_P_F_K_;		F	Precision boring cycle
*G80		It is written with other programs in the program segments		F	Cancel fixed cycle
G81		G81 X_Y_Z_R_F_;		F	Drilling cycle (point drilling cycle)
G82	09	G82 X_Y_Z_R_P_F_;		F	Drilling cycle and boring cycle
G83		G83 X_Y_Z_R_Q_F_;		F	Chip removal drilling cycle
G84		G84 X_Y_Z_R_P_F_;		F	Right tapping cycle
G85		G85 X_Y_Z_R_F_;		F	Boring hole cycle
G86		G86 X_Y_Z_R_F_;		F	Boring hole cycle
G87		G87 X_Y_Z_R_Q_P_F_;		F	Back boring cycle
G88		G88 X_Y_Z_R_P_F_;		F	Boring hole cycle
G89		G89 X_Y_Z_R_P_F_;		F	Boring hole cycle

G code	Group	Code form	Is HS HP mode valid	Function
*G90	03	They are written in program segments	T	Absolute value programming
G91				Incremental value programming
G92	00	G92 X_Y_Z_	T	Floating coordinate system setting
*G94	04	G94	T	Feed per minute
G95		G95	T	Feed per revolution
G96	15	G96S_	T	Constant peripheral speed control (cutting speed)
*G97		G97S_	T	Cancel constant peripheral speed control (cutting speed)
*G98	10	They are written in program segments	T	Return to the initial plane in a fixed cycle
G99				Return to the R point plane in a fixed cycle

Note 1: If the modal codes are in the same segment as the non-modal codes, the non-modal codes take precedence and change to the corresponding modes according to other modal codes in the same segment, but they are not executed.

Note 2: For G codes with * mark, when the power is turned on, the system is in the state of these G codes. (Some G codes are determined by the position parameter NO: 31#0~7)

Note 3: The G codes in group 00 are non-modal G codes except for G10, G11, and G92.

Note 4: If G codes not listed in the G code table are used, an alarm occurs, or instructing G codes without selection function also leads to an alarm.

Note 5: G codes from several different groups can be instructed in the same program segment. In principle, two or more G codes from the same group cannot be instructed in the same program segment. If the alarm is set to not occur for codes from the same group being in the same segment, the G code that appears later will apply.

Note 6: When the codes from groups 01 and 09 are in the same segment, the codes from group 01 will apply. In the fixed cycle mode, if the G codes from the group 01 are instructed, the fixed cycle is automatically canceled and the system is in the G80 state.

Note 7: The G codes are represented by group number depending on their types. It is set by position parameters NO: 35#0~7 and NO: 36#0~7 whether to clear the G codes of each group during reset or emergency stop.

Note 8: When the rotating and scaling codes are in the same segment as codes from group 01 or 09, the rotating and scaling codes will apply, and the mode of the codes from group 01 or 09 will be changed. The system will give an alarm when the rotating and scaling codes are in the same segment as the codes from group 00.

4.2 Simple G Code

4.2.1 Fast Positioning G00

Format: G00 X_Y_Z_

Function: With the G00 code, the tool moves fast to the position in the workpiece coordinate system specified by the absolute value code or incremental value code. The position parameter **N0:12#1** is used for setting, and one of the following two tool paths is selected (see Figure 4-2-1-1).

1. Linear interpolation positioning: The tool path is the same as the linear interpolation (G01), and the tool moves fast to the specified position in the shortest time without exceeding the speed of each axis.

2. Non-linear interpolation positioning: The tool moves fast at the speed of each axis, and the tool path is generally non-linear.

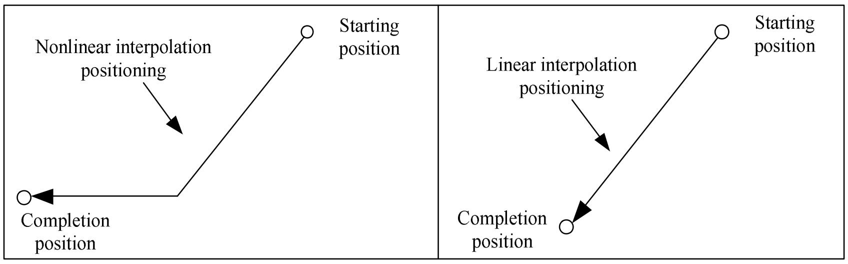


Fig. 4-2-1-1

Description:

1. After executing G00, the system changes the mode of the current tool movement to G00 mode. By changing the value of the system position parameter **N0:31#0**, it can be set whether the default mode is G00 (when the parameter value is 0) or G01 (when the parameter value is 1) when the power is turned on.
2. The tool does not move without specifying the position parameter. The system just changes the mode of the current tool movement to G00.
3. G00 and G0 are equivalent format.
4. The G0 speed of X, Y, Z, 4TH and 5TH axes is set by data parameters P88~P92.

Restrictions:

The fast movement speed is set by parameters. For example, F speed set in the G0 code is the cutting feed speed of the following machining segment.

For example:

G0 X0 Y10 F800;　　Fast feed at a speed set by system parameters

G1 X20 Y50;　　　Feed at the speed of F800

The fast positioning speed is adjusted by the keys on the operation panel (as shown in Figure 4-2-1-2) to F0, 25%, 50%, 100%; the speed corresponding to F0 is set by data parameter P93 and applies to each axis.

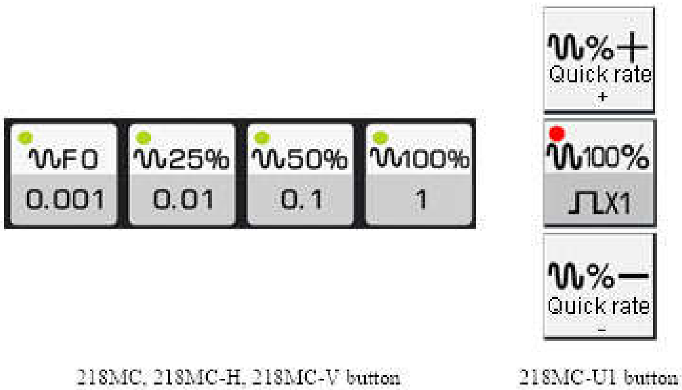


Figure 4-2-1-2 Fast feed override key

Note: During programming, pay attention to the position of table and workpiece to prevent tool collision.

4.2.2 Linear Interpolation G01

Format: G01 X_ Y_ Z_ F_

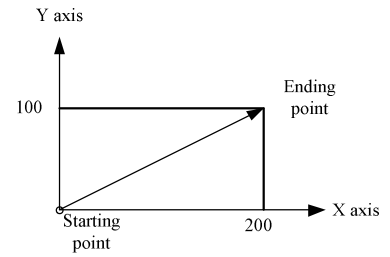
Function: The tool moves in the straight line to the specified position at the feed speed (mm/min) specified by parameter F.

Description:

1. X_ Y_ Z_ is the ending coordinate. For the concept of coordinate system, please refer to sections 2.4.1 to 2.4.4.
2. The F specified feed speed remains valid until a new F value is specified. The feed speed

instructed by the F code is calculated by interpolation along the linear path. If the F code does not instruct in the program, the feed goes at the feed speed of the default F value when the system is powered on. (Settings are shown in data parameter **P87**).

Program instance (Figure 4-2-2-1).



G01 X200 Y100 F200 ;

Note: The speed in each axis direction is as follows:

G01 Xα Yβ ZγFf ;

In this program segment:

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

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

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

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

Fig. 4-2-2-1

Note:

1. **The code parameters except for F are all position parameters. The upper limit of the cutting feed speed F is set by the data parameter P96. If the actual cutting speed (feed speed after the override is applied) exceeds the upper limit, it is limited to the upper limit. The unit is mm/min. The lower limit of the cutting feed speed F is set by the data parameter P97. If the actual cutting speed (feed speed after the override is applied) is lower than the limit, it is limited to the lower limit. The unit is mm/min.**
2. **The tool does not move when the position parameter is not specified after G01. The system just changes the mode of the current tool movement to G01. By changing the value of the system position parameter N0:31#0, it can be set whether the default mode is G00 (when the parameter value is 0) or G01 (when the parameter value is 1) when the power is turned on.**

4.2.3 Circular (Helical) Interpolation G02/G03

A. Circular interpolation G02/G03

G02 and G03 specification:

The circular interpolation in a plane completes the circular path in the specified plane running from the start point to the end point in the specified direction of rotation and along the radius (or circle center). As knowing the start and end points cannot completely determine the circular path, it is necessary to give:

- ➢ The direction of rotation of the circular arc (G02, G03)
- ➢ The plane of the circular interpolation (G17, G18, G19)
- ➢ The center coordinates or radius, from which two code formats are derived, together with center coordinates I, J, K or radius R programming.

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

The circular interpolation can be performed with the following codes, and the tool can move along the circular arc as follows:

Arc on the XY plane

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

Arc on the ZX plane

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

Arc on 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	Meaning
1	Specify plane	G17	Specify arc on XY plane
		G18	Specify arc on ZX plane
		G19	Specify arc on YZ plane
2	Turning direction	G02	Turn CW
		G03	Turn CCW
3	End position G90 mode	Two axes of X, Y, Z	End position coordinates in the workpiece coordinate system
	End position G91 mode	Two axes of X, Y, Z	Coordinates of end point relative to start point
4	Vector from start point to circle center	Two axes of I, J, K	Position coordinates of circle center relative to start point
	Arc radius	R	Arc radius
5	Feed speed	F	Tangential speed of arc

The CW and CCW are viewing the XY plane (ZX plane or YZ plane) from the positive direction of the Z-axis (Y-axis or X-axis) to its negative direction in the right-hand Cartesian coordinate system, as shown in Figure 4-2-3-1.

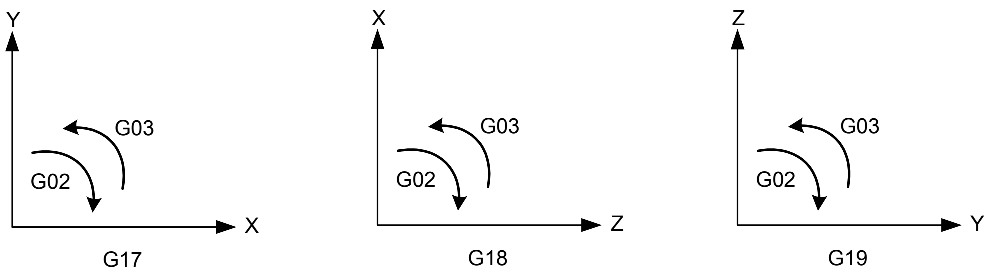


Fig. 4-2-3-1

By setting the position parameters N0:31#1 and #2, the default plane modal information at power-up can be specified.
The end point of the arc is specified by the parameter word X, Y or Z. The end point is expressed in absolute value when corresponding to the G90 command, and in incremental value when corresponding to G91. The incremental value is the coordinate of the end point relative to the start point. The center of the arc is specified by the parameter words I, J, and K,

which correspond to X, Y, and Z, respectively. The I, J, and K parameter values are the coordinates of the circle center relative to the start point of the arc, whether they are in the absolute mode G90 or the relative mode G91 (simply understood as the coordinates of the circle center with the start point being the coordinate origin temporarily), and are incremental values containing symbols, as shown in Figure 4-2-3-2.

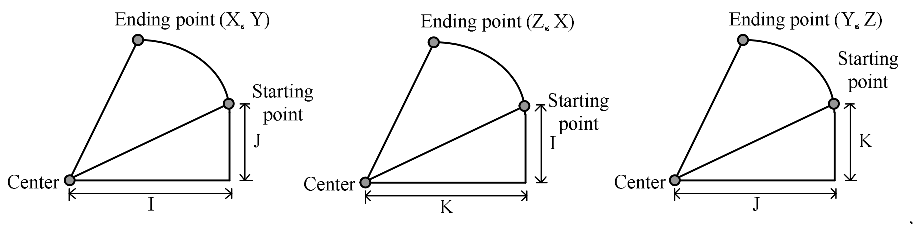


Fig. 4-2-3-2

I, J, K contain symbols according to the direction of the circle center relative to the start point. In addition to I, J, and K, the center of the arc can also be specified by radius R. This is shown as follows:

G02 X_ Y_ R_ ;
G03 X_ Y_ R_ ;

1. Now the following two circular arcs can be drawn, one larger than 180° and the other smaller than 180°. For an arc greater than 180°, the radius is specified by a negative value.

(For example, Figure 4-2-3-3) When the arc of ① is smaller than 180°

G91 G02 X60 Y20 R50 F300 ;

When the arc of ② is greater than 180°

G91 G02 X60 Y20 R-50 F300 ;

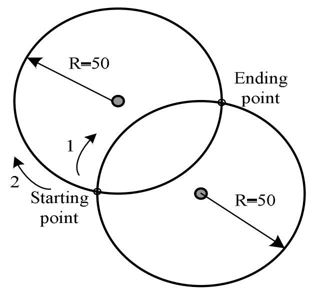


Fig. 4-2-3-3

2. For an arc equal to 180°, I, J, and K as well as R can be used for programming:

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

Equivalent to G90 G0 X0 Y0; G2 X20 R10 F100

Or G90 G0 X0 Y0; G2 X20 R-10 F100

Note: For an arc of 180°, positive and negative values of R do not affect the running path of the arc.

3. For an arc equal to 360°, only I, J, and K can be used for programming.

(Program example):

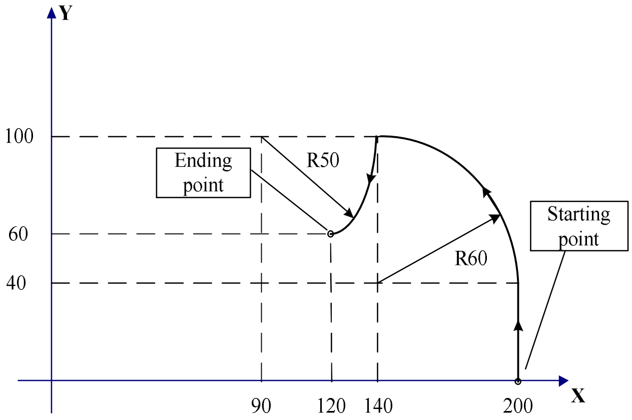


Fig. 4-2-3-4

The tool path in Figure 4-2-3-4 is programmed as follows:

1) Absolute value programming

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

Or

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

2) Incremental value programming

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

Or

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

Restrictions:

1. If the program specifies the addresses I, J, K, and R at the same time, the arc specified by the address R takes precedence, and the others are ignored.
2. If the arc radius parameter and the parameters from the start point to the arc center are not specified, the system will give an alarm.
3. For full circle interpolation, it is allowed to only specify the parameters I, J, and K from the start point to the arc center, rather than R.
4. Note the setting for selection of coordinate plane during circular interpolation.
5. If X, Y, and Z are all omitted, that is, the start and end positions are the same, and when R is specified (e.g.: G02R50;), the tool does not move.

B. Helical interpolation

Code format: G02/G03

Arc on the XY plane

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

Arc on the ZX plane

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

Arc on the YZ plane

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

Fig. 4-2-3-5

Function: Moving the tool to the specified position in a helical path from the current point at the feed speed specified by parameter F.

Description:

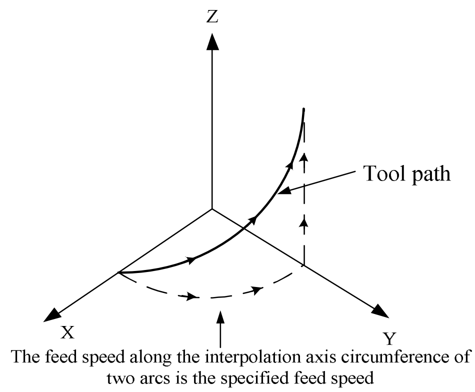


Fig. 4-2-3-6

The first two numbers of the code parameters are the position parameters. The parameter word is the number of the two axes (X, Y or Z) in the current plane. Those two position parameters specify where the tool shall move in the current plane. The third parameter word of the code parameters is the linear axis other than the circular interpolation axis. Its parameter value is the helix height. The specific meanings and restrictions of other code parameters are the same as circular interpolation.

If the system cannot machine a circle based on a given code parameter, the system returns an error message. After execution, the system changes the mode of the current tool movement to G02/G03 mode.

The feed speed along the circumference of the two circular interpolation axes is specified. The instruction method simply adds a movement axis other than a circular interpolation axis. The F code specifies the feed speed along the arc. Therefore, the feed speed of the linear axis is as follows:

$$F_C = F \times \frac{\text{Linear axis length}}{\text{Arc length}}$$

The feed speed is determined in the way that the feed speed of the linear axis does not exceed any limit.

Restrictions:

Note the setting for selection of coordinate plane during helical interpolation.

4.2.4 Absolute Value/Incremental Value Programming G90/G91

Format: G90/G91

Function: There are two methods of coding axis movement amount: absolute value coding and incremental value coding.

The absolute value coding is a method of programming with the coordinates of the end position of the axis movement. As the end position involves concept of coordinate system, please refer to 2.4.1 to 2.4.4.

The incremental value coding is a method of directly programming with the axis relative movement amount. The increment value has nothing to do with the coordinate system. Just give the movement direction and distance of the end position relative to the start position.

The absolute value code and the incremental value code are G90 and G91, respectively.

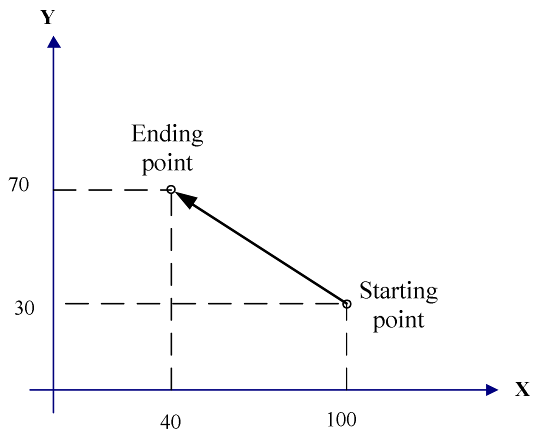


Fig. 4-2-4-1

The movement from the start point to the end point in Figure 4-2-4-1 is programmed with the absolute value code G90 and the incremental value code G91 as follows:

G90 G0 X40 Y70;

Or G91 G0 X-60 Y40;

The same action can be done in both ways, and the operator can use them as needed.

Description:

- No code parameters. It can be written with other codes in the program segments.
- G90 and G91 are the modal values in the same group. When G90 is specified, G90 mode (default mode) applies before G91 is specified. For G91, it remains valid before G90 mode is specified.

System parameters:

Setting position parameter **N0:31#4** can specify whether the default position parameter at power-up is G90 mode (when parameter is 0) or G91 mode (when parameter is 1).

4.2.5 Pause (G04)

Format: G04 X_ or P_

X, P: specified time.

Function: G04 performs a pause action and executes the next program segment at the specified time delay. In addition, for the cutting method, in the G64 mode a pause can be specified for an accurate stop check. Setting the position parameter No. 34#0 can specify a pause for the feed mode G95 per revolution.

Volume I Programming instructions

Table 4-2-5-1 Range of command values for pause time (with X command)

Least command increment	Command value range	Pause 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 Range of command values for pause time (with P command)

Least command increment	Command value range	Pause time unit
No.5#1=0	1~99999.999	0.001s or rev
No.5#1=1	1~99999.999	0.0001s or rev

Description:

1. G04 is a non-modal code and is valid only on the current line.
2. When the X and P parameters appear simultaneously, the X value is valid.
3. When the X and P values are set to negative values, an alarm will be given.
4. When neither X nor P is specified, the system does not perform a pause.
5. If an axis command other than X (Y, Z, U, V, W, A, B, C) is instructed after the G04 command, an alarm will be given.

4.2.6 Single Direction Positioning (G60)

Format: G60 X_ Y_ Z_

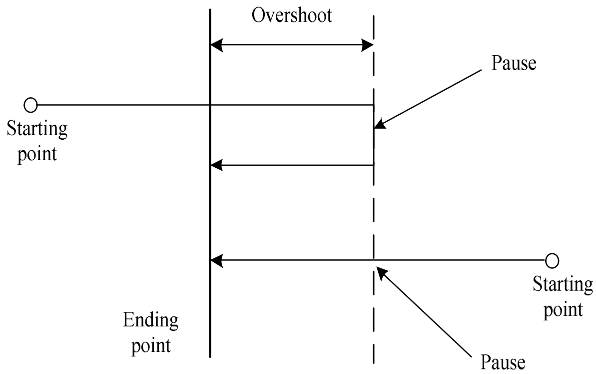


Fig. 4-2-6-1

Function: To eliminate the backlash of the machine for accurate positioning, the G60 can be used for accurate positioning in one direction.

Description:

G60 is a non-modal G code (which can be set to a modal value by position parameter **NO:48#0**) and is valid only in the specified program segment.

The parameters X, Y and Z indicate the coordinates of the end point in absolute value programming and the distance the tool moves in incremental value programming. Under tool offset, when single direction positioning is used, its path is the path after tool compensation. In the above figure, the marked overshoot can be set by the system parameters **P335**, **P336**, **P337**, and **P338**, the pause time can be set by **P334**, and positioning direction can be determined by the positive or negative of the set overshoot. Refer to system parameters for details.

Example 1:

G90 G00 X-10 Y10;

G60 X20 Y25; (1)
If the system parameters **P334** = 1, **P335** = -8, **P336** = 5; then, for the (1) statement, the tool path is AB → pause for 1s → BC

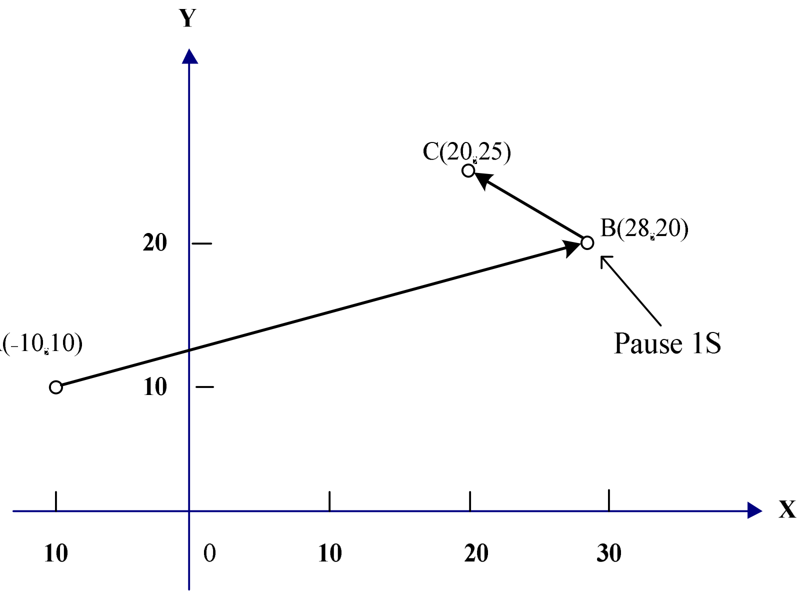


Fig. 4-2-6-2

System parameters:

Table 4-2-6-1

P334	Pause time in single direction positioning (unit: s)
P335	1st axis single direction positioning direction and over-travel (unit: mm)
P336	2nd axis single direction positioning direction and over-travel (unit: mm)
P337	3rd axis single direction positioning direction and over-travel (unit: mm)
P338	4th axis single direction positioning direction and over-travel (unit: mm)
P339	5th axis single direction positioning direction and over-travel (unit: mm)

Note 1: The symbols of data parameters P335 to P339 indicate the direction of single direction positioning, and the value of the parameters is over-travel amount.
Note 2: The over-travel amount is >0, and the positioning direction is positive.
Note 3: The over-travel amount is <0, and the positioning direction is negative.
Note 4: The over-travel amount = 0, and no single direction positioning is performed.

4.2.7 Online Change of System Parameters (G10)

Function: This function is used to set or modify the tool radius, length offset, external zero offset, workpiece zero offset, additional workpiece zero offset, data parameters, position parameter, etc. in the programs.

Format:

G10 L50 N_P _R_;	Set or modify a position parameter
G10 L51 N_ R_;	Set or modify a data parameter
G11;	Cancel parameter input mode

Parameter definition:

N: Parameter No. Parameter serial number to be modified.
P: Parameter bit number. Parameter bit number to be modified.
R: Modified value. Used to specify the modified value of a parameter.
The specified values can also be modified with the following code. Refer to the relevant sections for details:

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

Note 1: In the parameter input mode, other NC statements cannot be specified except for annotative statements.
Note 2: The G10 program segment must be instructed separately, or an alarm will occur. After using G10, remember to use G11 to cancel the parameter input mode, so as not to affect the normal use of the program.
Note 3: The parameter value modified by G10 must be within the range of the system parameters, or an alarm will occur.
Note 4: The modal code of the fixed cycle must be canceled before running G10, or the system will alarm.
Note 5: Parameters that are valid only after power-off and restart shall not be modified by G10.
Note 6: G20 and G21 cannot be modified online by G10.
Note 7: The G10 command is used to online modify the external zero offset, the workpiece zero offset, the additional workpiece zero offset or the tool offset. When modifying in the G91 mode, the system adds the command offset with the current offset; when modifying in the G90 mode, it is made by the command offset.
Note 8: The G10 mode is canceled when M00, M01, M02, M30, M99, M98, and M06 are executed.
Note 9: Position parameter No.0#7 (select mode 0: normal mode, 1: high-speed high-precision mode) does not support G10 online modification.

4.2.8 Workpiece Coordinate Systems G54~G59

Function: Specifying the current workpiece coordinate system and selecting the workpiece coordinate system by specifying the workpiece coordinate system G code in the program.
Format: G54~G59
Description:

1. No code parameters.
2. The system itself can set six workpiece coordinate systems, and any one of the coordinates can be selected through 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. At power-up, the system displays the workpiece coordinate systems G54~G59, G92 or the additional workpiece coordinate systems that were executed before power-off.
4. When different workpiece coordinate systems are called in the program segments, an axis moved as instructed will be positioned to the coordinate point in the new workpiece coordinate system; ...for an axis not moved as instructed, its coordinates will skip to the corresponding coordinates in the new workpiece coordinate system, without any change to the actual machine position.

Example: The machine coordinate corresponding to the G54 coordinate system origin is (10, 10, 10)
The machine coordinate corresponding to the G55 coordinate system origin is (30, 30, 30)
When the programs are executed sequentially, the absolute coordinates of the end point and the machine coordinates are displayed as follows:

Table 4-2-8-1

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

5. G10 can be used to change the external workpiece zero offset value or the workpiece zero offset value. Methods as below:
Using code G10 L2 Pp X_Y_Z_
P=0: External workpiece zero offset value (base offset).
P=1 to 6: Workpiece zero offset of workpiece coordinate systems 1 to 6.
X_Y_Z_: For the absolute value code (G90), the workpiece zero offset value of each axis. For incremental value code (G91), the offset of each axis added to the set workpiece zero (the added result as the new workpiece zero offset).
Using the G10 command, each workpiece coordinate system can be changed separately.

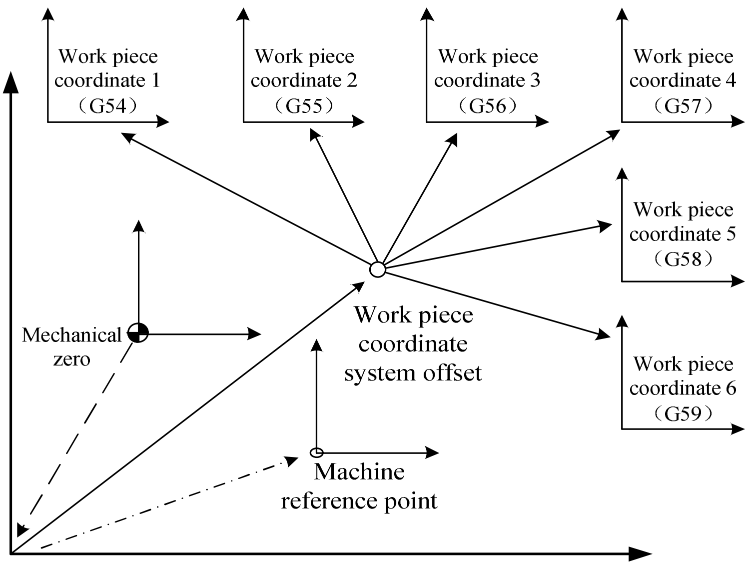


Fig. 4-2-8-1

As shown in Figure 4-2-8-1 above, after the machine is started, it is manually returned to the mechanical zero from which the machine coordinate system is established, thereby generating the machine reference point and determining the workpiece coordinate system. The workpiece coordinate system offset data parameters **P10~P13** correspond to the overall offsets of the six workpiece coordinate systems. The origins of the six workpiece coordinate systems can be specified by inputting the coordinate offset in the input mode or by setting the data parameters **P15~P43**. The six workpiece coordinate systems are set according to the distance from the mechanical zero to the respective coordinate system zero.

Volume I Programming instructions

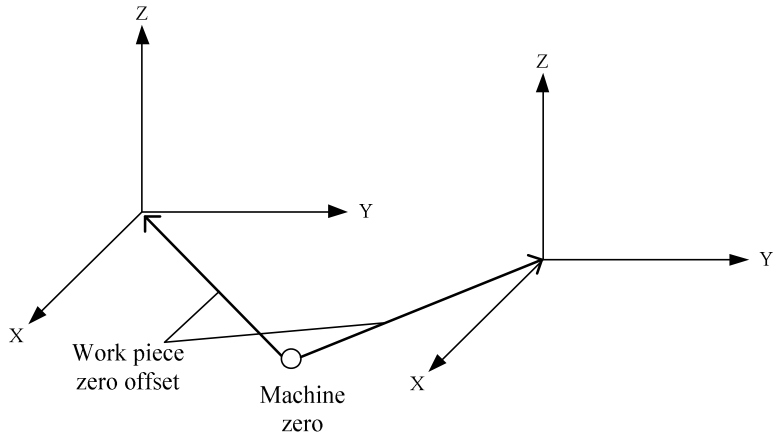


Fig. 4-2-8-2

Example:

N10 G55 G90 G00 X100 Y20;

N20 G56 X80.5 Z25.5;

In the above example, when the execution of N10 program segment begins, fast movement is made to the position of the workpiece coordinate system G55 (X=100, Y=20). When the execution of N20 program segment begins, fast movement is made to the position of the workpiece coordinate system G56. The absolute coordinate value automatically becomes the coordinate value in the G56 workpiece coordinate system (X=80.5, Z=25.5).

4.2.9 Additional Workpiece Coordinate Systems

In addition to the six workpiece coordinate systems (G54 to G59 coordinate systems), 50 additional workpiece coordinate systems can be used.

Format: G54 Pn

Pn: Code specifying additional workpiece coordinate systems. The range of Pn is 1~50.

The setting and restriction of the additional workpiece coordinate systems are consistent with the workpiece coordinate systems G54~G59.

The workpiece zero offset values can be set with G10 in the additional workpiece coordinate systems. Methods as below:

Code: G10 L20 Pn X_Y_Z_;

n=1 to 50: Codes of additional workpiece coordinate systems.

X_Y_Z_ : Axis address and offset value setting workpiece zero offset.

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

For the incremental value code (G91), the specified value is added to the current offset value to obtain the new offset.

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

4.2.10 Selection of Machine Coordinate System G53

Format: G53 X_ Y_ Z_

Function: Fast positioning the tool to the corresponding coordinate in the machine coordinate system.

Description:

1. When G53 is used in the program, the following code coordinates will be the coordinate values in the machine coordinates, and the machine will fast move to the specified position.
2. G53 is a non-modal code and is valid only in the current segment. It does not affect the previously defined coordinate system.

Restrictions:

Selection of machine coordinate system G53

When a position on the machine coordinate system is instructed, the tool moves fast to that position. The G53 used to select the machine coordinate system is a non-modal G code; that is, it is valid only in the program segment where the machine coordinate system is instructed. The absolute value G90 shall be specified for G53, and the G53 command is ignored when the incremental value (G91) is specified. When the tool is instructed to move to a special position on the machine, for example, the tool change position can be set at that point by using the movement program written with G53.

Note: When G53 is specified, tool radius compensation and tool length offset are temporarily canceled and will be restored in the next compensated axis program segment that is cached.

4.2.11 Floating Coordinate System G92

Format: G92 X_ Y_ Z_

Function: Setting the floating workpiece coordinate system. The three code parameters specify the absolute coordinate value of the current tool in the new floating workpiece coordinate system. This code does not result in movement of the motion axis.

Description:

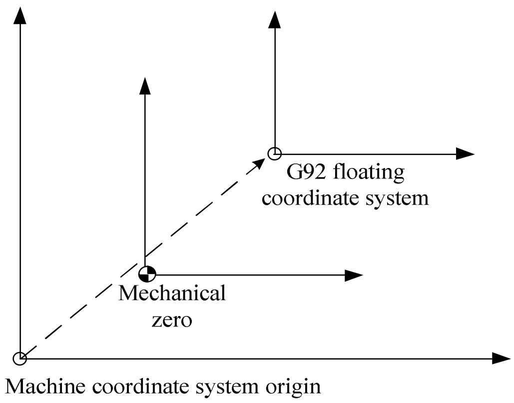


Fig. 4-2-11-1

1. As shown in Figure 4-2-11-1, the origin of the G92 floating coordinate system is the value in the machine coordinate system, and has no relation with the workpiece coordinate system. For the validity of G92 after setting, it is valid:
   1) Before calling the workpiece coordinate system
   2) Before the machine zeroing

   The G92 floating coordinate system is usually used for alignment during temporary workpiece machining. It usually runs at the beginning of the program or as instructed in MDI mode before automatically running the program.
2. There are two ways to determine the floating coordinate system:
   1) Determine the coordinate system by tool tip:

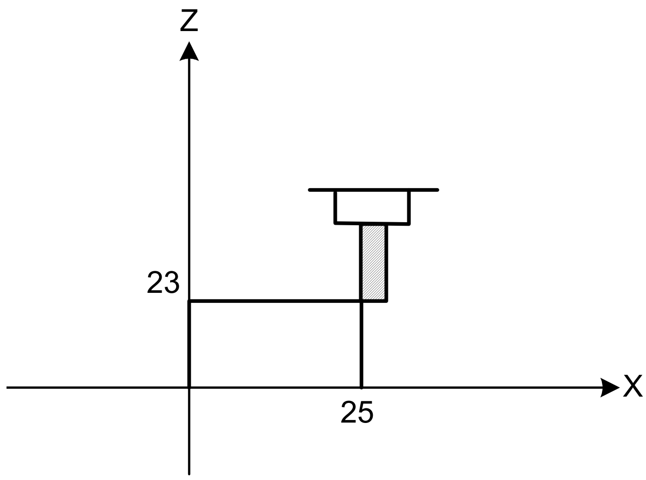


Fig. 4-2-11-2

As shown in Figure 4-2-11-2, G92 X25 Z23, the position of the tool tip is used as the point (X25, Z23) in the floating coordinate system.

2) Use a fixed point on the tool holder as the reference point coordinate system:

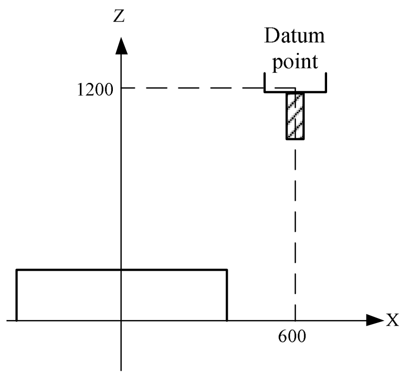


Fig. 4-2-11-3

As shown in Figure 4-2-11-3, G92 X600 Z1200 is used to instruct the setting of coordinate system (when a reference point on the tool holder is used as cutting point). Using a reference point on the tool holder as the start point, if movement is made by the absolute value code in the program, the reference point is moved to the instructed position, adding the tool length compensation, which is the difference from the reference point to the tool tip.

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

Note 2: Tool radius compensation must be canceled before using the G92 code.

4.2.12 Plane Selection G17/G18/G19

Format: G17/G18/G19

Function: For circular interpolation, tool radius compensation or drilling, boring, plane selection is required. Now the plane is selected through G17/G18/G19.

Description: If without command parameters, the system defaults to the G17 plane at power-up. The default plane after power-up can also be determined by setting the position parameters **N0:31#1, #2, #3**. Correspondence between code and plane:

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

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

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

If G17, G18, and G19 are in the program segments that are not instructed, the plane does not change.

Example: G18 X_ Z_; ZX plane

G0 X_ Y_; plane remains unchanged (ZX plane)

In addition, the movement code is not related to plane selection. In the case of the following code, the Y-axis does not exist on the ZX plane, so the Y-axis movement is not related to the ZX plane.

G18Y_;

Hint: At present, only the fixed cycle under the G17 plane is available. When programming, to be normal and strict, it is better to specify the plane in the corresponding program segment, especially when several people share the same system. This will avoid accidents or exceptions caused by programming errors.

4.2.13 Polar Coordinate Beginning/Cancellation G16/G15

Format: G16/G15

Function:

G16 specifies the beginning of the polar coordinate representation of the position parameters.

G15 specifies the cancellation of the polar coordinate representation of the position parameters.

Description:

No command parameters.

Setting G16 enables input of coordinate values with polar coordinate radius and angle. The positive direction of the angle is the counterclockwise turn of the first axis on the selected plane, and the negative direction is the clockwise turn. Both the radius and the angle can be in absolute value code or incremental value code (G90, G91).

After G16 appears, of the position parameter of the tool movement command the first axis represents the polar radius in the polar coordinate system, and the second axis represents the polar angle in the polar coordinate system.

By setting G15, the polar coordinate mode can be canceled to return the coordinate value to input with Cartesian coordinates.

Requirement for polar coordinate origin:

1. In the G90 absolute mode, when the G16 mode command is used, the workpiece coordinate system zero is taken as the polar coordinate origin.

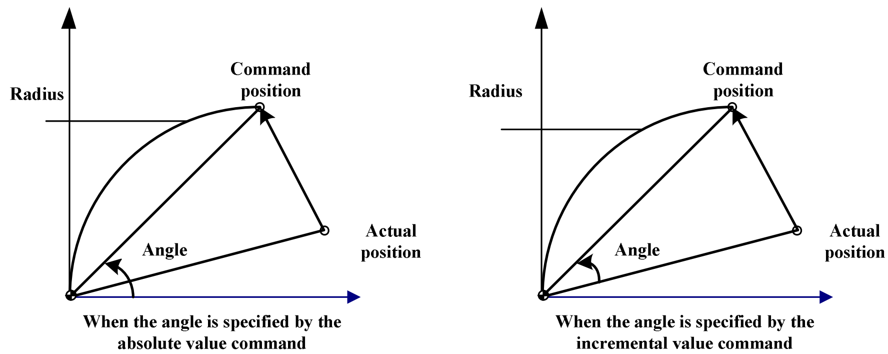


Fig. 4-2-13-1

2. In the G91 incremental mode, when the G16 mode command is used, the current point is taken as the polar coordinate origin.

For example: Bolt hole circle (the zero of the workpiece coordinate system is set as the polar coordinate origin, the X-Y plane is selected)

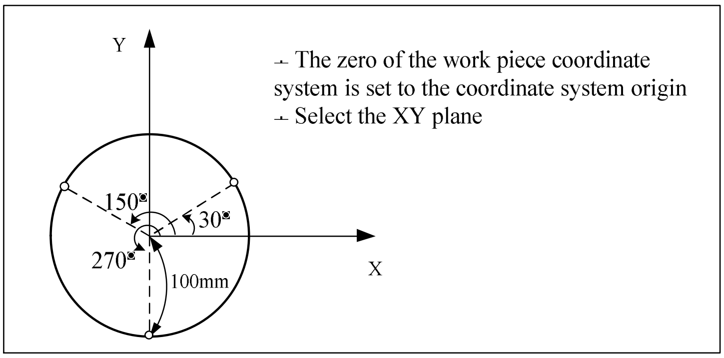


Fig. 4-2-13-2

- Specifying angle and radius by absolute values

  G17 G90 G16; specify the polar coordinate code and select the XY plane, set the zero of the workpiece coordinate system as the origin of the polar coordinate system
  G81 X100 Y30 Z-20 R -5 F200; specify a distance of 100mm and an angle of 30°
  Y150; specify a distance of 100mm and an angle of 150°
  Y270; specify a distance of 100mm and an angle of 270°
  G15 G80; cancel polar coordinate code

- Instructing angle by incremental value and polar radius by absolute value

  G17 G90 G16; specify the polar coordinate code and select the XY plane, set the zero of the workpiece coordinate system as the origin of the polar coordinate system
  G81 X100 Y30 Z-20 R -5 F200; specify a distance of 100mm and an angle of 30°
  G91 Y120; specify a distance of 100mm and an incremental angle of +120°
  Y120; specify a distance of 100mm and an incremental angle of +120°
  G15 G80; cancel polar coordinate code

In addition, when programming with polar coordinates, attention shall be paid to the setting of the current coordinate plane. The polar coordinate plane is related to the current coordinate plane. For example, in G91, if the current coordinate plane is G17, the X and Y-axis component of the current tool position is taken as the origin. If the current coordinate plane is G18, the Z and X-axis component of the current tool position is taken as the origin.

If the position parameter of the first hole cycle command is not specified after G16, the system uses the current position of the tool as the default position parameter of the hole cycle. The first fixed cycle code must be complete after the current polar coordinates. Otherwise, the tool movement is incorrect.

After G16, in addition to the hole cycle, the parameter word of the tool movement command position parameter is related to the specific plane selection mode. When the movement code follows immediately after canceling the polar coordinates with the G15 code, the current tool position is considered the start point of this movement code by default.

4.2.14 Scaling In The Plane G51/G50

Format:

G51 X_ Y_ Z_ P_ (X.Y.Z: The absolute value code of the scaling center coordinate value, P: Each axis is scaled equally)
… Scaled machining program segment
G50 Scaling cancellation
Or G51 X_ Y_Z_ I_ J_ K_ (the axes are scaled differently (I, J, K))
… Scaled machining program segment
G50 Scaling cancellation

Function:

G51 enables the programmed shape to be centered at the specified position, zooming in and out at the same or different scales. It shall be noted that G51 is recommended to specify a separate program segment (otherwise unexpected conditions may occur, causing damage to the workpiece and personal injury) and cancellation made with G50.

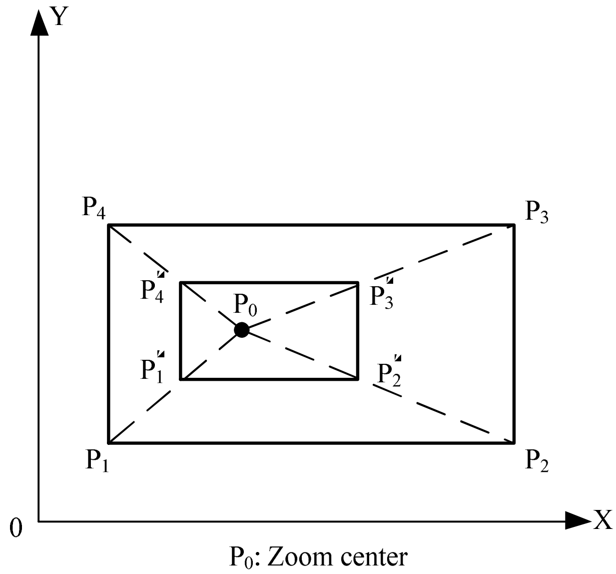


Figure 4-2-14-1 Scaling (P1P2P3P4 → P1'P2'P3'P4')

Description:

1. Zoom center: G51 can carry 3 position parameters X_Y_Z_, which are optional. The position parameters are used to specify the zero of G51. If no position parameter is specified, the system sets the current position of the tool as the zero. Regardless of whether the current positioning mode is absolute or incremental, the zero is specified by absolute positioning. In addition, in the polar coordinate G16 mode, the parameters in the G51 code are also expressed in a Cartesian coordinate system.
   Example: G17 G91 G54 G0 X10 Y10;
   G51 X40 Y40 P2; incremental mode, the zero is still the absolute coordinate in the G54 coordinate system (40, 40)
   G1 Y90; parameter Y is still incremental
2. Scaling: Regardless of whether the current mode is G90 or G91, the scaling is always expressed in absolute terms.
   In addition to being specified in the programs, scaling can also be set in the parameters. The data parameter P330 sets the scaling ratio for each axis, and the data parameters **P331~P333** correspond to the scaling ratios of the first, second, and third axes, respectively. If there is no scaling ratio code, when the position parameter N0:47#6 is set to 0, scaling is performed by the set value of the data parameter P330; when the position parameter N0:47#6 is set to 1, scaling is performed by the set value of data parameters P331~P333.
   If the value of the parameter P or I, J, K is specified as negative, the corresponding axis is mirrored.
3. Scaling settings: Position parameter **No: 60#5** sets whether the scaling function is used, position parameter **N0:47#3** sets whether the first axis scaling is valid, position parameter **N0:47#4** sets whether the second axis scaling is valid, position parameter **N0:47#5** sets whether the third axis scaling is valid, and position parameter **N0:47#6** sets the way how the scaling ratio is specified for each axis (0: P command for each axis; 1: I, J, or K command for each axis).
4. Scaling cancellation: When the movement code follows immediately after canceling the scaling with the G50 code, the coordinate scaling is canceled, and the tool position is considered the start point of this movement code.
5. In scaling status, it is not allowed to instruct the G codes (G27~G30, etc.) for return to the reference point and G codes (G53~G59, G54P1~G54P50, G92, etc.) for coordinate systems. If those G codes must be specified, they shall be specified after canceling the

scaling function, otherwise the system will alarm.

6. Even if different scaling ratios are specified for circular interpolation and each axis, the tool does not draw an elliptical path.

   When the scaling ratio of each axis is different, and the circular interpolation is programmed with the radius R, the interpolation figure is shown in Figure 4-2-14-2 (in the following example, the ratio is 2 for the X axis and 1 for the Y axis).

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

The above command is equivalent to the command below:

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

The ratio of radius (R) is zoomed based on I or J (whichever is larger).

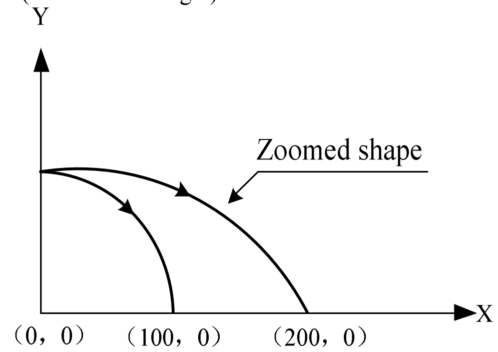


Figure 4-2-14-2 Scaling of circular interpolation

   When the scaling ratio of each axis is different, and the circular interpolation is programmed with I, J, or K, the system will alarm if the arc is not established.

7. Scaling is invalid for tool offset values, as shown in Figure 4-2-14-3.

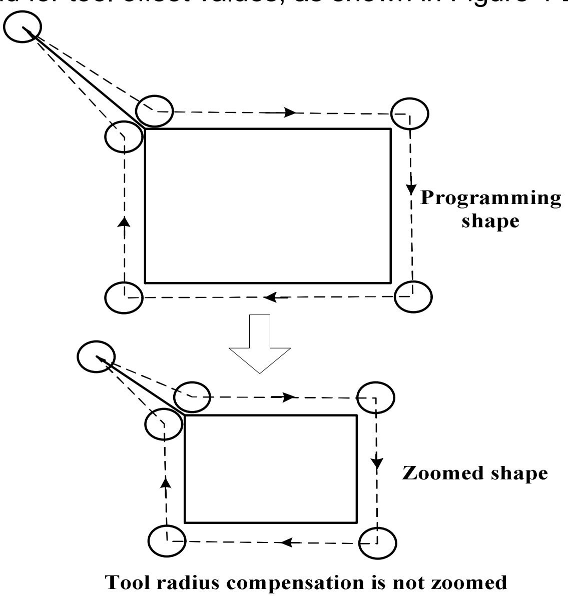


Figure 4-2-14-3 Scaling for tool radius compensation

Examples of mirrored programs:

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;
```

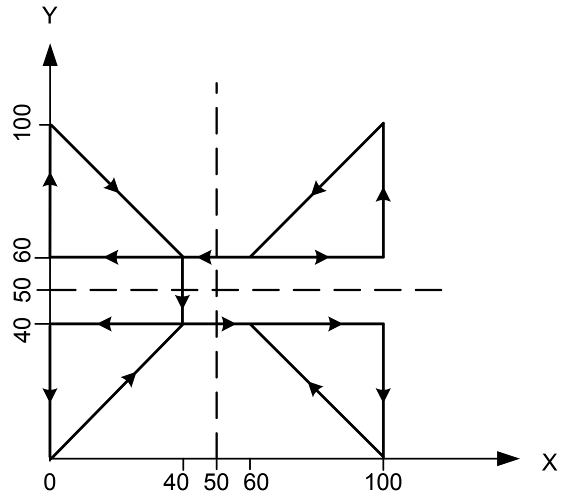


Fig. 4-2-14-4

Restrictions:

1. In the fixed cycle, the movement and scaling are invalid for Z-axis cut-in value Q, Z depth, return value d, Z-axis first cutting depth W and fast entry distance V.
2. When running manually, the movement distance cannot be increased or decreased with the scaling function.

Note 1: The position shows the scaled coordinate value.

Note 2: When there is an axis performing mirroring on the specified plane, the result is as follows:

1) Arc code............ .... Reverse direction of rotation
2) Tool radius compensation C.......... Reverse offset direction
3) Coordinate system rotation.............Reverse rotation angle
4) Change the direction of cutting feed

4.2.15 Coordinate System Rotation G68/G69

When the machining workpiece consists of many patterns of the same shape, it can be programmed with the coordinate rotation function by simply writing subprograms for the pattern elements and calling the subprograms through 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_: The absolute command of the two axes of X_, Y_ and Z_ corresponding to the specified coordinate plane (G17, G18, G19), which specifies the rotation center after G68.
R_: Angle displacement, with the positive value representing counterclockwise rotation.

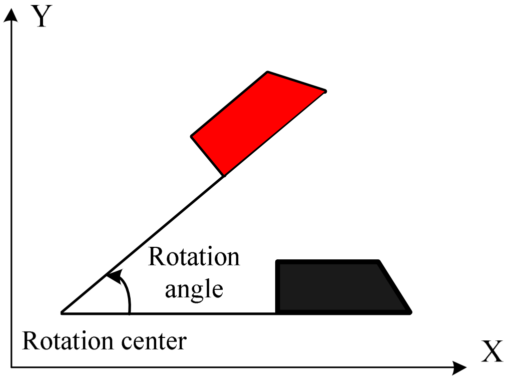


Fig. 4-2-15-1

Function: G68 enables the in-plane programmed shape to rotate around the specified center as the origin. G69 is used to cancel the rotation of the coordinate system.

Description:

1. G68 can carry 2 position parameters, which are optional. The position parameters are used to specify the center of the rotation. If no center of rotation is specified, the system uses the current tool position as the center of rotation. The position parameters are related to the current coordinate plane. X and Y are selected in G17; Z and X are selected in G18; Y and Z are selected in G19.
2. When the current positioning mode is absolute, the system uses the specified point as the center of rotation. When the positioning mode is relative, the system specifies the current point as the center of rotation. G68 can also carry a command parameter R, whose parameter value is the angle of rotation with positive value representing counterclockwise rotation. The angle of rotation is expressed in degrees. The angle of rotation used when there is no rotation angle code in the coordinate rotation is set by the data parameter P329.
3. In the G91 mode, the system uses the current tool position as the center of rotation; whether increment is performed on the angle of rotation is set by the position parameter **NO:47#0** (angle of coordinate rotation, 0: absolute code, 1: G90/G91 code).
4. When the system is in the rotation mode, the plane selection operation is not allowed, or an alarm will occur. Care shall be taken when programming.
5. In the coordinate system rotation mode, it is not allowed to instruct the G codes (G27~G30, etc.) for return to the reference point and G codes (G53~G59, G54P1~G54P50, G92, etc.) for coordinate systems. If those G codes must be specified, they shall be specified after canceling the rotation function, otherwise the system will alarm.
6. After the coordinate system rotation, tool radius compensation, tool length compensation, tool offset and other compensation operations can be performed.
7. When the coordinate system rotation code is executed in the scaling mode (G51), the coordinate value of the rotation center is also scaled, but the angle of rotation is not scaled. When the movement code is issued, the scaling is performed, followed by coordinate rotation.

Example 1: Rotation:

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

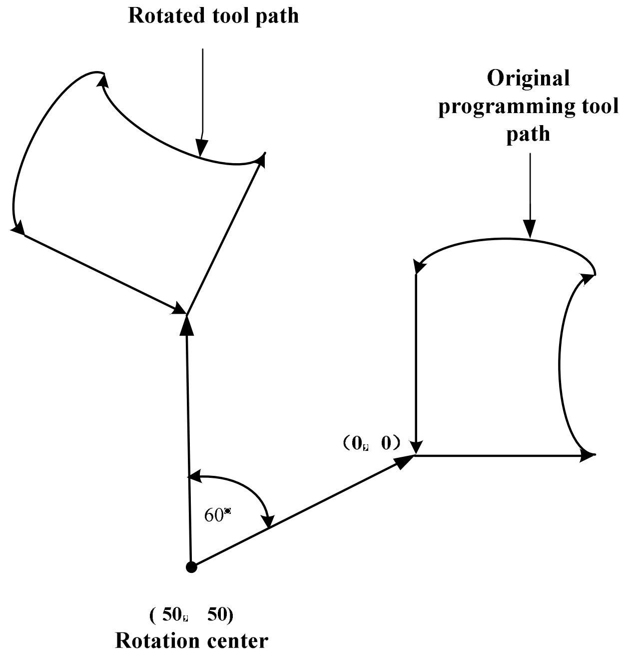


Fig. 4-2-15-2

Example 2: Scaling + rotation:

G51 X300 Y150 P0.5;
G68 X200 Y100 R45;
G01 G90 X400 Y100;
G91 Y100;
X-200;
Y-100;
X200;
G69 G50;
M30;

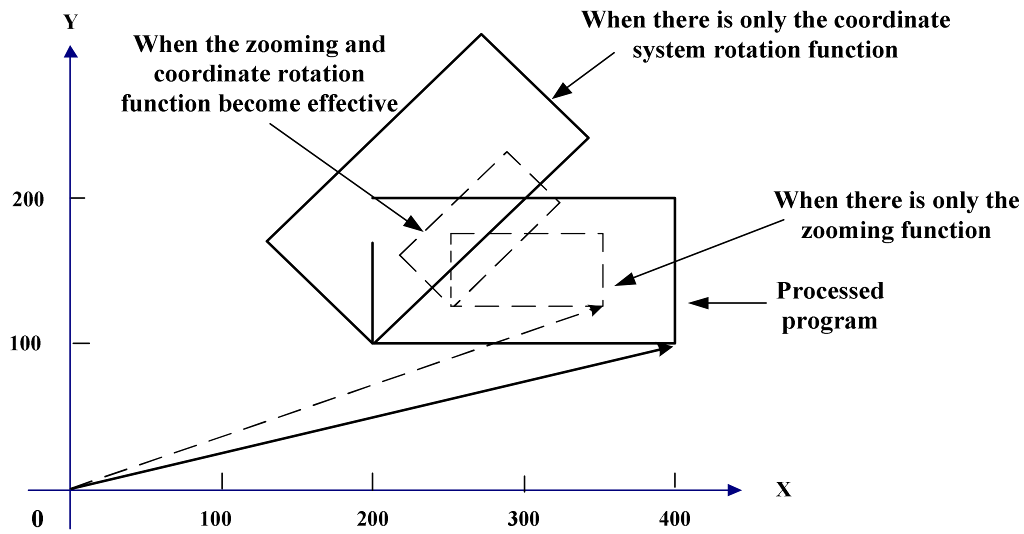


Fig. 4-2-15-3

Example 3: Repeating G68

Based on program (main program)

```
G92 X0 Y0 Z20 G69 G17;
M3 S1000;
G0 Z2;
G42 D01;                (Tool offset setting)
M98 P2100 (P02100);     (Subprogram call)
M98 P2200L7;            (Call 7 times)
G40;
G0 G90 Z20;
X0Y0;
M30;
Subprogram 2200
O2200
G91
G68 X0 Y0 R45.0;        (Relative rotation angle)
G90;
M98 P2100;              (Subprogram O2200 calls subprogram O2100)
M99;
Subprogram 2100
O2100 G90 G0 X0 Y-20; (Right tool compensation mode is established)
G01Z－2 F200;
X8.284;
X14.142 Y-14.142;
M99;
```

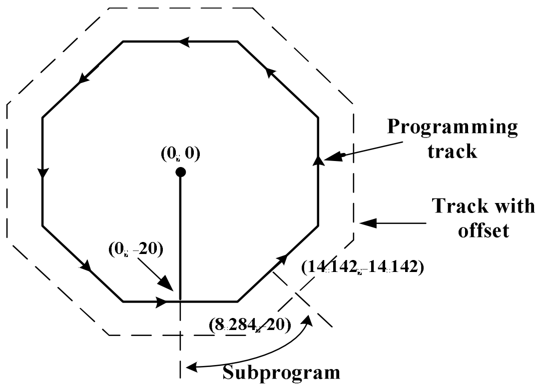


Fig. 4-2-15-4

4.2.16 Skip Function G31

Format: G31 X_Y_Z_

Function: After the G31 code, as with G01 linear interpolation can be instructed. If an external skip signal is input during the execution of the code, the execution of the code is interrupted for the execution of the next program segment. The skip function is used when the machining end point is not programmed, but is specified with a signal from the machine tool. For example, it is used for grinding. The skip function is also used to measure the workpiece size.

Description:

1. G31 is a non-modal G code and is valid only in the specified program segment.
2. When applying the tool radius compensation, if the G31 code is issued, the alarm will appear. The tool radius compensation shall be canceled before the G31 code.

Example:

The program segment following G31 is the single-axis movement instructed by the incremental value, as shown in Figure 4-2-16-1:

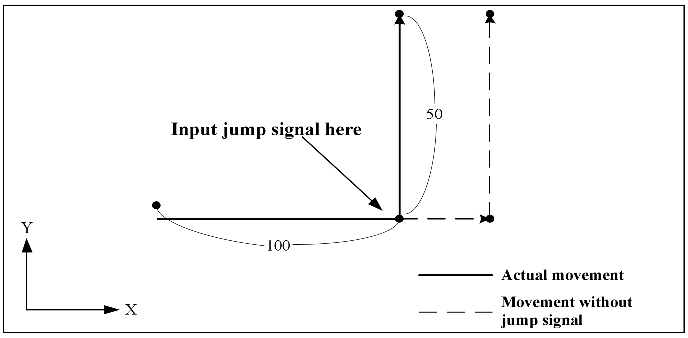


Figure 4-2-16-1 The program segment following is the single-axis movement instructed by the incremental value

The program segment following G31 is the single-axis movement instructed by the absolute value, as shown in Figure 4-2-16-2.

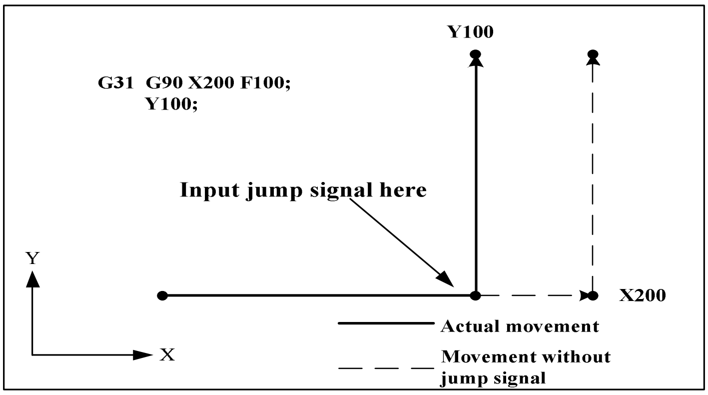


Figure 4-2-16-2 The program segment following is the single-axis movement instructed by the absolute value

The program segment following G31 is the two-axis movement instructed by the absolute value, as shown in Figure 4-2-16-3:

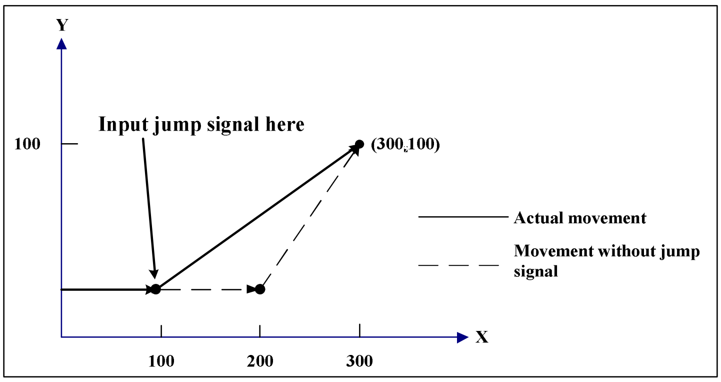


Figure 4-2-16-3 The program segment following is the two-axis movement instructed by the absolute value

Note: It can be set by position parameter NO: 04#7 (input as signal when the skip signal (0: 1, 1: 0).

4.2.17 Imperial/Metric Conversion of G20/G21

Format: G20: Imperial input
G21: Metric input

Function: It enables imperial/metric conversion of program input.
Description:

After the imperial/metric conversion, the units of the following values are changed:
F code instructed feed speed, position code, workpiece zero offset value, tool compensation value, scale unit of the manual pulse generator, and movement distance in incremental feed.
At power-on the G code is in the same state as what is before power-off.

Note: 1. When the imperial is converted to metric or the other way around, the tool compensation value must be preset based on the least input increment.
2. When the imperial is converted to the metric or the other way around, the operation of the first G28 code from the middle point is the same as the manual return to reference point.
3. When the least input increment and the least command increment are different, the maximum error is half of the least command increment, and this error does not accumulate.
4. The program input imperial/metric can be set by position parameter **N0:00#2**.
5. G20 or G21 must be specified in a separate program segment.

4.2.18 Any Angle Chamfer/Corner Arc

Format: , L_: Chamfer
, R_: Corner arc transition
Function: When the above code is added at the end of the linear interpolation (G01) or circular interpolation (G02, G03) program segment, the chamfer or transition arc is automatically added to the corner during machining. Program segments for chamfer or corner arc transition can be specified continuously.
Description:

1. Chamfer is behind L, specifying the distance from the virtual inflection point to the start and end points of the corner. The virtual inflection point is the actual corner point if it is assumed chamfer is not executed. Shown as follows:

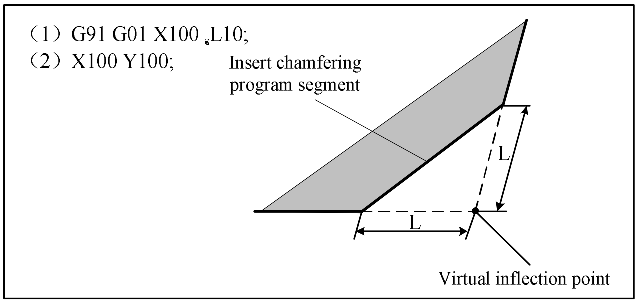


Fig. 4-2-18-1

2. Corner arc transition is behind R, specifying the radius of the corner arc, as shown below:

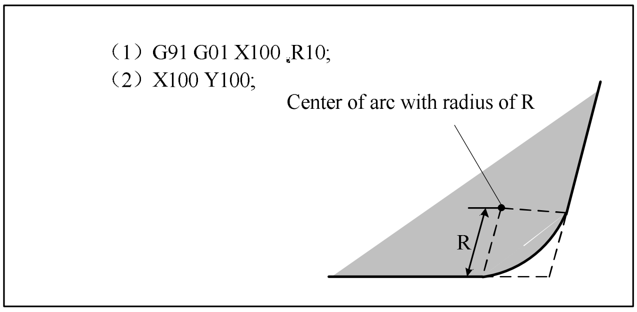


Fig. 4-2-18-2

Restrictions:

1. Chamfer and corner arc can only be executed in the specified plane. These functions cannot be executed on parallel axes.
2. If the inserted chamfer or arc transition program segment causes the tool to exceed the

original range of interpolation movement, an alarm will occur.
3. The corner arc transition cannot be specified in the thread machining program segments.
4. When the instructed chamfer value and corner value are negative, the system takes their absolute values.

4.3 Reference Point G Code

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

For the reference point, there are three code operation modes. As shown in Figure 4-3-1, G28 enables the tool to move through the middle point and automatically move to the reference point along the specified axis in the code. G29 enables the tool to move from the reference point, through the middle point, and automatically move to the specified point along the specified axis in the code.

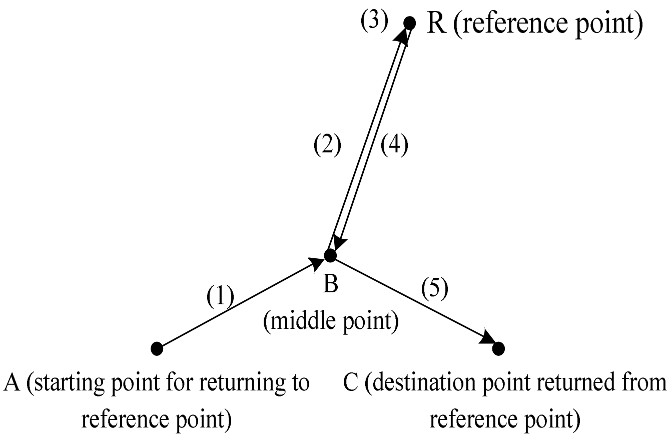


Fig. 4-3-1

4.3.1 Return To Reference Point G28

Format: G28 X_ Y_ Z_
Function: The G28 code is used to perform an operation that returns a reference point (a specific position on the machine) through a middle point.
Description:
Middle point:
The middle point is specified by the code parameter in G28 and can be represented by an absolute value code or an incremental value code. When this program segment is executed, the coordinate value of the middle point on the code axis is also stored for use by the G29 (return from reference point) code.
Note:
The coordinates of the middle points are stored in the **CNC**, but only the coordinate values on the axes instructed by G28 are stored at a time, and for other axes that are not instructed, the coordinate values on the axes previously instructed by G28 are used. Therefore, when the user uses G28 command, if the default middle point in the current system is unknown, it is best to specify each axis. Please consider with the N5 program segment in Example 1 below.

Volume I Programming instructions

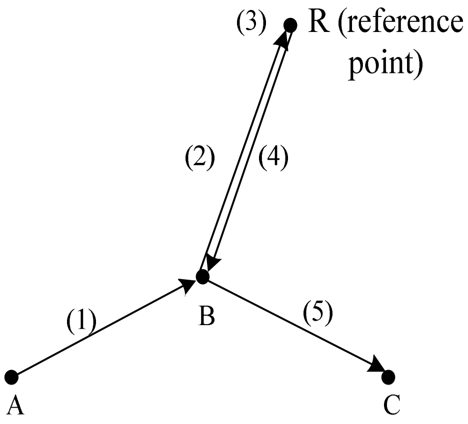


Fig. 4-3-1-1

1. The action of the G28 program segment can be decomposed into the following steps (see Figure **4-3-1-1**):

    (1) Move fast from the current position to the position of the middle point on the instructed axis (point A → point B).

    (2) Move fast from the middle point to the reference point (point B → point R).

2. G28 is a non-modal code and is valid only for the current segment.
3. Return to reference point is available in a combined way of single-axis or multi-axis movement. During the workpiece coordinate transformation, the coordinates of the middle point are stored in the CNC.

Example:

N1 G90 G54 X0 Y10;

N2 G28 X40; Set the middle point on the X axis to X40 in the G54 workpiece coordinate system, and return to the reference point through the point (40, 10), that is, return to reference point on the separate X axis.

N3 G29 X30; Return from the reference point to the point (30, 10) through the point (40, 10), that is, return to target point on the separate X axis.

N4 G01 X20;

N5 G28 Y60; Middle point Y60.

N6 G55; When the workpiece coordinate system is transformed, the middle point is changed from the point (40, 60) in the G54 workpiece coordinate system to the point (40, 60) in the G55 workpiece coordinate system.

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

G28 will automatically cancel the tool compensation. But this code is generally used during automatic tool change (that is, tool change at the reference point after returning to the reference point), so when using this code, in principle, the tool radius compensation and tool length compensation must be canceled first. 1st reference point setting

See data parameters P45~P49

4.3.2 Return To Reference Points 2, 3 and 4 (G30)

There are 4 reference points in the coordinate system, but for systems without absolute position detector, it is possible to return to Reference Points 2, 3 and 4 only when Automatic Return to Reference Point (G28) or Manual Return to Reference Point is executed.

Format:

G30 P2 X_ Y_ Z_; return to Reference Point 2 (P2 can be omitted)

G30 P3 X_ Y_ Z_; return to Reference Point 3

G30 P4 X_ Y_ Z_; return to Reference Point 4

Function: Execute G30 for return to the designated reference point through the intermediate point specified in G30.

Description:

1. X_ Y_ Z; specify the intermediate position (absolute value/incremental value code)
2. G30 shares the same setting and limitations with G28. Please see Data Parameters **P50 - P63** for setting of Reference Points 2, 3 and 4.
3. G30 can also be used together with G29 (return from a reference point) code, sharing the same setting and limitations with G28.

4.3.3 Automatic Return From A Reference Point (G29)

Format: G29 X_ Y_ Z_

Function: Execute G29 for return from a reference point (or the current point) through the intermediate point specified in G28 and G30.

Description:

1. The action in the G29 program segment can be decomposed into the following steps (see Figure 4-3-1-1):
   (1) Locate to the intermediate point specified in G28 and G30 from a reference point (or the current point) in a fast moving manner (Point R → Point B).
   (2) Locate to the designated point from the intermediate point in a fast moving manner (Point B → Point C).
2. G29 is non-modal information and only applicable to the current segment. In general, after definition of G28 and G30, you should immediately specify the code for return from a reference point.
3. The optional parameters X, Y and Z in the G29 code format are used to specify the target point for return from a reference point (i.e. Point C in Figure 4-3-1-1), and can be expressed with absolute value code or incremental value code. Program the incremental value and use the code value to specify the incremental value leaving the intermediate point. The case that some axle is not specified means that there is no movement of the axle relative to the intermediate point. G29 only followed by one-axle command means one-axle return and the other axles remain inactive.

Example:

G90 G0 X10 Y10;
G91 G28 X20 Y20; return to a reference point via the intermediate point (30, 30)
G29 X30; return from a reference point to the point (60, 30) via the intermediate point (30, 30), and attention should be paid to the incremental programming mode and the component of the X axis should be 60.

The intermediate point specified by G29 is assigned by G28 and G30. Definition, specification and system defaults of an intermediate point are detailed in the description of G28.

4.3.4 Return To A Reference Point For Testing (G27)

Format: G27 X_ Y_ Z_

Function: Execute G27 for return to a reference point for testing and use X_ Y_ Z_ to specify the reference point.

Description:

1. G27 enables the tool to locate in a fast moving manner. If the tool reaches the reference point, a signal of successful return will be triggered; if the tool arrives at a position other than the reference point, an alarm will be triggered.
2. For a machine tool in locked state, even if the G27 code is specified and the tool has automatically returned to the reference point, the signal of successful return will not be triggered.
3. In the offset mode, the position where the tool reaches as specified by G27 is a result of addition of the offset value. Therefore, if the position plus the offset value is not the reference position, the signal will fail and the alarm will be triggered. Usually, the tool offset should be canceled before using G27.

4. The position defined by the X, Y, and Z coordinate points specified by G27 is the position of the machine tool in the coordinate system.

4.4 Fixed Cycle (G Code)

A fixed cycle uses a program segment containing G functions to enable processing activities which may have been realized by multiple program segments, so as to simplify the programming. And it also makes it easy for programmers to program (this system only has a fixed cycle of the G17 plane).

General process of a fixed cycle:

A fixed cycle usually consists of 6 activities in sequence, as shown in Figure 4-4-1.

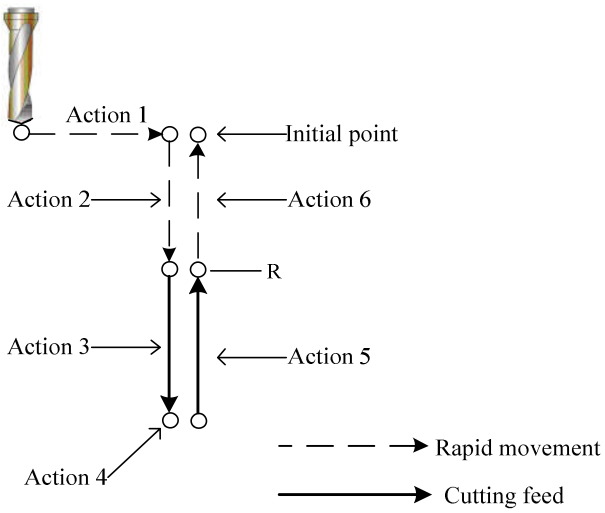


Fig. 4-4-1

Activity 1: positioning in the X and Y axis (it may also include another axis)
Activity 2: quickly move to Point R.
Activity 3: hole machining
Activity 4: operations at the bottom of the hole
Activity 5: return to Point R
Activity 6: quickly move to the initial point

Positioning is conducted in the XY plane, and hole machining is performed in the Z-axis direction. Specified activities in a fixed cycle depend on three ways. They are specified by G code.

1) Data form
   G90 absolute value; G91 incremental value
2) Return to the point plane
   G98 initial point plane; G99 R point plane
3) Hole machining method
   G73, G74, G76, G81 - G89

Initial point Z plane and R point plane

Initial point plane: It is the absolute position of the tool in the Z-axis direction before start of the fixed cycle

R point plane: Also known as safety plane, it is the position in the Z-axis direction when shifting from rapid movement to cutting feed in a fixed cycle, usually set at a certain distance above the surface of a workpiece to prevent the tool from hitting the workpiece and ensure sufficient distance to complete the acceleration process.

All data related to the fixed cycle (including data about hole position, hole machining and number of repetitions) are specified by G73/G74/G76/G81 - G89 to form a program segment.

Z, R: In case of lack of any of Parameters Z and R when drilling is conducted to form the first hole, the system will only change its mode and perform the Z-axis activities.

The format for 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

The basic meanings of hole position data and hole machining data are shown in Table 4-4-1.

Table 4-4-1

Specified content	Parameter	Description
Hole machining method	G	Please refer to Table **4-4-3** and pay attention to the above restrictions.
Hole position data	X，Y	Use absolute or incremental value to specify the hole position, and use the same control as G00 in terms of positioning.
Hole machining data	Z	As shown in Figure 4-4-2, use the incremental value to specify the distance from Point R to the hole bottom or use the absolute value to specify the coordinate values of the hole bottom. The feed rate is shown in Figure 4-4-1. Use the speed specified by F in Activity 3. Choose fast feeding in Activity 5 based on different hole machining methods or use the speed specified by F code.
	R	As shown in Figure 4-4-2, use the incremental value to specify the distance from the initial point plane to Point R, or use the absolute value to specify the coordinate values of Point R. The feed rate is shown in Figure 4-4-1. Fast feeding is used in Activities 2 and 6.
	Q	Specify G73, the depth per cutting in G83, or G76, translation distance in G87 (incremental value)
	P	Specify the pause time at the bottom of the hole. The fixed cycle code can have a parameter P_, which is used to specify the pause time when the tool has reached the Z plane. Unit: ms. The minimum value of the parameter is set by Data Parameter P296, while the maximum value is set by P297.
	F	Specify the cutting feed rate
	K	Specify the number of repetitions in the parameter value of K_, and K is valid only in the specified program segment. It can be omitted with the default of one repetition. The maximum frequency of drilling is 99999. When a negative value is specified, it is executed according to its absolute value. When it is zero, the drilling operation is not performed. Only change the mode

Restrictions:

- The fixed cycle G is a modal code that remains valid until the G code that cancels the fixed cycle is specified.
- The G code that cancels the fixed cycle includes G80 and the G code in Group 01.
- Once the machining data is specified in the fixed cycle, it will remain valid until the fixed cycle is canceled. Therefore, all the necessary hole machining data should be specified at the beginning of the fixed cycle, and only the data to be changed will be specified in the subsequent fixed cycle.

Note 1: The cutting speed in the F command will be maintained even if the fixed cycle is canceled.

Note 2: When the cycle is fixed, the Z-axis (cutting axis) scaling function is invalid.
Note 3: In the single-segment mode, the fixed cycle generally adopts a three-stage machining mode: positioning → R plane → initial plane.
Note 4: In the fixed cycle, when the system bit parameter NO: 36#1 is 1, in case of reset or emergency stop, the hole machining data and hole position data will be eliminated. Examples of maintained and eliminated data above mentioned are shown below.

Table 4-4-2

Sequence	Designation of data	Description
①	G00X_M3;	
②	G81X_Y_Z_R_F_;	The required values for Z, R and F should be assigned at the beginning.
③	Y_;	Due to sharing the same hole machining method and hole machining data specified in Hole ②, G81 and Z-R-F- can be omitted. The hole moves by Y, and is processed once with the G81 method
④	G82X_P_;	Only move in the X-axis direction relative to the position of Hole ③. Machine the hole with the G82 method, and use Z, R and F specified in ② and P specified in ④ as hole machining data to machine the hole.
⑤	G80X_ Y_	No hole machining. Cancel all of the hole machining data.
⑥	G85X_Z_R_P_;	Since all the data is canceled in ⑤, Z and R need to be specified again. F is the same as that specified in ②, so it can be omitted. P is not needed in this program segment, and thus just save it
⑦	X_Z_;	Hole machining with the Z value different from that in ⑥. And the hole only moves in the X-axis direction.
⑧	G89X_Y_;	Use Z specified in ⑦, R and P specified in ⑥ and F specified in ② as the hole machining data to machine the hole with the G89 method.
⑨	G01X_Y_;	Eliminate the hole machining method and relevant data.

A. Absolute value/incremental value codes of the fixed cycle (G90/G91)

The change in the movement distance along the drilling axis relative to G90 and G91 is as shown in Figure 4-4-2 (usually programmed with G90. If programmed with G91, then Z and R are processed according to the plus-minus sign of the command).

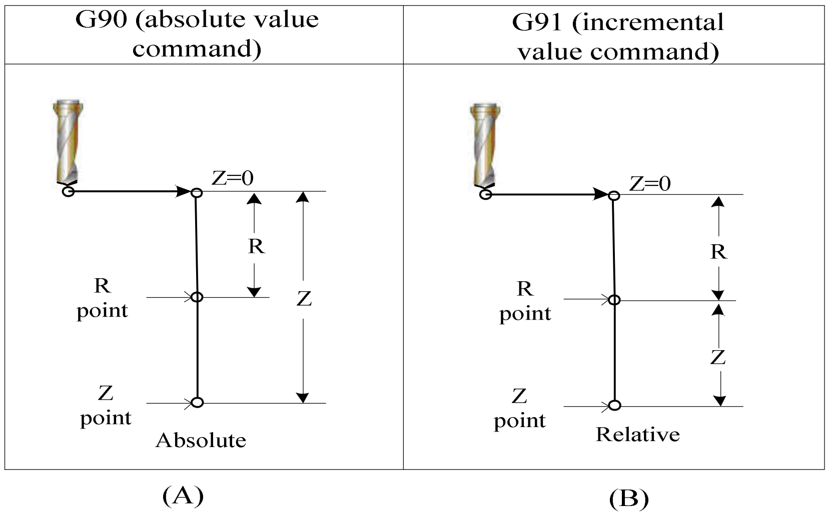


Fig. 4-4-2

B. Code for the fixed cycle to return to the plane (G98/G99)

When the tool reaches the bottom of the hole, it can return to the R point plane or the initial position plane. The tool can return to the initial point plane or the R point plane depending on G98 and G99.

In general, G99 is used for the first drilling and G98 is used for the last drilling. Even if the hole is machined using G99, the initial plane will remain unchanged. The activities included in codes G98 and G99 are shown in the figure below.

The system will choose G98 by default.

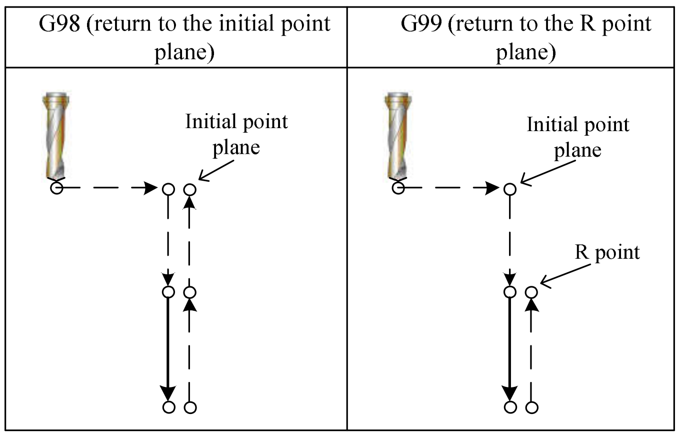


Fig. 4-4-3

Each fixed cycle diagram will use following symbols:

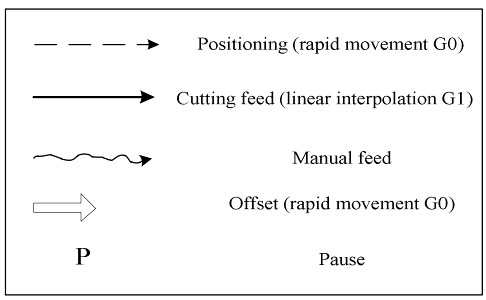


Fig. 4-4-4

Fixed Cycle Comparison Table (G73 - G89)

Table 4-4-3

G code	Drilling (-Z direction)	Bottom activity	Retracting (+Z direction)	Purpose
G73	Intermittent feed		Rapid movement	High-speed deep hole machining
G74	Cutting feed	Suspend the positive rotation of spindle	Rapid movement	Anti-tapping cycle
G76	Cutting feed	Spindle orientation stops	Rapid movement	Precision boring
G80				Cancel the fixed cycle
G81	Cutting feed		Rapid movement	Drilling, spot drilling
G82	Cutting feed	Stop cutting	Rapid movement	Drilling, boring step hole
G83	Intermittent feed		Rapid movement	Deep hole machining cycle
G84	Cutting feed	Stop cutting → negative rotation of spindle	Cutting feed	Tapping
G85	Cutting feed		Cutting feed	Boring
G86	Cutting feed	Spindle stop	Rapid movement	Boring
G87	Cutting feed	Positive rotation of spindle	Rapid movement	Boring
G88	Cutting feed	Pause → Spindle stop	Manual → Positive rotation of spindle	Boring
G89	Cutting feed	Pause	Cutting feed	Boring

Restrictions:

The tool radius offset (D) will be ignored during fixed cycle positioning.

4.4.1 High-Speed Deep Hole Machining Cycle (G73)

Format: G73 X_Y_Z_R_Q_F_K_

Function: This cycle is designed to perform high-speed deep drilling, which performs intermittent cutting feed to the bottom of the hole, and quickly retracts from the hole while feeding and removes the chips. The activity diagram is shown in Figure 4-4-1-1.

Description:

X_Y: Hole positioning data;
Z: Incremental programming defines the distance from Point R to the bottom of the hole; absolute programming defines the absolute coordinate values of the bottom of the hole;
R: Incremental programming defines the distance from the initial point plane to Point R; absolute programming defines the absolute coordinate values of Point R;
Q: Cutting depth of each cutting feed;
F: Cutting feed rate;
K: Repetition times

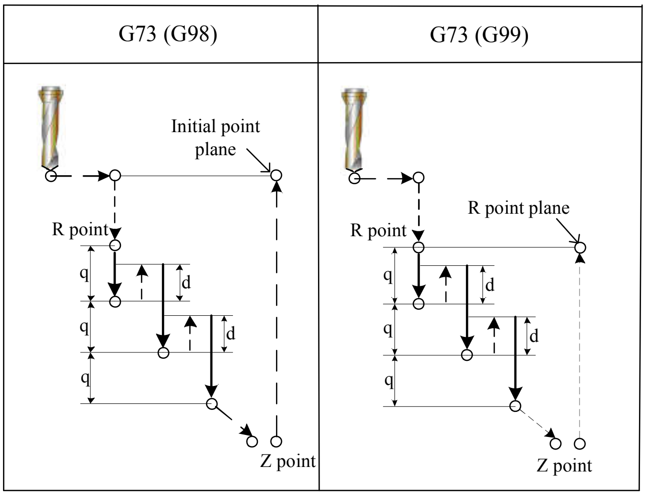


Fig. 4-4-1-1

Z, R: In case of lack of both of Parameters Z and R when drilling is conducted to form the first hole, the system will perform the Z-axis activities.

Q: When the code parameter Q is specified, the intermittent feed as shown in the figure above will be performed. At the moment, the system will retract using the retraction amount d (as shown in Fig. 4-4-1-1.) set in Data Parameter P270, and the tool will intermittently perform the fast moving retraction with the distance d for each feed.

When **G73** and the M code are specified by the same program segment, the M code is executed during positioning of the first hole, and then the system deals with the next drilling activity.

When repetition times K is specified, the M code is executed only for the first hole, and not for subsequent holes.

Note 1: In the current version, M00, M01, M02, M06, M30, M98, and M99 are post-program executed, and the above M code will be executed after the current segment is executed.

Note 2: When the bit parameter NO: 43# 1=0, in case of deep drilling (G73, G83) without specified cutting depth, no alarm will be triggered, and in this case, the code parameter Q is not specified or Q is specified as 0, and the system will perform hole positioning in the X and Y planes, but not perform drilling. When the bit parameter NO: 43#1=1, in case of deep drilling (G73, G83) without specified cutting depth, an alarm will be triggered. That is, when the code parameter Q is not specified or Q is specified as 0, the system will provide the alarm: "0045: Address Q is not found or Q value is 0 (G73/G83)". If Q is specified as a negative value, the system will perform intermittent feed with its absolute value.

Note 3: Tool length compensation: When the tool length compensation G43, G44 or G49 shares the same program segment with the fixed cycle command, add or cancel the offset when positioning to Point R; in the fixed cycle mode, when the tool compensation G43, G44 or G49 is alone in a program segment, the system can add or cancel the offset in real time.

Restrictions:

When the G73 command and the G code (G00 to G03, G60 are modal codes (when the position parameter NO: 48#0 is 1)) in Group 01 of the same program segment are used, the system will execute the final modal code.

Tool radius compensation: In this fixed cycle command, the tool radius compensation will be ignored because the command function does not require tool radius compensation.

Example:

M3 S1500; the spindle starts rotating.
G90 G99 G73 X0 Y0 Z-15 R-10 Q5 F120; positioning, Hole 1 drilling, and then return to Point R
Y-50; Positioning, Hole 2 drilling, and then return to Point R
Y-80; Positioning, Hole 3 drilling, and then return to Point R
Y-10; Positioning, Hole 4 drilling, and then return to Point R
Y-10; Positioning, Hole 5 drilling, and then return to Point R
G98 Y75; Positioning, Hole 6 drilling, and then return to the initial position plane
G80;
G28 G91 X0 Y0 Z0; return to reference point

M5; the spindle stops rotating
M30;

Note: In the case of Holes 2 - 6 machining in the above examples, although Q is omitted, the chips will also be removed.

4.4.2 Drilling Cycle and Point Drilling Cycle (G81)

Format: G81 X_ Y_ Z_ R_ F_ K_
Function: This cycle is used for normal drilling cutting feed, which is carried out to the bottom of the hole, and then the tool will quickly retract from the bottom.
Description:
X_Y_: Hole positioning data;
Z_: Incremental programming defines the distance from Point R to the bottom of the hole; absolute programming defines the absolute coordinate values of the bottom of the hole;
R_: Incremental programming defines the distance from the initial point plane to Point R; absolute programming defines the absolute coordinate values of Point R;
F_: Cutting feed rate;
K_: Repetition times (if necessary).

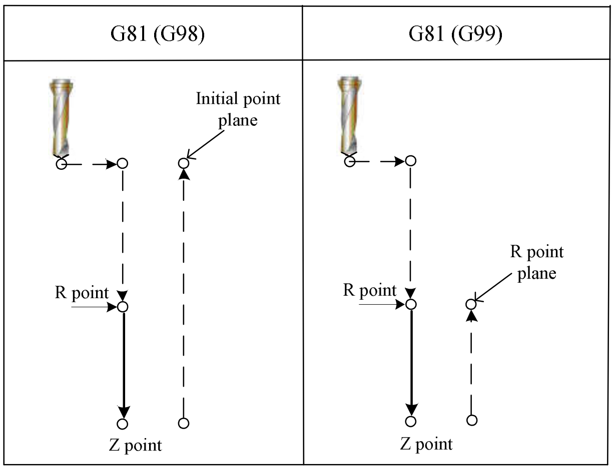


Fig. 4-4-2-1

Z, R: In case of lack of any of Parameters Z and R when drilling is conducted to form the first hole, the system will only change its mode and not perform the Z-axis activities.
After positioning along the X and Y axis, move quickly to Point R, perform drilling from Point R to Point Z, and then quickly retract the tool.
Rotate the spindle with the auxiliary function M code before G81 is specified.
When G81 and the M code are specified by the same program segment, the M code is executed during positioning of the first hole, and then the system deals with the next drilling activity.
When repetition times K is specified, the M code is executed only for the first hole, and not for subsequent holes.

Note: In the current version, M00, M01, M02, M06, M30, M98, and M99 are post-program executed, and the above M code will be executed after the current segment is executed.

Tool length compensation: When the tool length compensation G43, G44 or G49 shares the same program segment with the fixed cycle command, add or cancel the offset when positioning to Point R; in the fixed cycle mode, when the tool compensation G43, G44 or G49 is alone in a program segment, the system can add or cancel the offset in real time.

Example:
M3 S2000; the spindle starts rotating
G90 G99 G81 X300 Y-250 Z-150 R-10 F120; positioning, Hole 1 drilling, and then return to Point R
Y-550; Positioning, Hole 2 drilling, and then return to Point R

Y-750;	Positioning, Hole 3 drilling, and then return to Point R
Y1000;	Positioning, Hole 4 drilling, and then return to Point R
Y-550;	Positioning, Hole 5 drilling, and then return to Point R
G98 Y-750;	Positioning, Hole 6 drilling, and then return to the initial position plane
G80;	
G28 G91 X0 Y0 Z0;	Return to reference point
M5;	The spindle stops rotating
M30;	

Restrictions: When the G81 command and the G code (G00 to G03, G60 are modal codes (when the position parameter NO: 48#0 is 1)) in Group 01 of the same program segment are used, the system will execute the final modal code.

Tool radius compensation: In this fixed cycle command, the tool radius compensation will be ignored because the command function does not require tool radius compensation.

4.4.3 Drilling Cycle and Boring Cycle (G82)

Format: G82 X_ Y_ Z_ R_ P_ F_ K_;

Function: This cycle is used for normal drilling; cutting feed to the bottom of the hole and pause execution, and then the tool will quickly retract from the bottom.

Description:

X_Y_: Hole positioning data;

Z_: Incremental programming defines the distance from Point R to the bottom of the hole; absolute programming defines the absolute coordinate values of the bottom of the hole;

R_: Incremental programming defines the distance from the initial point plane to Point R; absolute programming defines the absolute coordinate values of Point R;

F_: Cutting feed rate;

P_: Pause time in the bottom of the hole;

K_: Repetition times

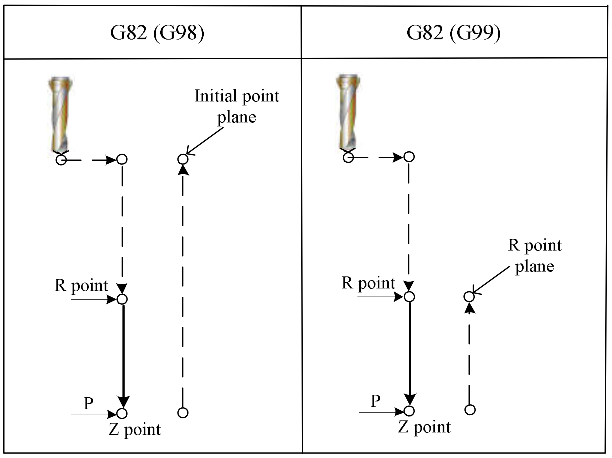


Fig. 4-4-3-1

After positioning along the X and Y axis, move quickly to Point R, and perform drilling from Point R to Point Z When it reaches the bottom of the hole, the pause will be executed and the tool will retract quickly.

Rotate the spindle with the auxiliary function M code before G82 is specified.

When G82 and the M code are specified by the same program segment, the M code is executed during positioning of the first hole, and then the system deals with the next drilling activity.

When repetition times K is specified, the M code is executed only for the first hole, and not for subsequent holes.

Note: In the current version, M00, M01, M02, M06, M30, M98, and M99 are post-program executed, and the above M code will be executed after the current segment is executed.

Tool length compensation: When the tool length compensation G43, G44 or G49 shares the same program segment with the fixed cycle command, add or cancel the offset when positioning to Point R; in the fixed cycle mode, when the tool compensation G43, G44 or G49 is alone in a program segment, the system can add or cancel the offset in real time.

P is a modal code; the minimum value of the parameter is set by Data Parameter **P296**, while the maximum value is set by **P297**. When the P value is less than that set by **P296**, it will run at the minimum value; when the P value is greater than that set by **P297**, it will run at the maximum value.

Example:

M3 S2000; the spindle starts to rotate

G90 G99 G82 X300 Y-250 Z-150 R-100 P1000 F120; Positioning, Hole 1 drilling, pause for 1 second at the bottom and then return to Point R

Y-550;	Positioning, Hole 2 drilling, pause for 1 second at the hole bottom and then return to Point R
Y-750;	Positioning, Hole 3 drilling, pause for 1 second at the bottom and then return to Point R
X1000.;	Positioning, Hole 4 drilling, pause for 1 second at the bottom and then return to Point R
Y-550;	Positioning, Hole 5 drilling, pause for 1 second at the bottom and then return to Point R
G98 Y-750;	Positioning, Hole 6 drilling, pause for 1 second at the bottom and then return to the initial position plane
G80;	Cancel the fixed cycle
G28 G91 X0 Y0 Z0;	Return to reference point
M5;	the spindle stops rotating
M30;	

Restrictions: When the G82 command and the G code (G00 to G03, G60 are modal codes (when the position parameter NO: 48#0 is 1)) in Group 01 of the same program segment are used, the system will execute the final modal code.

Tool radius compensation: In this fixed cycle command, the tool radius compensation will be ignored because the command function does not require tool radius compensation.

4.4.4 Chip removal drilling cycle (G83)

Format: G83 X_ Y_ Z_ R_ Q_ F_ K_

Function: This cycle is used for deep drilling, which performs intermittent cutting feed to the bottom of the hole, and chips are removed from the hole during drilling.

Description:

X_Y_: Hole positioning data;

Z_: Incremental programming defines the distance from Point R to the bottom of the hole; absolute programming defines the absolute coordinate values of the bottom of the hole;

R_: Incremental programming defines the distance from the initial point plane to Point R; absolute programming defines the absolute coordinate values of Point R;

Q_: Cutting depth of each cutting feed;

F_: Cutting feed rate;

K_: Repetition times

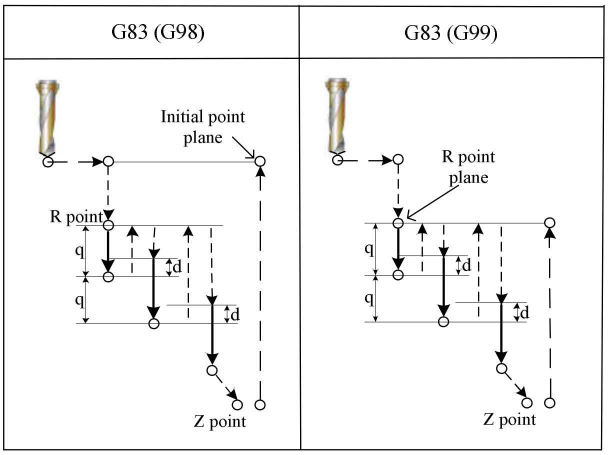


Fig. 4-4-4-1

Q: It is cutting depth for each cutting feed, which must be expressed in incremental value. In the second and subsequent cutting feeds, quickly move to the point with the distance of d before the end of the previous drilling, and then perform the cutting feed again, and the value of d is set by Parameter **P295**. As shown in the figure **4-4-4-1.**

A positive value must be specified in Q, and the negative sign will be ignored, and the system will still deal with it as a positive value.

Q is specified in the program segment in which drilling is performed, and Q will be stored as modal data if specified in the program segment in which drilling is not performed.

Rotate the spindle (M code) with the auxiliary function before G83 is specified.

When G83 and the M code are specified by the same program segment, the M code is executed during positioning of the first hole, and then the system deals with the next drilling activity.

When repetition times K is specified, the M code is executed only for the first hole, and not for subsequent holes.

Note 1: In the current version, M00, M01, M02, M06, M30, M98, and M99 are post-program executed, and the above M code will be executed after the current segment is executed.

Note 2: When the bit parameter NO: 43# 1=0, in case of deep drilling (G73, G83) without specified cutting depth, no alarm will be triggered, and in this case, the code parameter Q is not specified or Q is specified as 0, and the system will perform hole positioning in the X and Y planes, but not perform drilling. When the bit parameter NO: 43#1=1, in case of deep drilling (G73, G83) without specified cutting depth, an alarm will be triggered. That is, when the code parameter Q is not specified or Q is specified as 0, the system will provide the alarm: “0045: Address Q is not found or Q value is 0 (G73/G83)”. If Q is specified as a negative value, the system will perform intermittent feed with its absolute value.

Tool length compensation: When the tool length compensation G43, G44 or G49 shares the same program segment with the fixed cycle command, add or cancel the offset when positioning to Point R; in the fixed cycle mode, when the tool compensation G43, G44 or G49 is alone in a program segment, the system can add or cancel the offset in real time.

Example:

M3 S2000; The spindle starts rotating

G90 G99 G83 X300 Y-250 Z-150 R-100 Q15 F120; Positioning, Hole 1 drilling, and then return to Point R

Y-550; Positioning, Hole 2 drilling, and then return to Point R

Y-750; Positioning, Hole 3 drilling, and then return to Point R

X1000; Positioning, Hole 4 drilling, and then return to Point R

Y-550; Positioning, Hole 5 drilling, and then return to Point R

G98 Y-750; Positioning, Hole 6 drilling, and then return to the initial position plane
G80;
G28 G91 X0 Y0 Z0; Return to reference point
M5; The spindle stops rotating
M30;

Restrictions: When the G83 command and the G code (G00 to G03, G60 are modal codes (when the position parameter NO: 48#0 is 1)) in Group 01 of the same program segment are used, the system will execute the final modal code, i.e. G60.

Tool radius compensation: In this fixed cycle command, the tool radius compensation will be ignored because the command function does not require tool radius compensation.

4.4.5 Tapping Cycle (G74 Or G84)

Format: G74/G84 X_ Y_ Z_ R_ P_ F_K_

Function: This cycle performs tapping. In the tapping cycle, a pause will be performed when the tapping shaft reaches the bottom of the hole, and then the spindle reversely rotates back to the tapping shaft. (G74 is a left tapping cycle and G84 is a right tapping cycle)

Description:

X_Y_: Hole positioning data;

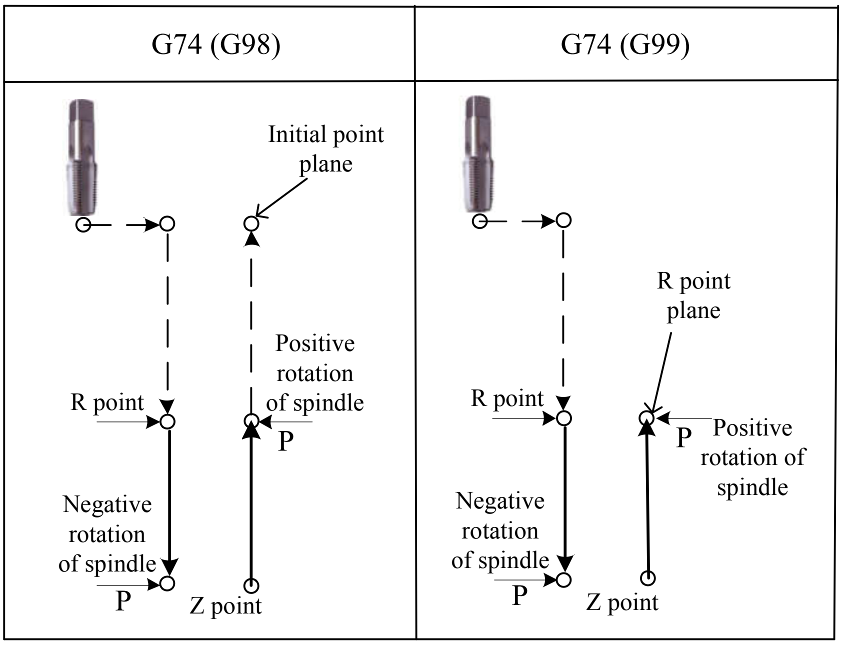
Z_: Incremental programming defines the distance from Point R to the bottom of the hole; absolute programming defines the absolute coordinate values of the bottom of the hole;

R_: Incremental programming defines the distance from the initial point plane to Point R; absolute programming defines the absolute coordinate values of Point R;

P_: Pause time in the bottom of the hole;

F_: Tapping feed rate;

K_: Repetition times; (specified when needed)



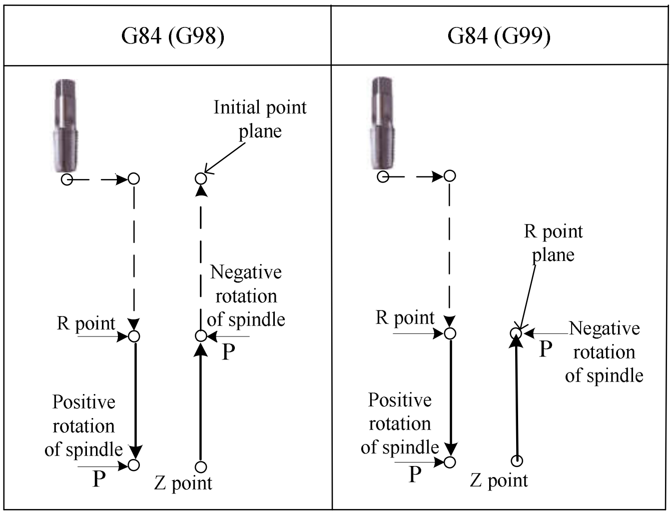


Fig. 4-4-5-1

In case of the G74 command, the spindle rotates clockwise (while in case of the G84 command, the spindle rotates counterclockwise) to perform tapping. A pause will be executed when it reaches the bottom of the hole. And the spindle will rotate reversely and retract to the tapping shaft at the specified feed rate. Threads will be formed in this process.

Example:

G94	Feed mode per minute;
M29 S1000;	The spindle stops rotating and its rotation speed is specified
G43 / G44 H10;	Call the tool length compensation
G90 G99 G74 / G84 X100 Y110 Z -50 R5 P3000 F100;	Positioning, Hole 1 tapping, then return to Point R
Y150;	Positioning, Hole 2 tapping, then return to Point R
G91 X50 K5;	Use X100, Y150 as the reference point, along the X axis Perform 5 times of tapping with an increment of 50mm
G98 Y-750;	Positioning, Hole 8 tapping, then return to the initial point
G80;	Cancel the tapping cycle
G28 G91 X0 Y0 Z0;	Return to reference point
M30;	Program ends

Tool length compensation: When the tool length compensation G43, G44 or G49 shares the same program segment with the fixed cycle command, add or cancel the offset when positioning to Point R; in the fixed cycle mode, when the tool compensation G43, G44 or G49 is alone in a program segment, the system can add or cancel the offset in real time.

Thread lead: In the feed mode per minute, the relationship between the thread lead and the feed rate and the spindle speed:

Feed rate F = screw tap thread pitch × spindle speed S

For example: For tapping of M12×1.5 threaded holes on a component, the following parameters can be used:

S500=500 r/min; F=1.5×500=750 mm/min;

In case of multiple thread, multiply by the number of heads to get the F value.

In the feed mode per rotation, the thread lead is equal to the feed rate.

For example:

Feed mode per minute:		Feed mode per rotation:	
Spindle speed 1000 r/min;		Spindle speed 1000 r/min;	
Thread lead 1.0 mm;		Thread lead 1.0 mm;	
Then Z-axis feed rate = 1000*1 = 1000 mm/min;		Then Z-axis feed rate = thread lead = 1mm/r;	
G94	feed mode per minute	G95	feed mode per rotation
G00 X120 Y100;	Positioning	G00 X120 Y100;	Positioning
M29 S1000;	Specify rigid mode specified	M29 S1000;	Specify rigid mode
G84 Z-100 R-20 F1000;	Right rigid tapping	G84 Z-100 R-20 F1;	Right rigid tapping
G80	Cancel the tapping cycle	G80	Cancel the tapping cycle

G28 G91 X0 Y0 Z0 Return to reference point G28 G91 X0 Y0 Z0 Return to reference point
M30 Program ends M30 Program ends

Restrictions:

G code: When the G74/G84 command and the G code (G00 to G03, G60 are modal codes (when the position parameter NO: 48#0 is 1)) in Group 01 of the same program segment are used, the system will execute the final modal code.

M code: Rotate the spindle with the auxiliary function M code before G74/G84 is specified. If the spindle rotates without command, the system automatically will adjust to clockwise rotation (G74)/counterclockwise rotation (G84) based on the current spindle command speed in the R plane.

When G74/G84 and the M code are specified by the same program segment, the M code is executed during positioning of the first hole, and then the system deals with the next tapping activity.

When repetition times K is specified, the M code is executed only for the first hole, and not for subsequent holes.

Note: In the current version, M00, M01, M02, M06, M30, M98, and M99 are post-program executed, and the above M code will be executed after the current segment is executed.

S command: If the specified spindle speed exceeds the maximum spindle speed during tapping (Data Parameter P257: spindle upper limit speed during tapping cycle), the system will send an alarm; the maximum spindle speed during rigid tapping is set by Data Parameter P294 - P296.

F command: If the specified F value exceeds the upper limit of the cutting feed rate (data parameter: the upper limit is set by P96), the upper limit shall prevail.

P command: P is a modal code; the minimum value of the parameter is set by Data Parameter **P296**, while the maximum value is set by **P297**. When the P value is less than that set by **P296**, it will run at the minimum value; when the P value is greater than that set by **P297**, it will run at the maximum value.

Shaft switch: The fixed cycle must be canceled before switching the tapping shaft. If the tapping shaft is changed in the rigid mode, the system will send No. 206 alarm.

Override: During tapping, the feedrate override and spindle speed override are 100% by default, and the machine tool will not stop working when the hold button is pressed until the return activity is completed.

Tool radius compensation: In this fixed cycle command, the tool radius compensation will be ignored because the command function does not require tool radius compensation.

Program restart: In the tapping cycle, the program restart function is invalid.

4.4.6 Precision Boring Cycle (G76)

Format: G76 X_Y_Z_Q_R_P_F_K_

Function: This cycle is applied to precision boring of holes.

When the bottom of the hole is reached, the spindle will stop rotating and the cutting tool will leave the workpiece surface for return.

Retraction marks should be prevented to avoid impact on smooth finish of the machined surface and damages to the tool.

Description:

X_Y_: Hole positioning data;

Z_: Incremental programming defines the distance from Point R to the bottom of the hole; absolute programming defines the absolute coordinate values of the bottom of the hole;

R_: Incremental programming defines the distance from the initial point plane to Point R; absolute programming defines the absolute coordinate values of Point R;

Q_: Offset of the bottom of the hole;

P_: Pause time in the bottom of the hole;

F_: Cutting feed rate;

K_: Number of times of precision boring

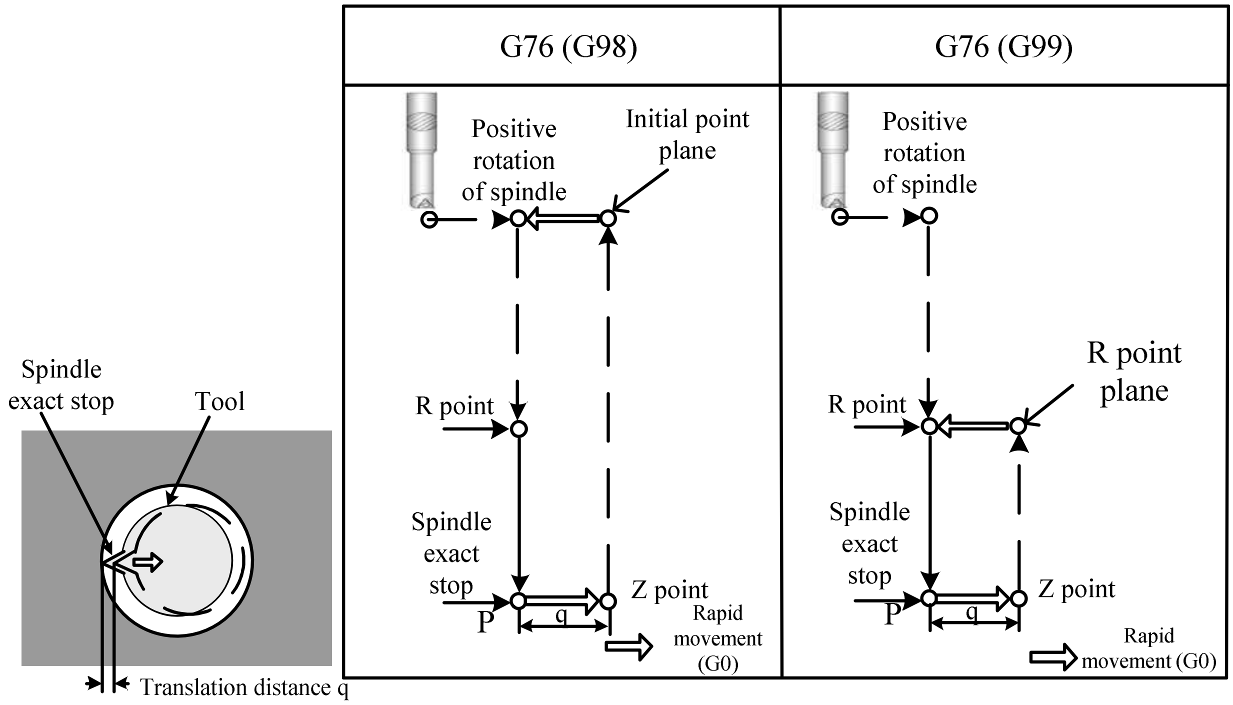


Fig. 4-4-6-1

When the tool reaches the bottom of the hole, the spindle will stop at a fixed rotation position and the tool will retract in the opposite direction of the tool tip. This ensures that the machined surface is not damaged to achieve precise and efficient boring. The retraction distance is specified by parameter Q. The retraction direction and retraction axis are specified by Bit Parameters N0:42#4 and N0:42#5. And Q must be a positive value. Even if a negative value is used, the negative sign does not work. The offset of Q at the bottom of the hole is the modal value that is stored in the fixed cycle and must be specified carefully. The reason is that it is also used as the cutting depth of G73 and G83.

Rotate the spindle with the auxiliary function M code before G76 is specified.

When G76 and the M code are specified by the same program segment, the M code is executed during positioning of the first hole, and then the system deals with the next activity. When repetition times K is specified, the M code is executed only for the first hole, and not for subsequent holes.

Note: In the current version, M00, M01, M02, M06, M30, M98, and M99 are post-program executed, and the above M code will be executed after the current segment is executed.

Tool length compensation: When the tool length compensation G43, G44 or G49 shares the same program segment with the fixed cycle command, add or cancel the offset when positioning to Point R; in the fixed cycle mode, when the tool compensation G43, G44 or G49 is alone in a program segment, the system can add or cancel the offset in real time.

Shaft switch: The fixed cycle must be canceled before the drilling shaft is changed.

Boring: No boring is performed in program segments without X, Y, Z or other axis.

Example:

M3 S500; the spindle starts rotating

G90 G99 G76 X300 Y-250 Z-150 R-100 Q5 P1000 F120; Positioning, Hole 1 boring, then return to Point R, orientation in the bottom of the hole, movement for 5 mm and pause at the bottom for 1s

Y-550; Positioning,	Hole 2 boring, and then return to Point R
Y-750; Positioning,	Hole 3 boring, and then return to Point R
X1000; Positioning,	Hole 4 boring, and then return to Point R
Y-550; Positioning,	Hole 5 boring, and then return to Point R
G98 Y-750; Positioning,	Hole 6 boring, and then return to the initial position plane
G80 G28 G91 X0 Y0 Z0;	Return to reference point
M5;	The spindle stops rotating

Volume I Programming instructions

Restrictions: When the G76 command and the G code (G00 to G03, G60 are modal codes (when the position parameter NO: 48#0 is 1)) in Group 01 of the same program segment are used, the system will execute the final modal code.

Tool radius compensation: In this fixed cycle command, the tool radius compensation will be ignored because the command function does not require tool radius compensation.

Note: In this command, the feed shaft and the feed direction are fixed, and the rotation of the G68 coordinate system has no impact on the feed direction.

4.4.7 Boring Cycle (G85)

Format: G85 X_ Y_ Z_ R_ F_ K_

Function: This cycle is used for boring.

Description:

X_Y_: Hole positioning data;

Z_: Incremental programming defines the distance from Point R to the bottom of the hole; absolute programming defines the absolute coordinate values of the bottom of the hole;

R_: Incremental programming defines the distance from the initial point plane to Point R; absolute programming defines the absolute coordinate values of Point R;

F_: Cutting feed rate;

K_: Repetition times

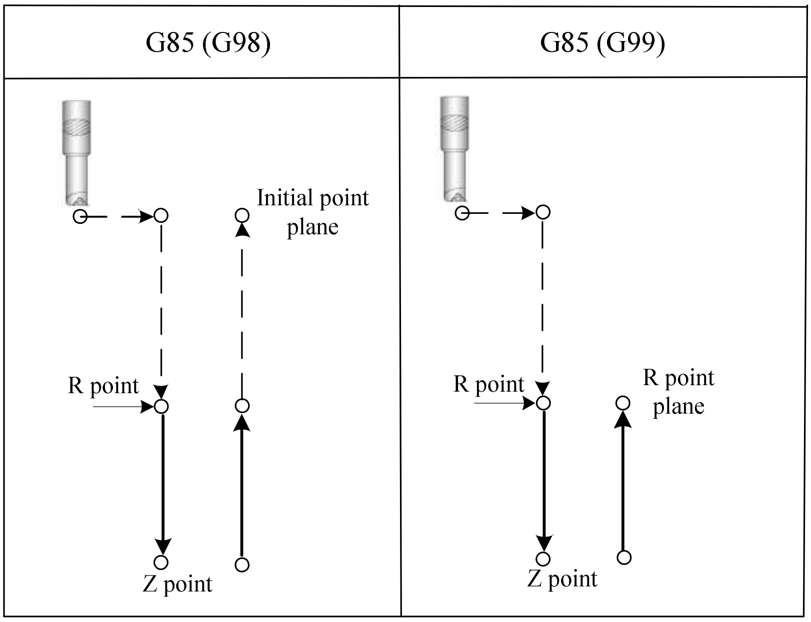


Fig. 4-4-7-1

After positioning along the X and Y axis, move quickly to Point R, then perform boring from Point R to Point Z. When the bottom of the hole is reached, perform cutting feed and then return to Point R.

Rotate the spindle with the auxiliary function M code before G85 is specified.

When G85 and the M code are specified by the same program segment, the M code is executed during positioning of the first hole, and then the system deals with the next activity.

When repetition times K is specified, the M code is executed only for the first hole, and not for subsequent holes.

Note: In the current version, M00, M01, M02, M06, M30, M98, and M99 are post-program executed, and the above M code will be executed after the current segment is executed.

Tool length compensation: When the tool length compensation G43, G44 or G49 shares the same program segment with the fixed cycle command, add or cancel the offset when positioning to Point R; in the fixed cycle mode, when the tool compensation G43, G44 or G49 is alone in a program segment, the system can add or cancel the offset in real time.

Shaft switch: The fixed cycle must be canceled before the drilling shaft is changed.

Boring: No boring is performed in program segments without X, Y, Z or other axis.

Example:

M3 S100; The spindle starts rotating

G90 G99 G85 X300 Y-250 Z-150 R-120 F120; positioning, Hole 1 boring, and then return to Point R

Y-550; Positioning, Hole 2 boring, and then return to Point R

Y-750; Positioning, Hole 3 boring, and then return to Point R

X1000; Positioning, Hole 4 boring, and then return to Point R

Y-550; Positioning, Hole 5 boring, and then return to Point R

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

G80;

G28 G91 X0 Y0 Z0; Return to reference point

M5; The spindle stops rotating

M30;

Restrictions: When the G85 command and the G code (G00 to G03, G60 are modal codes (when the position parameter NO: 48#0 is 1) in Group 01 of the same program segment are used, the system will execute the final modal code.

Tool radius compensation: In this fixed cycle command, the tool radius compensation will be ignored because the command function does not require tool radius compensation.

4.4.8 Boring Cycle (G86)

Format: G86 X_ Y_ Z_ R_ F_ K_;

Function: This cycle code is used for boring cycle. (No pause is required at the bottom of the hole)

Description:

X_Y_: Hole positioning data;

Z_: Incremental programming defines the distance from Point R to the bottom of the hole; absolute programming defines the absolute coordinate values of the bottom of the hole;

R_: Incremental programming defines the distance from the initial point plane to Point R; absolute programming defines the absolute coordinate values of Point R;

F_: Cutting feed rate;

K_: Repetition times

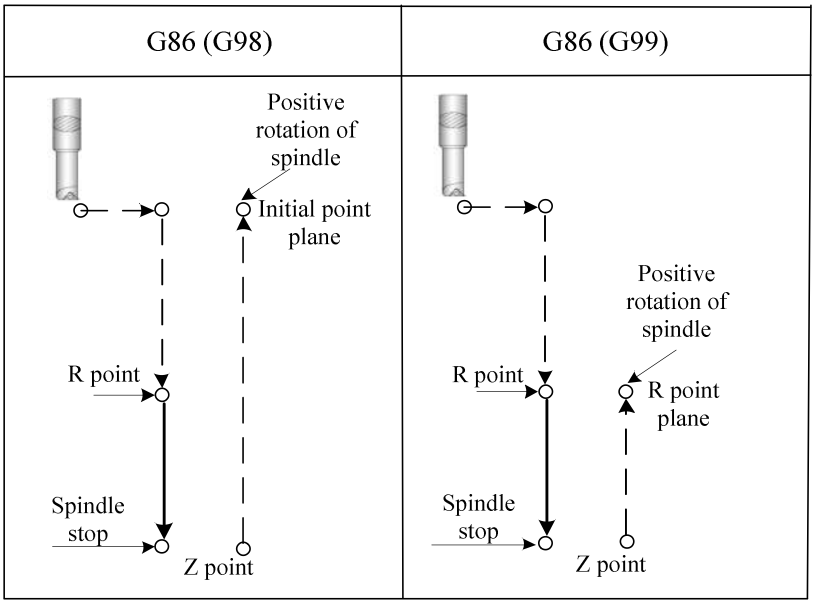


Fig. 4-4-8-1

After positioning along the X and Y axis, move quickly to Point R, and then perform boring from Point R to Point Z. When the spindle stops at the bottom of the hole, the tool retracts with rapid movement.

Rotate the spindle with the auxiliary function M code before G86 is specified.
When G86 and the M code are specified by the same program segment, the M code is executed during positioning of the first hole, and then the system deals with the next activity. When repetition times K is specified, the M code is executed only for the first hole, and not for subsequent holes.

Note: In the current version, M00, M01, M02, M06, M30, M98, and M99 are post-program executed, and the above M code will be executed after the current segment is executed.

Tool length compensation: When the tool length compensation G43, G44 or G49 shares the same program segment with the fixed cycle command, add or cancel the offset when positioning to Point R; in the fixed cycle mode, when the tool compensation G43, G44 or G49 is alone in a program segment, the system can add or cancel the offset in real time.
Shaft switch: The fixed cycle must be canceled before the drilling shaft is changed.
Boring: No boring is performed in program segments without X, Y, Z or other axis.

Example:

```
M3 S2000;                    The spindle starts rotating
G90 G99 G86 X300 Y-250 Z-150 R-100 F120; Positioning, Hole 1 boring, and then return to Point R
Y-550;                       Positioning, Hole 2 boring, and then return to Point R
Y-750;                       Positioning, Hole 3 boring, and then return to Point R
X1000;                       Positioning, Hole 4 boring, and then return to Point R
Y-550;                       Positioning, Hole 5 boring, and then return to Point R
G98 Y-750;                   Positioning, Hole 6 boring, and then return to the initial position plane
G80;
G28 G91 X0 Y0 Z0;            Return to reference point
M5;                          The spindle stops rotating
M30;
```

Restrictions: When the G86 command and the G code (G00 to G03, G60 are modal codes (when the position parameter NO: 48#0 is 1)) in Group 01 of the same program segment are used, the system will execute the final modal code.

Tool radius compensation: In this fixed cycle command, the tool radius compensation will be ignored because the command function does not require tool radius compensation.

4.4.9 Boring Cycle and Back Boring Cycle (G87)

Format: G87 X_Y_Z_R_Q_P_ F_;

Function: This cycle performs precision boring

Description:

X_Y_: Hole positioning data;
Z_: Incremental programming defines the distance from the R point to Pint Z; absolute programming defines the absolute coordinate values of Point Z;
R_: Incremental programming defines the distance from the initial point plane to Point R; absolute programming defines the absolute coordinate values of Point R; (the bottom of the hole)
Q_: Offset of the bottom of the hole;
P_: Pause time in the bottom of the hole;
F_: Cutting feed rate;

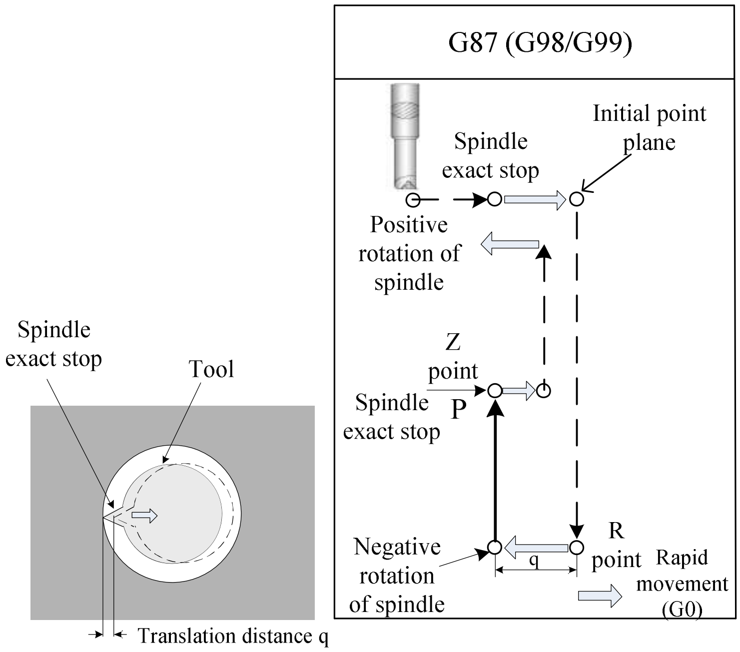


Fig. 4-4-9-1

After positioning along the X and Y axis, the spindle stops the tool when oriented, moves in the opposite direction of the tool tip, and moves at the feed speed at Point R at the bottom of the hole. Then the tool moves in the direction of the tool tip while the spindle rotates reversely along the positive boring on the Z axis to Point Z. When oriented again at Point Z, the spindle stops at a fixed rotational position and the tool retracts in the opposite direction of the tool tip and returns to the initial plane. The tool shifts from the spindle and rotates forward in the direction of the tool tip and performs the machining in the next program segment.

The retraction distance is specified by Parameter Q. The retraction direction and retraction axis are specified by System Parameters **N0:42#4** and **N0:42#5**. And Q must be a positive value. Even if a negative value is used, the negative sign does not work. The offset of Q at the bottom of the hole is a modal value that is stored in the fixed cycle and must be specified carefully, because it is also used as the cutting depth for G73 and G83.

Rotate the spindle with the auxiliary function M code before G87 is specified.

When G87 and the M code are specified by the same program segment, the M code is executed during positioning of the first hole, and then the system deals with the next drilling activity.

When repetition times K is specified, the M code is executed only for the first hole, and not for subsequent holes.

Note: In the current version, M00, M01, M02, M06, M30, M98, and M99 are post-program executed, and the above M code will be executed after the current segment is executed.

Tool length compensation: When the tool length compensation G43, G44 or G49 shares the same program segment with the fixed cycle command, add or cancel the offset when positioning to Point R; in the fixed cycle mode, when the tool compensation G43, G44 or G49 is alone in a program segment, the system can add or cancel the offset in real time.

The fixed cycle can only be performed on the G17 plane.

Boring: No boring is performed in program segments without X, Y, Z or other auxiliary axis.

Hint: When programming the back boring cycle, remember that the Z value and the R value should be specified. In general, the Z position here is above the R position. Otherwise, the system will give an alarm.

Example:

M3 S500; the spindle starts rotating

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

(Positioning, Hole 1 boring, oriented at the initial position and then offset by 5mm and pause at Point Z for 1 second)

Y-550; Positioning, Hole 2 boring, and then return to the initial position plane

Volume I Programming instructions

Y-750; Positioning, Hole 3 boring, and then return to the initial position plane
X1000; Positioning, Hole 4 boring, and then return to the initial position plane
Y-550; Positioning, Hole 5 boring, and then return to the initial position plane
G98 Y-750; Positioning, Hole 6 boring, and then return to the initial position plane
G80 G28 G91 X0 Y0 Z0; Return to reference point
M5; The spindle stops rotating
M30;

Restrictions: When the G87 command and the G code (G00 to G03, G60 are modal codes (when the position parameter NO: 48#0 is 1)) in Group 01 of the same program segment are used, the system will execute the final modal code.

Tool radius compensation: In this fixed cycle command, the tool radius compensation will be ignored because the command function does not require tool radius compensation.

Note: In this command, the feed shaft and the feed direction are fixed, and the rotation of the G68 coordinate system has no impact on the feed direction.

4.4.10 Boring Cycle (G88)

Format: G88 X_Y_Z_R_ P_F_

Function: This cycle is used for boring.

Description:

X_Y_: Hole positioning data;
Z_: Incremental programming defines the distance from Point R to the bottom of the hole; absolute programming defines the absolute coordinate values of the bottom of the hole;
R_: Incremental programming defines the distance from the initial point plane to Point R; absolute programming defines the absolute coordinate values of Point R;
P_: Pause time in the bottom of the hole;
F_: Cutting feed rate.

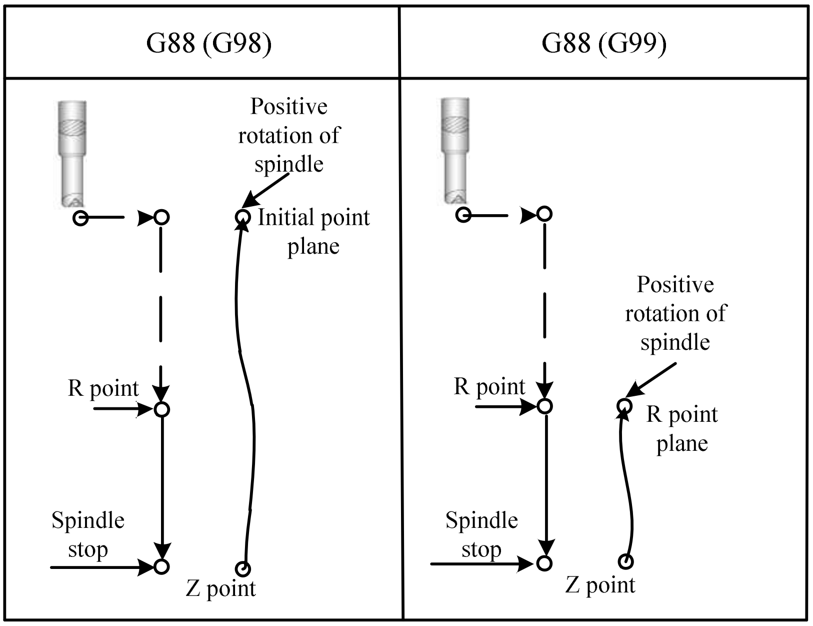


Fig. 4-4-10-1

After positioning along the X and Y axis, quickly move to Point R, and then perform boring from Point R to Point Z. When the boring is completed, the pause is executed, and then the spindle stops, and the tool is manually returned from Point Z point at the bottom to Point R (in case of G99) or the initial point (in case of G98), and the spindle rotates forward.

Rotate the spindle with the auxiliary function M code before G88 is specified.

When G88 and the M code are specified by the same program segment, the M code is executed during positioning of the first hole, and then the system deals with the next drilling activity.

When repetition times K is specified, the M code is executed only for the first hole, and not for subsequent holes.

Note: In the current version, M00, M01, M02, M06, M30, M98, and M99 are post-program executed, and the above M code will be executed after the current segment is executed.

P is a modal code; the minimum value of the parameter is set by Data Parameter P296, while the maximum value is set by P297. When the P value is less than that set by P296, it will run at the minimum value; when the P value is greater than that set by P297, it will run at the maximum value.

Tool length compensation: When the tool length compensation G43, G44 or G49 shares the same program segment with the fixed cycle command, add or cancel the offset when positioning to Point R; in the fixed cycle mode, when the tool compensation G43, G44 or G49 is alone in a program segment, the system can add or cancel the offset in real time.

Shaft switch: The fixed cycle must be canceled before switching the boring shaft.

Boring: No boring is performed in program segments without X, Y, Z or other auxiliary axis.

Example:

M3 S2000;　　Spindle starts rotating

G90 G99 G88 X300 Y-250 Z-150 R-100 P1000 F120;　　Positioning, Hole 1 boring, and then return to Point R

Y-550;　　Positioning, Hole 2 boring, and then return to Point R

Y-750;　　Positioning, Hole 3 boring, and then return to Point R

X1000;　　Positioning, Hole 4 boring, and then return to Point R

Y-550;　　Positioning, Hole 5 boring, and then return to Point R

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

G80 G28 G91 X0 Y0 Z0; Return to reference point

M5;　　The spindle stops rotating

Restrictions: When the G88 command and the G code (G00 to G03, G60 are modal codes (when the position parameter NO: 48#0 is 1)) in Group 01 of the same program segment are used, the system will execute the final modal code.

Tool radius compensation: In this fixed cycle command, the tool radius compensation will be ignored because the command function does not require tool radius compensation.

4.4.11 Boring Cycle (G89)

Format: G89 X_ Y_ Z_ R_ P_ F_ K_

Function: This cycle is used for boring.

Description:

X_Y_: Hole positioning data;

Z_: Incremental programming defines the distance from Point R to the bottom of the hole; absolute programming defines the absolute coordinate values of the bottom of the hole;

R_: Incremental programming defines the distance from the initial point plane to Point R; absolute programming defines the absolute coordinate values of Point R;

P_: Pause time in the bottom of the hole;

F_: Cutting feed rate;

K_: Repetition times

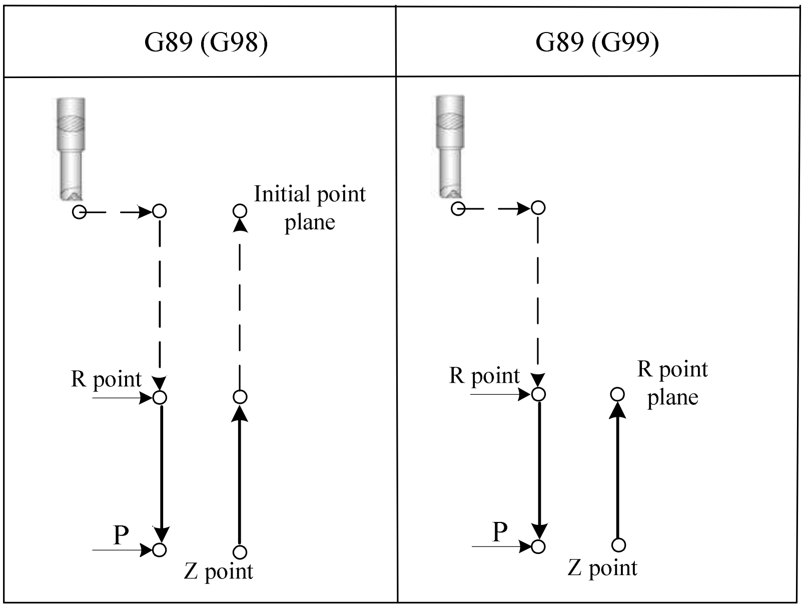


Fig. 4-4-11-1

This cycle is almost identical to G85, except that this cycle performs a pause at the bottom of the hole.

Rotate the spindle with the auxiliary function M code before G89 is specified.

When G89 and the M code are specified by the same program segment, the M code is executed during positioning of the first hole, and then the system deals with the next drilling activity.

When repetition times K is specified, the M code is executed only for the first hole, and not for subsequent holes.

Note: In the current version, M00, M01, M02, M06, M30, M98, and M99 are post-program executed, and the above M code will be executed after the current segment is executed.

P is a modal code; the minimum value of the parameter is set by Data Parameter P296, while the maximum value is set by P297. When the P value is less than that set by P296, it will run at the minimum value; when the P value is greater than that set by P297, it will run at the maximum value.

Tool length compensation: When the tool length compensation G43, G44 or G49 shares the same program segment with the fixed cycle command, add or cancel the offset when positioning to Point R; in the fixed cycle mode, when the tool compensation G43, G44 or G49 is alone in a program segment, the system can add or cancel the offset in real time.

Shaft switch: The fixed cycle must be canceled before switching the boring shaft.

Boring: No boring is performed in program segments without X, Y, Z or other auxiliary axis.

Example:

```
M3 S100;                     The spindle starts rotating
G90 G99 G89 X300 Y-250 Z-150 R-120 P1000 F120; positioning, Hole 1 boring, then return to
                                               Point R and pause at the bottom of the
                                               hole for 1 second
Y-550;                       Positioning, Hole 2 boring, and then return to Point R
Y-750;                       Positioning, Hole 3 boring, and then return to Point R
X1000;                       Positioning, Hole 4 boring, and then return to Point R
Y-550;                       Positioning, Hole 5 boring, and then return to Point R
G98 Y-750;                   Positioning, Hole 6 boring, and then return to the initial position plane
G80;
G28 G91 X0 Y0 Z0;            Return to reference point
```

M5;   The spindle stops rotating
M30;

Restrictions: When the G89 command and the G code (G00 to G03, G60 are modal codes (when the position parameter NO: 48#0 is 1)) in Group 01 of the same program segment are used, the system will execute the final modal code.

Tool radius compensation: In this fixed cycle command, the tool radius compensation will be ignored because the command function does not require tool radius compensation.

4.5 Rigid Cycle (G Code)

4.5.1 Left Rigid Tapping (G74)

Format: G74 X_Y_Z_R_P_F_K_

Function: In rigid mode, the spindle motor works as a servo motor, and this code can realize left high-speed high-precision tapping.

Description:

X_Y_: Hole positioning data

Z_: Incremental programming defines the distance from Point R to the bottom of the hole; absolute programming defines the absolute coordinate values of the bottom of the hole;

R_: Incremental programming defines the distance from the initial point plane to Point R; absolute programming defines the absolute coordinate values of Point R;

P_: Pause time at the bottom of the hole or pause time at Point R during backing.

F_: Cutting feed rate.

K_: Repetition times (It should be specified when needed)

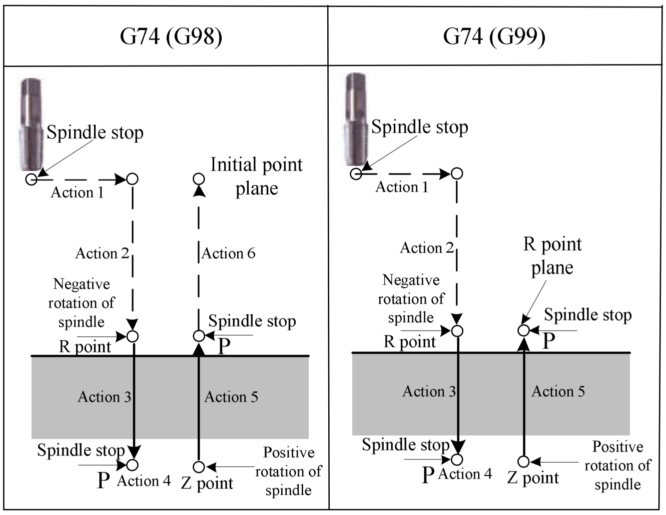


Fig. 4-5-1-1

After positioning along the X and Y axis, quickly move from the Z axis to Point R, and execute G74 to make the spindle rotate reversely. The tapping is performed from Point R to Point Z. When the tapping is completed, the spindle stops and a pause is executed. And then the spindle rotates reversely and the tool returns to Point R. When the spindle stops, perform a quick move to the initial position. The feedrate override and the spindle override are considered to be 100% when the tapping is being performed.

Rigid mode: In the position mode (position parameter NO: 46#1 set to 1, K parameter NO: 7#7 set to 1), the rigid mode can be specified when M29 S***** is specified before the tapping code.

Tool length compensation: When the tool length compensation G43, G44 or G49 shares the same program segment with the fixed cycle command, add or cancel the offset when positioning to Point R; in the fixed cycle mode, when the tool compensation G43, G44 or G49 is alone in a program segment, the system can add or cancel the offset in real time.

Thread lead: In the feed mode per minute, the relationship between the thread lead and the feed rate and the spindle speed:

Feed rate F = screw tap thread pitch × spindle speed S

For example: For tapping of M12×1.5 threaded holes on a component, the following parameters can be used:

S500=500 r/min; F=1.5×500=750 mm/min;

In case of multiple thread, multiply by the number of heads to get the F value.

In the feed mode per rotation, the thread lead is equal to the feed rate.

For example:

Feed mode per minute:		Feed mode per rotation:	
Spindle speed 1000 r/min;		Spindle speed 1000 r/min;	
Thread lead 1.0 mm;		Thread lead 1.0 mm;	
Then Z-axis feed rate = 1000*1 = 1000 mm/min;		Then Z-axis feed rate = thread lead 1 = 1mm/r;	
G94	feed mode per minute	G95	Feed mode per rotation
G00 X120 Y100;	Positioning	G00 X120 Y100;	Positioning
M29 S1000;	Specify rigid mode	M29 S1000;	Specify rigid mode
G74 Z-100 R-20 F1000;	left rigid tapping	G74 Z-100 R-20 F1;	Left rigid tapping
G80	Cancel the tapping cycle	G80	Cancel the tapping cycle
G28 G91 X0 Y0 Z0	Return to reference point	G28 G91 X0 Y0 Z0	Return to reference point
M30	Program ends	M30	Program ends

Restrictions:

G code: When the G74 command and the G code (G00 to G03, G60 are modal codes (when the position parameter NO: 48#0 is 1)) in Group 01 of the same program segment are used, the system will execute the final modal code.

M code: Rotate the spindle with the auxiliary function M code before G74 is specified. If the spindle rotates without command, the system will automatically adjust to clockwise rotation based on the current spindle command speed in the R plane.

When G74 and the M code are specified by the same program segment, the M code is executed during positioning of the first hole, and then the system deals with the next drilling activity.

When repetition times K is specified, the M code is executed only for the first hole, and not for subsequent holes.

Note: In the current version, M00, M01, M02, M06, M30, M98, and M99 are post-program executed, and the above M code will be executed after the current segment is executed.

S command: If the specified spindle speed exceeds the maximum spindle speed during tapping (Data Parameter P257: spindle upper limit speed during tapping cycle), the system will send an alarm; the maximum spindle speed during rigid tapping is set by Data Parameter P294 - P296.

F command: If the specified F value exceeds the upper limit of the cutting feed rate (data parameter: P96 sets the upper limit of cutting feed), the upper limit shall prevail.

P command: P is a modal code; the minimum value of the parameter is set by Data Parameter **P296**, while the maximum value is set by **P297**. When the P value is less than that set by **P296**, it will run at the minimum value; when the P value is greater than that set by **P297**, it will run at the maximum value.

Shaft switch: The fixed cycle must be canceled before switching the tapping shaft. If the tapping shaft is changed in the rigid mode, the system will send No. 206 alarm.

Override: During rigid tapping, the feedrate override and spindle speed override are 100% by default, and the machine tool will not stop working when the feed hold button and reset button are pressed until the return activity is completed.

Tool radius compensation: In this fixed cycle command, the tool radius compensation will be ignored because the command function does not require tool radius compensation.

Program restart: In the tapping cycle, the program restart function is invalid.

Note: In the process of soft tapping, rigid tapping or deep-hole rigid tapping, it is necessary to cancel the constant surface cutting speed with G97 first, or otherwise there will be incorrect thread or broken tap.

4.5.2 Right Rigid Tapping (G84)

Format: G84 X_Y_Z_R_P_F_K_

Function: In rigid mode, the spindle motor works as a servo motor and can realize high-speed high-precision tapping. It can be guaranteed that the starting position of tapping is consistent without changing Point R. That is, the tapping is repeatedly performed in one position, and the thread will not be buckled in a mess or rotted.

Description:

X_Y_: Hole positioning data;

Z_: Incremental programming defines the distance from Point R to the bottom of the hole; absolute programming defines the absolute coordinate values of the bottom of the hole;

R_: Incremental programming defines the distance from the initial point plane to Point R; absolute programming defines the absolute coordinate values of Point R;

P_: Pause time at the bottom of the hole or pause time at Point R during backing;

F_: Cutting feed rate;

K_: Repetition times (It should be specified when needed)

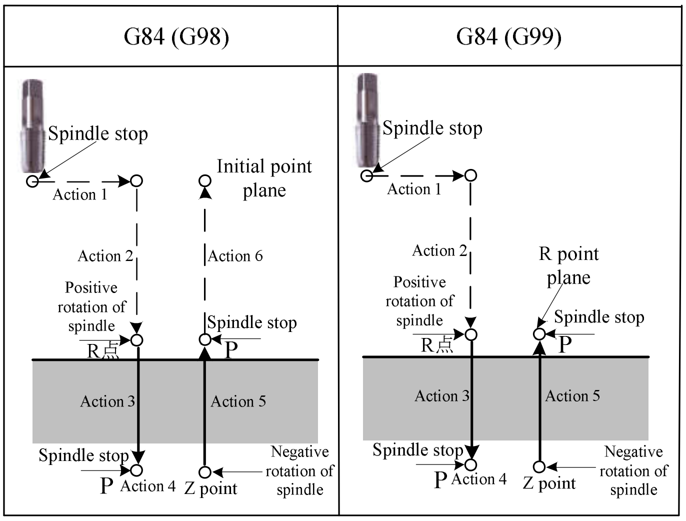


Fig. 4-5-2-1

After positioning along the X and Y axis, quickly move from the Z axis to Point R, and execute G84 to make the spindle rotate reversely. The tapping is performed from Point R to Point Z. When the tapping is completed, the spindle stops and a pause is executed. And then the spindle rotates reversely and the tool returns to Point R, and the spindle stops. Then perform a quick move to the initial position. The feedrate override and the spindle override are considered to be 100% when the tapping is being performed.

Rigid mode: In the position mode (position parameter NO: 46#1 set to 1, K parameter NO: 7#7 set to 1), the rigid mode can be specified when M29 S***** is specified before the tapping code.

Tool length compensation: When the tool length compensation G43, G44 or G49 shares the same program segment with the fixed cycle command, add or cancel the offset when positioning to Point R; in the fixed cycle mode, when the tool compensation G43, G44 or G49 is alone in a program segment, the system can add or cancel the offset in real time.

Thread lead: In the feed mode per minute, the relationship between the thread lead and the feed rate and the spindle speed:

Feed rate F = screw tap thread pitch × spindle speed S

For example: For tapping of M12×1.5 threaded holes on a component, the following parameters can be used:

S500=500 r/min; F=1.5×500=750 mm/min;

In case of multiple thread, multiply by the number of heads to get the F value.

In the feed mode per rotation, the thread lead is equal to the feed rate.

For example: Feed mode per minute: Feed mode per rotation:

Spindle speed 1000 r/min;		Spindle speed 1000 r/min;	
Thread lead 1.0 mm;		Thread lead 1.0 mm;	
Then Z-axis feed rate = 1000*1 = 1000 mm/min;		Then Z-axis feed rate = thread lead 1 = 1mm/r;	
G94	feed mode per minute	G95	feed mode per rotation
G00 X120 Y100;	Positioning	G00 X120 Y100;	Positioning
M29 S1000;	Specify rigid mode	M29 S1000;	Specify rigid mode
G84 Z-100 R-20 F1000;	Right rigid tapping	G84 Z-100 R-20 F1;	Right rigid tapping
G80	Cancel the tapping cycle	G80	Cancel the tapping cycle
G28 G91 X0 Y0 Z0	Return to reference point	G28 G91 X0 Y0 Z0	Return to reference point
M30	Program ends	M30	Program ends

Restrictions:

G code: When the G84 command and the G code (G00 to G03, G60 are modal codes (when the position parameter NO: 48#0 is 1)) in Group 01 of the same program segment are used, the system will execute the final modal code.

M code: Rotate the spindle with the auxiliary function M code before G84 is specified. If the spindle rotates without command, the system will automatically adjust to anti-clockwise rotation based on the current spindle command speed in the R plane.

When G84 and the M code are specified by the same program segment, the M code is executed during positioning of the first hole, and then the system deals with the next drilling activity.

When repetition times K is specified, the M code is executed only for the first hole, and not for subsequent holes.

Note: In the current version, M00, M01, M02, M06, M30, M98, and M99 are post-program executed, and the above M code will be executed after the current segment is executed.

S command: If the specified spindle speed exceeds the maximum spindle speed during tapping (Data Parameter P257: spindle upper limit speed during tapping cycle), the system will send an alarm; the maximum spindle speed during rigid tapping is set by Data Parameter P294 - P296.

F command: If the specified F value exceeds the upper limit of the cutting feed rate (data parameter: the upper limit is set by P96), the upper limit shall prevail.

P command: P is a modal code; the minimum value of the parameter is set by Data Parameter **P296**, while the maximum value is set by **P297**. When the P value is less than that set by **P296**, it will run at the minimum value; when the P value is greater than that set by **P297**, it will run at the maximum value.

Shaft switch: The fixed cycle must be canceled before switching the tapping shaft. If the tapping shaft is changed in the rigid mode, the system will send No. 206 alarm.

Override: During rigid tapping, the feedrate override and spindle speed override are 100% by default, and the machine tool will not stop working when the feed hold button and reset button are pressed until the return activity is completed.

Tool radius compensation: In this fixed cycle command, the tool radius compensation will be ignored because the command function does not require tool radius compensation.

Program restart: In the tapping cycle, the program restart function is invalid.

Note: In the process of soft tapping, rigid tapping or deep-hole rigid tapping, it is necessary to cancel the constant surface cutting speed with G97 first, or otherwise there will be incorrect thread or broken tap.

4.5.3 Deep-Hole Tapping (Chip Removal) Cycle

Format: G84（or G74） X_Y_Z_R_P_Q_F_K_

Function: In deep-hole tapping, several advances of the tool are performed until the bottom of the hole is reached.

Description:

X_Y_: Hole positioning data;

Z_: Incremental programming defines the distance from Point R to the bottom of the hole; absolute programming defines the absolute coordinate values of the bottom of the hole;

R_: Incremental programming defines the distance from the initial point plane to Point R; absolute programming defines the absolute coordinate values of Point R;

P_: Pause time at the bottom of the hole or pause time at Point R during backing;

Q_: Cutting depth of each cutting feed;

F_: Cutting feed rate;

V_: Backing distance d, which will be set by Data Parameter **P300** if not specified;
K_: Repetition times (specified when needed).

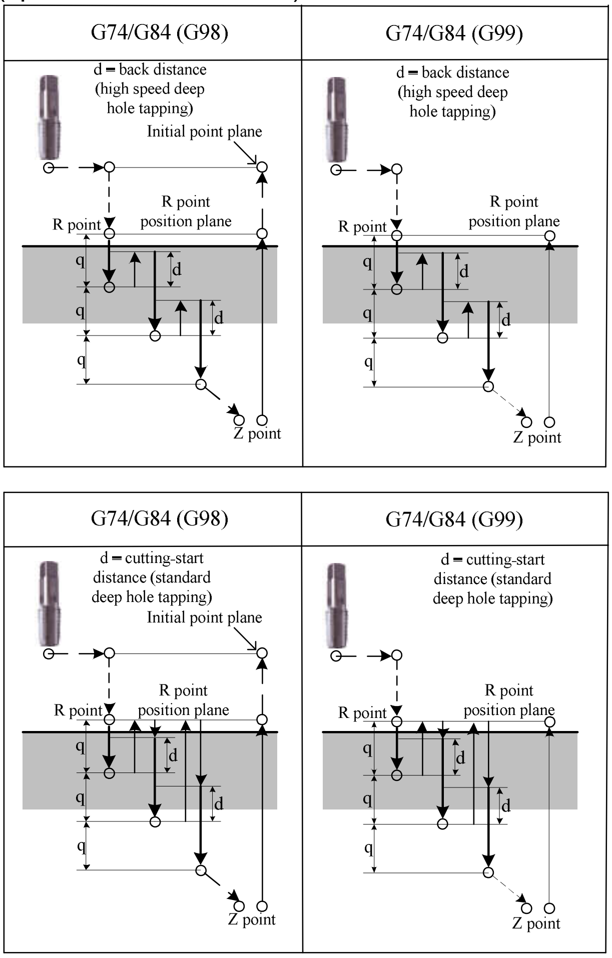


Fig. 4-5-3-1

Table 4-5-3-1

Deep-hole tapping cycle	Parameter Setting	Tapping method
Deep-hole soft tapping cycle	NO: 46#1=0 and NO: K007#7=0	When NO: 44#5=1, it is high-speed deep-hole tapping cycle; When NO: 44#5=0, it is standard deep-hole tapping cycle.
Deep-hole rigid tapping cycle	NO: 46#1=1 and NO: K007#7=1	When NO: 44#5=1, it is high-speed deep-hole tapping cycle; When NO: 44#5=0, it is standard deep-hole tapping cycle.

Deep-hole tapping cycle is divided into deep-hole soft tapping cycle and deep-hole rigid tapping cycle, which is set by the position parameter **NO: 46#1**.

Deep-hole soft tapping cycle: When the position parameter **NO: 46#1=0** and **NO: K007#7**=0, it is deep-hole soft tapping cycle, which is divided into high-speed deep-hole tapping cycle and standard soft tapping cycle, depending on the position parameter **NO: 44#5**.

High-speed deep-hole tapping cycle: When the position parameter **NO:44#5=1**, it is high-speed deep-hole tapping cycle: Perform rapid movement to Point R after positioning along the X and Y axis, perform cutting at the cutting depth Q (depth of each cutting feed) from Point R, and then retract the tool at the distance d (which is specified by the fixed cycle parameter V, and will be by the data parameter **P300** if not specified). When to retract from rigid and whether the override is valid or not are set by Position Parameter **NO:44#4**; the retraction speed override is specified by **NO:45#3**; cutting feed in rigid tapping and whether to use the same time constant for retraction are set by Position Parameter **NO:45#2**; the process of rigid tapping, selection of feedrate override and whether the signal for override cancellation is valid or not are set by Position Parameter **NO:45#4**. When Point Z is reached, the spindle stops, and then rotates reversely and moves backwards.

Standard deep-hole (soft) tapping cycle: When the position parameter **NO:44#5=0**, it is standard deep-hole tapping cycle: Perform rapid movement to Point R after positioning along the X and Y axis, perform cutting at the cutting depth Q (depth of each cutting feed) from Point R, and perform a return to Point R. Position Parameter **NO:44#4** is used to set when to retract from rigid tapping and whether the override is valid; **NO:45#3** is used to specify the retraction speed override and the position at a distance of d (set by Data Parameter **P300**) from Point R to the destination of last cutting and require re-cutting at the cutting speed of F; and **NO:45#2** is used to set whether the same time constant is adopted for cutting feed and retraction of rigid tapping. When Point Z is reached, the spindle stops, and then rotates reversely and moves backwards.

Standard deep-hole (rigid) tapping cycle: In the position mode (position parameter NO: 46.1 set to 1, K parameter NO: 7.7 set to 1), specify M29 S***** as deep-hole rigid tapping cycle before the tapping code, and use the standard deep-hole tapping cycle. The setting method is identical to that for soft standard deep-hole tapping.

Tool length compensation: When the tool length compensation G43, G44 or G49 shares the same program segment with the fixed cycle command, add or cancel the offset when positioning to Point R; in the fixed cycle mode, when the tool compensation G43, G44 or G49 is alone in a program segment, the system can add or cancel the offset in real time.

Restrictions:

G code: When the G74/G84 command and the G code (G00 to G03, G60 are modal codes (when the position parameter NO: 48#0 is 1)) in Group 01 of the same program segment are used, the system will execute the final modal code.

M code: Rotate the spindle with the auxiliary function M code before G74/G84 is specified. If the spindle rotates without command, the system will automatically adjust to anti-clockwise rotation based on the current spindle command speed in the R plane.

When G74/G84 and the M code are specified by the same program segment, the M code is executed during positioning of the first hole, and then the system deals with the next drilling activity.

When repetition times K is specified, the M code is executed only for the first hole, and not for subsequent holes.

(In the current version, M00, M01, M02, M06, M30, M98, and M99 are post-program executed, and the above M code will be executed after the current segment is executed.)

S command: If the specified spindle speed exceeds the maximum spindle speed during tapping (Data Parameter P257: spindle upper limit speed during tapping cycle), the system will send an alarm; the maximum spindle speed during rigid tapping is set by Data Parameter P294 - P296.

F command: If the specified F value exceeds the upper limit of the cutting feed rate (data parameter: the upper limit is set by P96), the upper limit shall prevail.

P command: P is a modal code; the minimum value of the parameter is set by Data Parameter **P296**, while the maximum value is set by **P297**. When the P value is less than that set by **P296**, it will run at the minimum value; when the P value is greater than that set by **P297**, it will run at the maximum value.

Shaft switch: The fixed cycle must be canceled before switching the tapping shaft. If the tapping shaft is changed in the rigid mode, the system will send No. 206 alarm.

Override: During tapping, the feedrate override and spindle speed override are 100% by default, and the machine tool will not stop working when the feed hold button and reset button are pressed until the return activity is completed.

Tool radius compensation: In this fixed cycle command, the tool radius compensation will be ignored because the command function does not require tool radius compensation.

Program restart: In the tapping cycle, the program restart function is invalid.

Note: In the process of soft tapping, rigid tapping or deep-hole rigid tapping, it is necessary to cancel the constant surface cutting speed with G97 first, or otherwise there will be incorrect thread or broken tap.

4.6 Compound Cycle (G Code)

The compound cycle comparison table (G22 - G38) is shown in Table 4-6-1.

Table 4-6-1

G code	Drilling (-Z direction)	Bottom activity	Retracting (+Z direction)	Purpose
G22	Cutting feed		Rapid movement	Groove rough milling inside circle (CCW)
G23	Cutting feed		Rapid movement	Groove rough milling inside circle (CW)
G24	Cutting feed		Rapid movement	Finish milling cycle inside full circle (CCW)
G25	Cutting feed		Rapid movement	Finish milling cycle inside full circle (CW)
G26	Cutting feed		Rapid movement	Finish milling cycle outside circle (CCW)
G32	Cutting feed		Rapid movement	Finish milling cycle outside circle (CW)
G33	Cutting feed		Rapid movement	Rectangular groove rough milling (CCW)
G34	Cutting feed		Rapid movement	Rectangular groove rough milling (CW)
G35	Cutting feed		Rapid movement	Finish milling cycle inside rectangular groove (CCW)
G36	Cutting feed		Rapid movement	Finish milling cycle inside rectangular groove (CW)
G37	Cutting feed		Rapid movement	Finish milling cycle outside rectangle (CCW)
G38	Cutting feed		Rapid movement	Finish milling cycle outside rectangle (CW)

Restrictions:

The tool radius offset (D) will be ignored during fixed cycle positioning.

4.6.1 Groove Rough Milling Inside Circle (G22/G23)

Format:

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

Function: Starting from the center of the circle, multiple circular interpolations are performed in a spiral manner until a circular groove of the programmed size is formed by machining.

Description:

G22: groove rough milling inside circle (CCW);
G23: groove rough milling inside circle (CW)
X, Y: Starting position of the X, Y plane;
Z: Machining depth, which is an absolute position in case of G90, and a position relative to the R reference plane in case of G91;
R: R reference plane position, which is an absolute position in case of G90, and a position relative to the start point of this program segment in case of G91;
I: Radius of groove inside circle;
L: Width increment in XY plane of cutting
W: First cutting depth in the direction of Z axis, which is a distance from the R reference plane to the bottom, and should be greater than 0 (if the first cutting depth exceeds the groove bottom, the machining will be done directly at the groove bottom);
Q: Cutting depth of each cutting feed;
V: Distance from the unmachined surface at the time of fast entry;
D: Tool compensation number (the corresponding tool compensation value will be determined according to the given serial number);
K: Repetition times

Cycle process:

⑴ Quickly locate to the position determined by offset of the specified point (X, Y) to the negative direction of X axis by a tool radius D multiplied by spiral entry factor;
⑵ Quickly move down to the R point plane;
⑶ Cutting in a down spiral manner at the cutting speed for W distance depth → cutting feed to the circle center
⑷ Mill out a circular surface with its radius of I in a spiral manner from the center outwardly with a progressive increase of the L value;
⑸ The Z axis quickly returns to the R reference plane;
⑹ The X and Y axis are quickly positioned to the starting position
⑺ The Z axis rapidly drops with a distance of V from the unmachined surface;
⑻ Depth of Z-axis downward cutting (Q+V);
⑼ Repeat Activity (4) to (8) until the machining for a circular surface with total depth is done.
⑽ Return to the initial point plane or the R point plane according to the specified G98 or G99.

Code track:

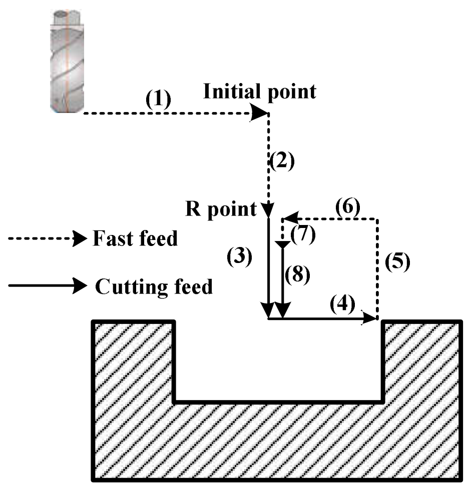


Fig. 4-6-1-1

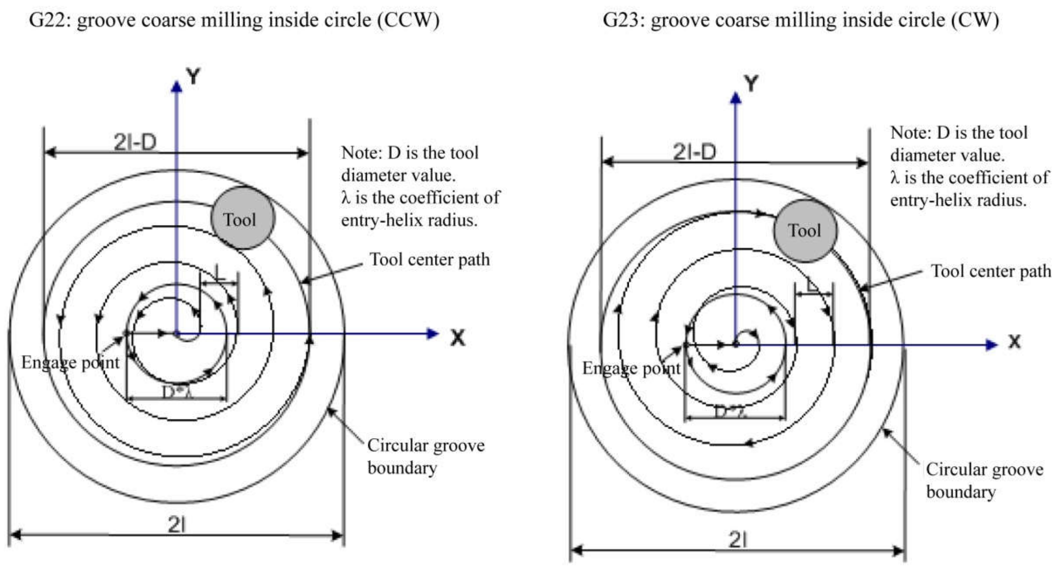


Fig. 4-6-1-2

Note:

1. When this code is used, it is recommended to change NO: 12#1 to 1.
2. The coefficient of the radius of spiral entry in groove cycle must be set greater than 0, depending on Data Parameter P269.

Example: Rough milling of a groove inside circle with the fixed cycle G22 is as shown below:

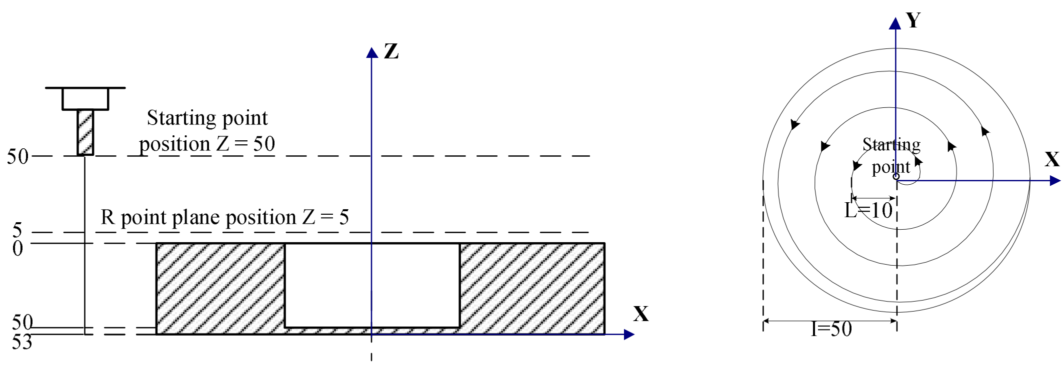


Fig. 4-6-1-3

G90 G00 X50 Y50 Z50; (G00 fast positioning)

G99 G22 X25 Y25 Z-50 R5 I50 L10 W20 Q10 V10 D1 F800; (Perform groove rough milling inside circle)

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

M30;

Restrictions: When the G22/G23 command and the G code (G00 to G03, G60 are modal codes (when the position parameter NO: 48#0 is 1)) in Group 01 of the same program segment are used, the system will execute the final modal code.

Tool radius compensation: During positioning with this fixed cycle command, the tool radius offset will be ignored and during cutting feed, the tool radius compensation specified by the program will be called.

4.6.2 Finish Milling Cycle Inside Full Circle (G24/G25)

Format:

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

Function: The tool finishes a full circle inside circle with the specified radius value I and direction, and returns upon completion of finish milling.

Description:

G24: finish milling cycle inside full circle (CCW).
G25: finish milling cycle inside full circle (CW).
X, Y: Starting position of the X, Y plane;
Z: Machining depth, which is an absolute position in case of G90, and a position relative to the R reference plane in case of G91;
R: R reference plane position, which is an absolute position in case of G90, and a position relative to the start point of this program segment in case of G91;
I: Finish-milling circle radius
J: Distance between finish-milling origin and finish-milling circle center;
D: Tool compensation number (the corresponding tool compensation value will be determined according to the given serial number);
K: Repetition times

Cycle process:

(1) Fast positioning on the XY plane;
(2) Quickly move down to the R point plane;
(3) Cutting feed to the starting point of hole bottom machining;
(4) Perform circular interpolation from the starting point with Transition Arc 1 as trajectory;
(5) Perform full circle interpolation with the inner circle of finish milling as trajectory;
(6) Perform circular interpolation with Transition Arc 4 as trajectory and return to the starting point;
(7) Return to the initial point plane or the R point plane according to the specified G98 or G99.

Code track:

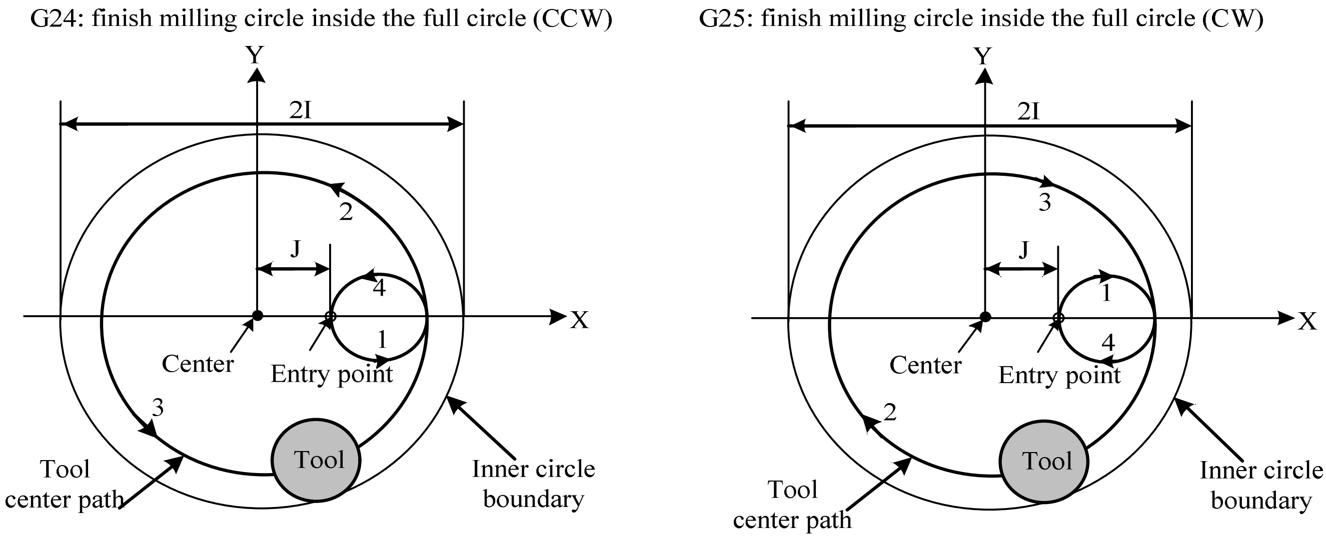


Fig. 4-6-2-1

Note: When this code is used, it is recommended to change NO: 12#1 to 1.

Example: Use the fixed cycle G24 for finish milling of a roughly milled circular groove as shown below

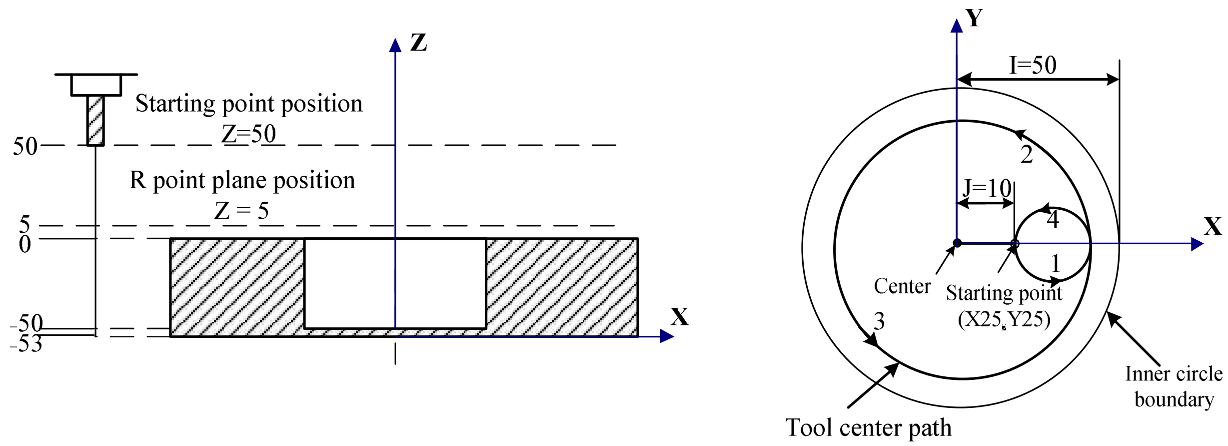


Fig. 4-6-2-2

G90 G00 X50 Y50 Z50;	(G00 fast positioning)
G99 G24 X25 Y25 Z-50 R5 I50 J10 D1 F800;	(Start the fixed cycle to move down to the bottom of the hole for finish milling cycle inside circle)
G80 X50 Y50 Z50;	(Cancel the fixed cycle and return from the R point plane)
M30;	

Restrictions: When the G24/G25 command and the G code (G00 to G03, G60 are modal codes (when the position parameter NO: 48#0 is 1)) in Group 01 of the same program segment are used, the system will execute the final modal code.

Tool radius compensation: During positioning with this fixed cycle command, the tool radius offset will be ignored and during cutting feed, the tool radius compensation specified by the program will be called.

4.6.3 Finish Milling Cycle Outside Circle (G26/G32)

Format:

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

Function: The tool finishes a full circle outside circle with the specified radius value I and direction, and returns upon completion of finish milling.

Description:

G26: finish milling cycle outside circle (CCW).

G32: finish milling cycle outside circle (CW)

X, Y: Starting position of the X, Y plane;

Z: Machining depth, which is an absolute position in case of G90, and a position relative to the R reference plane in case of G91;

R: R reference plane position, which is an absolute position in case of G90, and a position relative to the start point of this program segment in case of G91;

I: Finish-milling circle radius

J: Distance between finish-milling origin and finish-milling circle edge;

D: Tool compensation number (the corresponding tool compensation value will be determined according to the given serial number);

K: Repetition times

Cycle process:

(1) Fast positioning on the XY plane;

(2) Quickly move down to the R point plane;

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

(4) Perform circular interpolation from the starting point with Transition Arc 1 as trajectory;
(5) Perform full circle interpolation with Arc 2 and Arc 3 as trajectory;
(6) Perform circular interpolation with Transition Arc 4 as trajectory and return to the starting point;
(7) Return to the initial point plane or the R point plane according to the specified G98 or G99.

Code track:

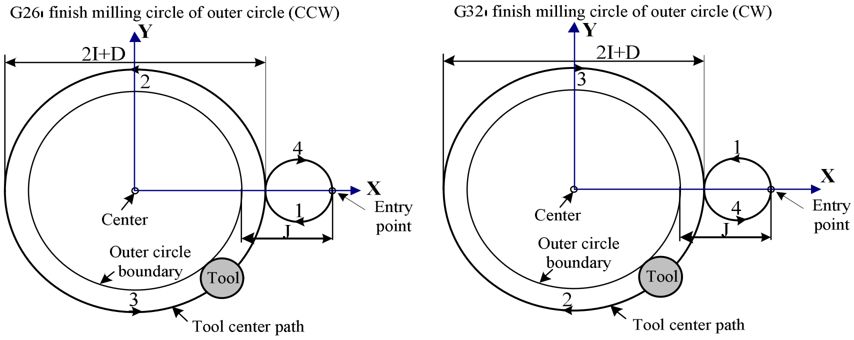


Fig. 4-6-3-1

Description:

For finish milling outside circle, the interpolation direction of transition arc differs from that of finish milling arc, and the interpolation direction in the code description refers to the interpolation direction of finish milling arc.

Example: Use the fixed cycle G26 for finish milling of a roughly milled circular groove as shown below

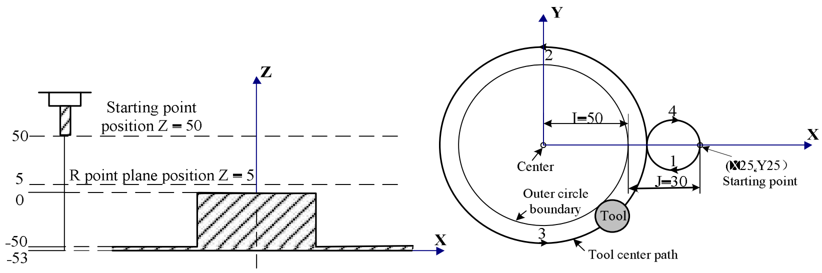


Fig. 4-6-3-2

G90 G00 X50 Y50 Z50; (G00 fast positioning)
G99 G26 X25 Y25 Z-50 R5 I50 J30 D1 F800; (Start the fixed cycle to move down to the bottom of the hole for finish milling cycle outside circle)
G80 X50 Y50 Z50; (Cancel the fixed cycle and return from the R point plane)
M30;

Restrictions: When the G26/G32 command and the G code (G00 to G03, G60 are modal codes (when the position parameter NO: 48#0 is 1)) in Group 01 of the same program segment are used, the system will execute the final modal code.

Tool radius compensation: During positioning with this fixed cycle command, the tool radius offset will be ignored and during cutting feed, the tool radius compensation specified by the program will be called.

4.6.4 Rectangular Groove Rough Milling (G33/G34)

Format:

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

Function: Starting from the rectangle center, the specified parameter data is used for linear cutting cycle until the programmed rectangular groove is machined.

Description:

G33: Rectangular groove rough milling (CCW);

G34: Rectangular groove rough milling (CW);

X, Y: Starting position of the X, Y plane;

Z: Machining depth, which is an absolute position in case of G90, and a position relative to the R reference plane in case of G91;

R: R reference plane position, which is an absolute position in case of G90, and a position relative to the start point of this program segment in case of G91;

I: Rectangular groove width in the X-axis direction;

J: Rectangular groove width in the Y-axis direction;

L: Cutting width increment in the specified plane;

W: First cutting depth in the direction of Z axis, which is a distance from the R reference plane to the bottom, and should be greater than 0 (if the first cutting depth exceeds the groove bottom, the machining will be done directly at the groove bottom);

Q: Cutting depth of each cutting feed;

V: Distance from the unmachined surface at the time of fast entry;

U: Corner arc radius, which indicates no corner arc transition if omitted;

D: Tool compensation number (the corresponding tool compensation value will be determined according to the given serial number);

K: Repetition times

Cycle process:

(1) Fast positioning to the starting point of spiral entry on the XY plane;

(2) Quickly move down to the R point plane;

(3) Use the radius compensation value multiplied by the value of Data Parameter No. 269 as the diameter for spiral entry W distance;

(4) Feeding to the rectangle center;

(5) Mill out a rectangular surface from the center outwardly with progressive increase of the L value;

(6) The Z axis quickly returns to the R reference plane;

(7) Fast positioning to the starting point of spiral entry on the XY plane;

(8) The Z axis rapidly drops with a distance of V from the unmachined surface;

(9) Depth of Z-axis downward cutting (Q+V);

(10) Repeat Activity (4) to (9) until the machining for a rectangular surface with total depth is done.

(11) Return to the initial point plane or the R point plane according to the specified G98 or G99.

Code track:

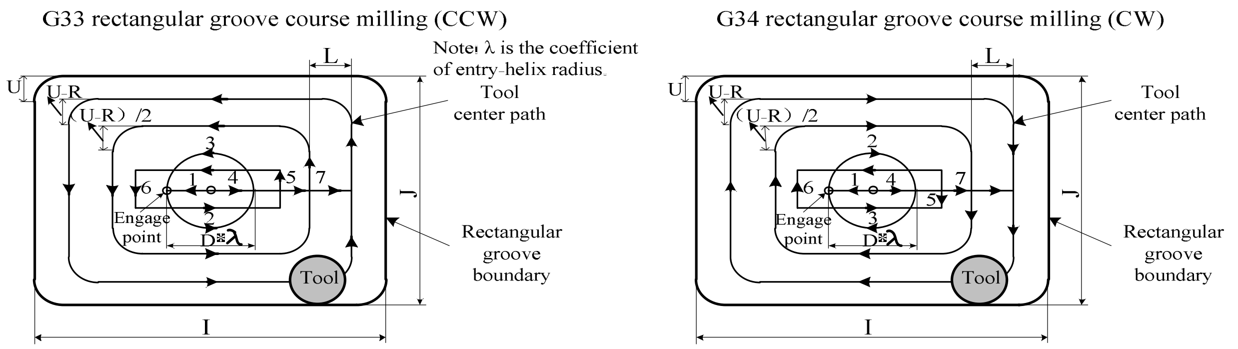


Fig. 4-6-4-1

Note: When this code is used, it is recommended to change NO: 12#1 revised as 1

Example: Rough milling of a groove inside rectangle with the fixed cycle G33 is as shown below:

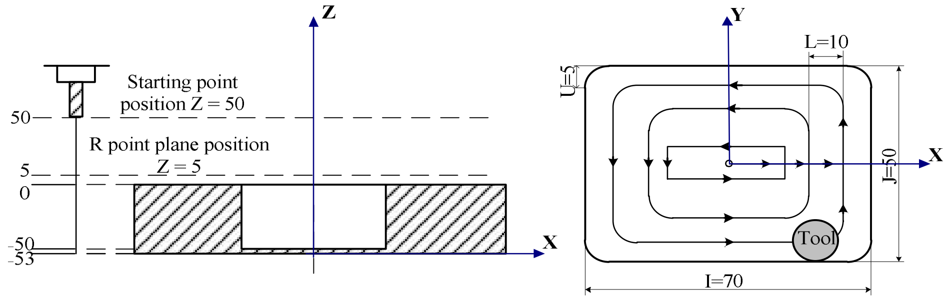


Fig. 4-6-4-2

G90 G00 X50 Y50 Z50;	(G00 fast positioning)
G99 G33 X25 Y25 Z-50 R5 I70 J50 L10 W20 Q10 V10 U5 D1 F800;	(Perform groove rough milling inside rectangle)
G80 X50 Y50 Z50;	(Cancel the fixed cycle and return from the R point plane)

M30;

Restrictions: When the G33/G34 command and the G code (G00 to G03, G60 are modal codes (when the position parameter NO: 48#0 is 1)) in Group 01 of the same program segment are used, the system will execute the final modal code.

Tool radius compensation: During positioning with this fixed cycle command, the tool radius offset will be ignored and during cutting feed, the tool radius compensation specified by the program will be called.

4.6.5 Finish Milling Cycle Inside Rectangular Groove (G35/G36)

Format:

G98/G99 G35/G36 X_ Y_ Z_ R_ I_ J_ L_ U_ D_ F_ K_;

Function: The tool performs finish milling inside rectangle with the specified width and direction and returns upon completion of finish milling.

Description:

G35: Finish milling cycle inside rectangular groove (CCW).

G36: Finish milling cycle inside rectangular groove (CW).

X, Y: Starting position of the X, Y plane;

Z: Machining depth, which is an absolute position in case of G90, and a position relative to the R reference plane in case of G91;

R: R reference plane position, which is an absolute position in case of G90, and a position relative to the start point of this program segment in case of G91;
I: Rectangle width in the X-axis direction;
J: Rectangle width in the Y-axis direction;
L: Distance between finish-milling origin and rectangular side X-axis positive direction;
U: Corner arc radius, which indicates no corner arc transition if omitted; When 0<|U|<tool radius, it will give an alarm;
D: Tool compensation number (the corresponding tool compensation value will be determined according to the given serial number);
K: Repetition times

Cycle process:

(1) Fast positioning to the starting position on the XY plane;
(2) Quickly move down to the R point plane;
(3) Cutting feed to the bottom of the hole;
(4) Perform circular interpolation from the starting point with Transition Arc 1 as trajectory;
(5) Perform linear and circular interpolation with 2-3-4-5-6 as trajectory;
(6) Perform circular interpolation with Transition Arc 7 as trajectory and return to the starting point;
(7) Return to the initial point plane or the R point plane according to the specified G98 or G99.

Code track:

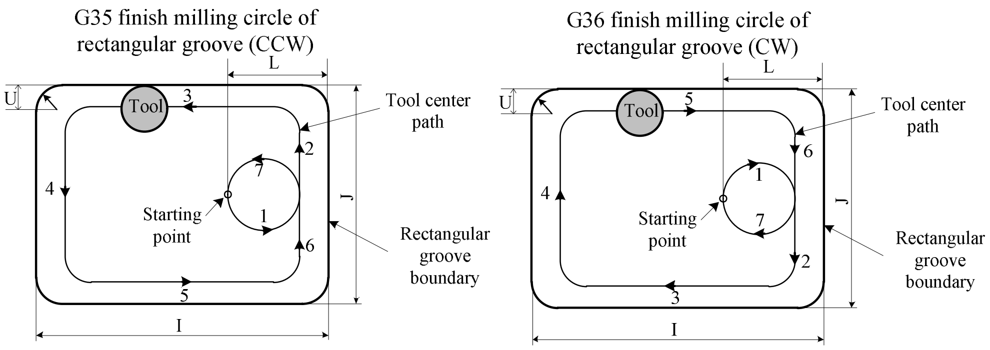


Fig. 4-6-5-1

Note: When this code is used, it is recommended to change NO: 12#1 revised as 1

Example: Use the fixed cycle G35 for finish milling of a roughly milled groove as shown below

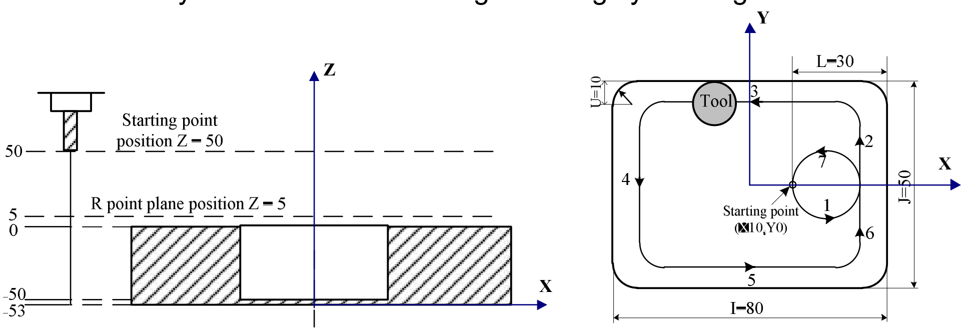


Fig. 4-6-5-2

G90 G00 X50 Y50 Z50; (G00 fast positioning)
G99 G35 X10 Y0 Z-50 R5 I80 J50 L30 U10 D1 F800; (start the fixed cycle to move down to the hole bottom for milling inside rectangular groove)
G80 X50 Y50 Z50; (Cancel the fixed cycle and return from the R point plane)
M30;

Restrictions: When the G35/G36 command and the G code (G00 to G03, G60 are modal codes (when the position parameter NO: 48#0 is 1)) in Group 01 of the same program segment are used, the system will execute the final modal code.

Tool radius compensation: During positioning with this fixed cycle command, the tool radius offset will be ignored and during cutting feed, the tool radius compensation specified by the program will be called.

4.6.6 Finish Milling Cycle Outside Rectangle (G37/G38)

Format:

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

Function: The tool performs finish milling outside rectangle with the specified width and direction and returns upon completion of finish milling.

Description:

G37: Finish milling cycle outside rectangle (CCW).
G38: Finish milling cycle outside rectangle (CW).
X, Y: Starting position of the X, Y plane;
Z: Machining depth, which is an absolute position in case of G90, and a position relative to the R reference plane in case of G91;
R: R reference plane position, which is an absolute position in case of G90, and a position relative to the start point of this program segment in case of G91;
I: Rectangle width in the X-axis direction;
J: Rectangle width in the Y-axis direction;
L: Distance between finish-milling origin and rectangular side X-axis direction;
U: Corner arc radius, which indicates no corner arc transition if omitted;
D: Tool compensation number (the corresponding tool compensation value will be determined according to the given serial number);
K: Repetition times

Cycle process:

(1) Fast positioning to the starting position on the XY plane;
(2) Quickly move down to the R point plane;
(3) Cutting feed to the bottom of the hole;
(4) Perform circular interpolation from the starting point with Transition Arc 1 as trajectory;
(5) Perform linear and circular interpolation with 2-3-4-5-6 as trajectory;
(6) Perform circular interpolation with Transition Arc 7 as trajectory and return to the starting point;
(7) Return to the initial point plane or the R point plane according to the specified G98 or G99.

Code track:

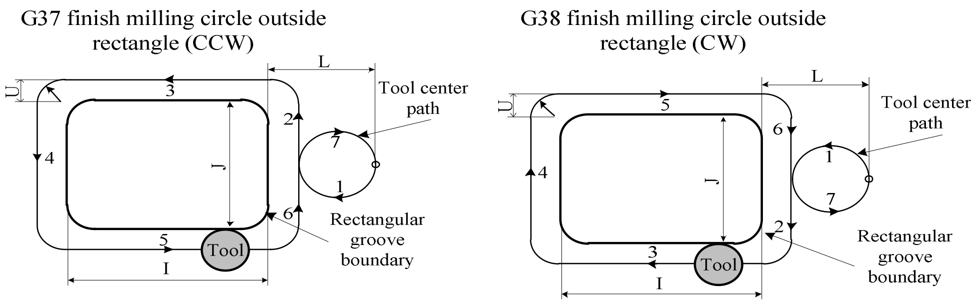


Fig. 4-6-6-1

Description: For finish milling outside rectangle, the interpolation direction of transition arc differs from that of finish milling arc, and the interpolation direction in the code description refers to the interpolation direction of finish milling arc.

Example: Perform finish milling outside rectangle using the fixed cycle G37.

G90 G00 X50 Y50 Z50;　　　　　　　(G00 fast positioning)

G99 G37 X25 Y25 Z-50 R5 I80 J50 L30 U10 D1 F800;	(Perform finish milling outside rectangle at the bottom of the hole in fixed cycle)
G80 X50 Y50 Z50;	(Cancel the fixed cycle and return from the R point plane)
M30;	

Restrictions: When the G37/G38 command and the G code (G00 to G03, G60 are modal codes (when the position parameter NO: 48#0 is 1)) in Group 01 of the same program segment are used, the system will execute the final modal code.

Tool radius compensation: During positioning with this fixed cycle command, the tool radius offset will be ignored and during cutting feed, the tool radius compensation specified by the program will be called.

4.6.7 Cancel Fixed Cycle (G80)

Format: G80

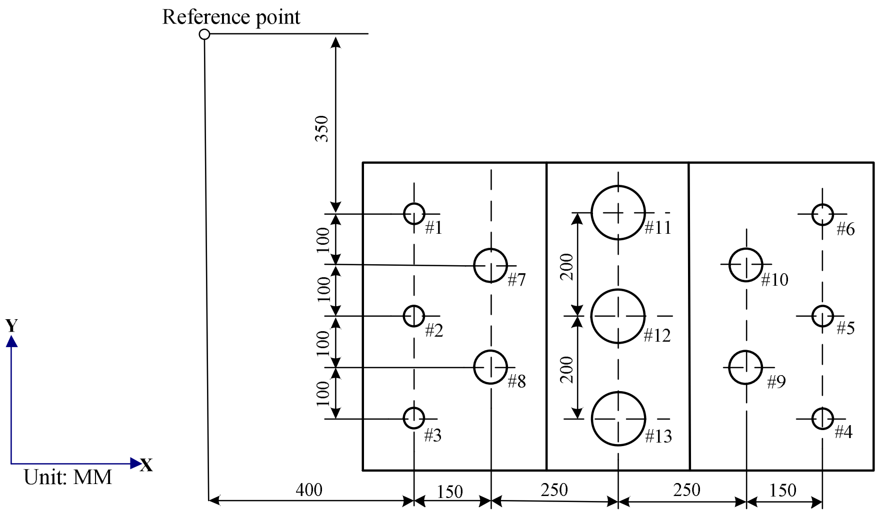
Function: Cancel the fixed cycle

Description: Cancel all fixed cycles and perform normal operations. R and Z points are also cancelled. Other drilling and boring data are also eliminated.

Example:

M3 S100;	The spindle starts rotating
G90 G99 G88 X300 Y-250 Z-150 R-120 F120;	
	Positioning, Hole 1 boring, and then return to Point R
Y-550;	Positioning, Hole 2 boring, and then return to Point R
Y-750;	Positioning, Hole 3 boring, and then return to Point R
X1000;	Positioning, Hole 4 boring, and then return to Point R
Y-550;	Positioning, Hole 5 boring, and then return to Point R
G98 Y-750;	Positioning, Hole 6 boring, and then return to the initial position plane
G80;	
G28 G91 X0 Y0 Z0;	Return to reference point and cancel the fixed cycle
M5;	The spindle stops rotating
M30;	

For example: The tool length compensation is used below to illustrate the use of fixed cycle.



 # 1 - 6...Drilling for Φ10 holes

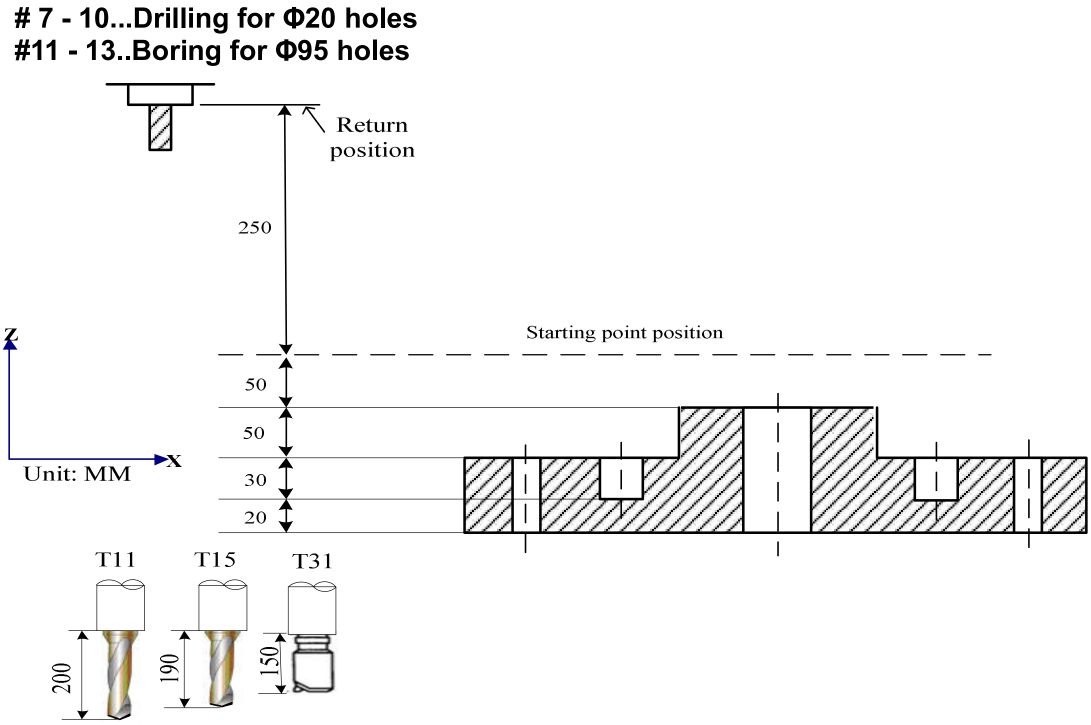


Fig. 4-6-7-1

The values of Offset Number 11, 15 and 31 are 200, 190 and 150 respectively, set as offset. The program is as follows:

N001 G92 X0 Y0 Z0 ;	The coordinate system is set at the reference point.
N002 G90 G00 Z250 T11 M6 ;	Replace the tool.
N003 G43 Z0 H11 ;	Perform tool length compensation at the initial point.
N004 S300 M3 ;	The spindle starts.
N005 G99 G81 X400 Y-350 ; Z-153 R-97 F120 ;	Hole 1 machining after positioning.
N006 Y-550 ;	Hole 2 machining after positioning and return to the R point plane.
N007 G98 Y-750 ;	Hole 3 machining after positioning and return to the initial point plane.
N008 G99 X1200 ;	Hole 4 machining after positioning and return to the R point plane.
N009 Y-550 ;	Hole 5 machining after positioning and return to the R point plane.
N010 G98 Y-350 ;	Hole 6 machining after positioning and return to the initial point plane.
N011 G00 X0 Y0 M5 ;	Return to the reference point and the spindle stops.
N012 G49 Z250 T15 M6 ;	Cancel the tool length compensation and replace the tool.
N013 G43 Z0 H15 ;	Tool length compensation on the initial point plane.
N014 S200 M3 ;	The spindle starts.
N015 G99 G82 X550 Y-450 ; Z-130 R-97 P30 F70 ;	Hole 7 machining after positioning and return to the R point plane.

N016 G98 Y-650 ;	Hole 8 machining after positioning and return to the initial point plane.
N017 G99 X1050 ;	Hole 9 machining after positioning and return to the R point plane.
N018 G98 Y-450 ;	Hole 10 machining after positioning and return to the initial point plane.
N019 G00 X0 Y0 M5 ;	Return to the reference point and the spindle stops.
N020 G49 Z250 T31 M6 ;	Cancel the tool length compensation and replace the tool.
N021 G43 Z0 H31 ;	Tool length compensation on the initial point plane.
N022 S100 M3 ;	The spindle starts.
N023 G85 G99 X800 Y-350 ; Z-153 R47 F50 ;	Hole 11 machining after positioning and return to the R point plane.
N024 G91 Y-200 ; Y-200 ;	Hole 12 and Hole 13 machining after positioning and return to the R point plane.
N025 G00 G90 X0 Y0 M5 ;	Return to the reference point and the spindle stops.
N026 G49 Z0 ;	Cancel the tool length compensation.
N027 M30 ;	The program ends.

4.7 Tool compensation (G code)

4.7.1 Tool Length Compensation (G43, G44, G49)

Function:

G43: specify the forward compensation of tool length.

G44 specify the reverse compensation of tool length.

G49: cancel the tool length compensation.

Format:

The system supports two tool length offset methods (A/B), and uses the position parameter **N0: 39#0** to set the tool length offset mode.

Method A:

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

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;

Cancel the tool length offset mode: G49 or H0.

Description:

The above code is used to move the final position of specified axis command by an offset. The difference between the tool length value assumed during programming (usually set as the first tool) and the tool length value used in actual machining is preset in the offset memory, so there is no need to change the program, and it is possible to machine parts using tools of different lengths by only changing tool length compensation value.

G43 and G44 are used to specify different offset directions, and H code to specify the offset number.

1. Offset direction

G43: Positive offset (most commonly used)

G44: Negative offset

Whether it is an absolute value code or an incremental value code, in case of G43, the coordinate value of final position of the specified axis movement command in the program is added to the offset specified by the H code (set in the offset memory); in case of G44, minus the offset specified by the H code, and then use the calculated result as the coordinate value of final position.

G43 and G44 are modal G codes and valid before encountering other G codes in the same group.

2. Specification of depth offset

A length offset number will be specified by the H code, and the depth offset corresponding to this offset number is added to or subtracted from the Z-axis movement code value in the program to form a new Z-axis movement code. The offset number can be specified from H00 to H255 as needed.

The depth offset can be set within the following range:

Table 4-7-1-1

	Range
Compensation quantity H (input in mm)	-999.999 mm～+999.999mm
Compensation quantity H (input in inch)	-39.3700 inch～+39.3700 inch

Offset Number 00, that is, the depth offset corresponding to H00 is 0 and cannot be set in the system.

Note: When the depth offset is changed due to change in the offset number, the old depth offset is directly replaced with the new one, instead of the new depth offset added to the old one.

For example:

H01........................ ....Depth offset 20

H02......................... ....Depth offset 30

G90 G43 Z100 H01; .... Move from Z to 120

G90 G43 Z100 H02; .... Move from Z to 130

3. Effective order of the offset number

Once the length offset mode is established, the current offset number takes effect immediately, and when the offset number changes, the new offset value will immediately replace the old one. For example:

```
O×××××;
G43 Z10 H01;   (1) Offset Number H01 becomes effective
G44 Z20 H02;   (2) Offset Number H02 becomes effective
Z30 H03;       (3) Offset Number H03 becomes effective
G49;           (4) Cancel the tool offset at the end of this segment
M30;
```

4. Cancel tool length compensation

Cancel tool compensation with G49 or H00. After the command G49, the system immediately cancels the tool length compensation; after the command H00, the compensation axis address or compensation command must be programmed, or otherwise the tool length compensation cannot be canceled.

Note: 1. Tool length offset method B: After two or more axes are executed, G49 is used to cancel the offset of all axes while H00 is only used to cancel the offset of the axis perpendicular to the specified plane.

2. It is recommended to add Z-axis movement code for establishment and cancellation of length compensation. Otherwise, the length compensation will be established or canceled with the current point. Therefore, when using G49, please make sure that the Z-axis is at a safe height to prevent collision or damage to workpiece.

5. Specific examples of tool length compensation

a) Tool length compensation (Holes 1, 2 and 3 machining)

b) H01=offset-4

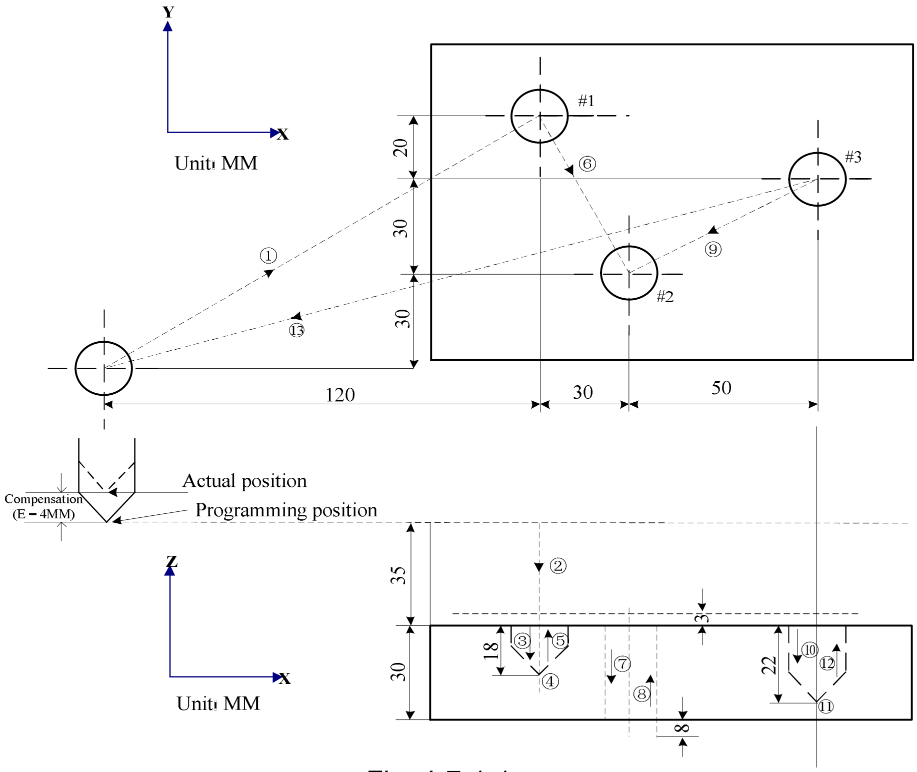


Fig. 4-7-1-1

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 Tool Radius Compensation (G40/G41/G42)

Code format:

G41 D_ X_Y_;
G42 D_ X_Y_;
G40 X_Y_;

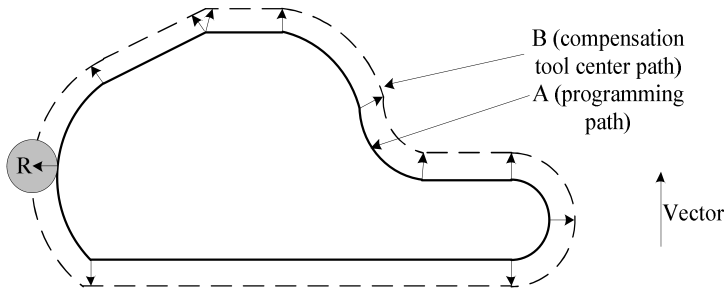
Function:

G41: specify the left-side compensation in tool movement direction.
G42: specify the right-side compensation in tool movement direction.
G40: cancel the tool radius compensation.

Description:

1. Tool radius compensation function

As shown in the figure below, Workpiece A is cut with a tool of radius R; the tool center path is B as shown in the figure, and Path B is at a distance of R from A. The distance the tool offsets from Workpiece A at the radius is called compensation.



Compensation and vector

Fig. 4-7-2-1

The programmer uses the tool radius compensation mode to program the machining process. During machining, the tool diameter is measured and recorded into the CNC memory, and the tool path becomes the compensation path B.

2. Compensation quantity (D value)

A radius offset number will be specified by the D code, and the depth offset corresponding to this offset number is added to or subtracted from the movement code value in the program to form a new movement code. The offset number can be specified from D00 to D255 as needed. Whether the compensation quantity is measured by diameter value or by radius value is set by Position Parameter **N0: 40#7.**

The LCD/MDI panel can be used to preset the depth offset corresponding to the offset number in the offset memory in advance.

The compensation quantity can be set within the following range:

Table 4-7-2-1

	Range
Compensation quantity D (input in mm)	-999.999mm～+999.999mm
Compensation quantity D (input in inch)	-39.3700 inch～+39.3700 inch

Note: The compensation quantity of D00 is set to 0 by default in the system, which cannot

be set or modified by users.
The compensation plane must be changed after canceling the compensation mode. Without canceling the compensation mode, the system will give an alarm when any change is made to the compensation plane.

3. Plane selection and vector
Compensation calculation is performed in the plane selected by G17, G18 and G19. This plane is called compensation plane. For example, when the XY plane is selected, the compensation calculation and the vector calculation are performed with (X, Y) in the program. The coordinate values of axes that are not in the compensation plane are not affected by compensation.
When three-axis control is performed at the same time, only the tool path projected on the compensation plane is compensated.
The compensation plane must be changed after canceling the compensation mode.

Table 4-7-2-2

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

4. G40, G41 and G42
Use G40, G41 and G42 to cancel and execute the tool radius compensation vector. They can be combined with G00 and G01 to define a mode which determines the value and direction of compensation vector.

Table 4-7-2-3

G code	Function
G40	Cancel tool radius compensation
G41	Tool radius left compensation
G42	Tool radius right compensation

5. G53, G28 and G30 in tool radius compensation mode
When G53, G28 or G30 is specified in the tool radius compensation mode, the offset vector of the tool radius offset axis is canceled when moving to the specified position (canceled when moving to the command position in case of G53 and canceled when moving to the reference point in case of G28 or G30), and the axes other than the tool radius offset axis are not canceled. When G53 is in the same segment as G41//G42, all axes will cancel the radius compensation when moving to the command position; when G28 or G30 is in the same segment as G41//G42, all axes will do so when moving to the reference point. The canceled tool radius compensation vector will be restored in the next compensation-plane program segment for recovery.

Note: In the compensation mode, it is possible to use Position Parameter NO: 40#2 to determine whether the compensation is temporarily canceled when G28 or G30 is specified to move to the intermediate point.

Cancel tool radius compensation (G40)

In the G00 or G01 state, use the following code, G40 X__ Y__;
The linear movement from the old vector of the starting point toward the end point. In the G00 mode, each axis moves quickly to the end point. Use this code to make the system state switch from tool compensation state to cancelled tool compensation.
If it is only G40, the tool will not move when there is no command for X__ Y__**.**

Tool radius compensation - left (G41)

1) In case of G00, G01

G41 X__ Y__ D__; At the end point of the program segment, the code forms a new vector perpendicular to the direction of (X, Y), and the tool moves from the tip of the old vector at the starting point to the tip of the new vector.

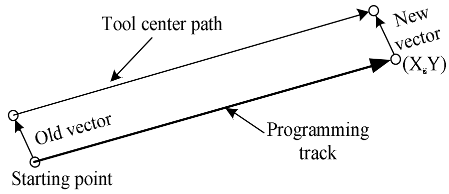


Fig. 4-7-2-2

When the old vector is zero, the code is used to cause the tool state to switch from the cancelled tool offset to the tool radius compensation. At this moment, the offset value is specified by the D code.

2) In case of G02, G03

G41……;
……
……
G02 /G03 X__ Y__ R__ ;

The above program can from a new vector, which is located on the line connecting the center and end point of the arc. In view of the arc's forward direction pointing to the left (or right), the tool center moves along the arc from the old vector tip of the arc toward the new vector tip, however, provided that the old vector has been correctly formed.
The offset vector is centrifugal or points to the arc center from the starting or end point.

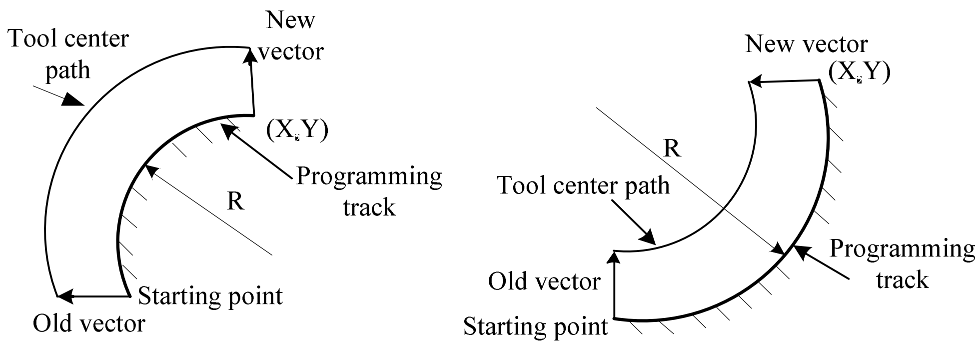


Fig. 4-7-2-3

Tool radius compensation - right (G42)

G42 is the opposite of G41, and the tool will offset on the right side of the workpiece along the tool advance direction. That is, the vector direction defined by G42 is exactly opposite to that defined by G41. The offset method is identical to G41 except that the vector direction is opposite.

1) In case of G00, G01

G42 X__ Y__ D__ ;
G42 X__ Y__ ;

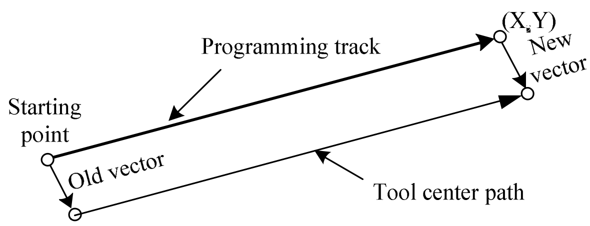


Fig. 4-7-2-4

2) In case of G02, G03

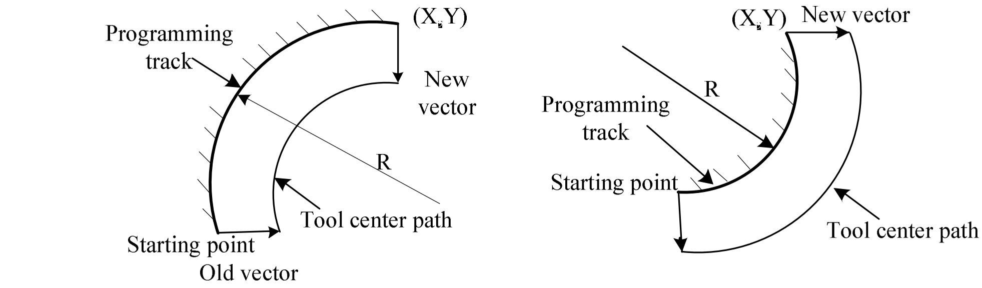


Fig. 4-7-2-5

6. General considerations about offset:

(A) Designation of offset number

G41, G42 and G40 are modal codes, and the offset number is specified by the D code. It can be specified anywhere before the state shifts from the cancelled offset to the tool radius compensation.

(B) Shift from the cancelled offset state to the tool radius compensation state

The movement code when entering into the tool radius compensation state from the cancelled offset state must be positioning (G00) or linear interpolation (G01), and circular interpolation (G02, G03) cannot be used.

(C) Conversion of left and right tool radius compensation

When the offset direction changes from left to right, or from right to left, it will usually go through the offset cancelled state. However, positioning (G00) or linear interpolation (G01) can be directly converted without going through the offset cancelled state. The tool path at this time is as shown below:

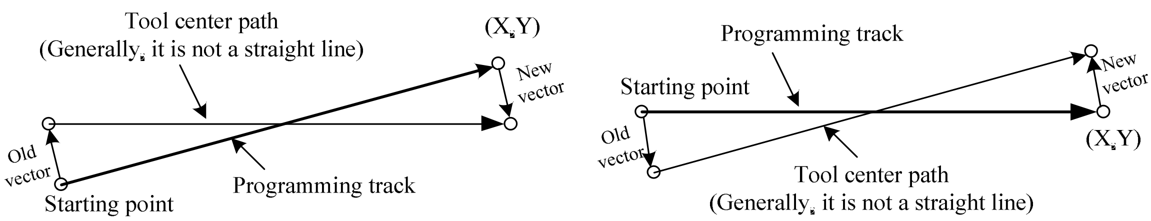


Fig. 4-7-2-6

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

(D) Change in offset

The change in offset generally occurs in the offset cancelled state during tool replacement, but can also occur in the offset state for positioning (G00) and linear interpolation, as shown in the following figure.

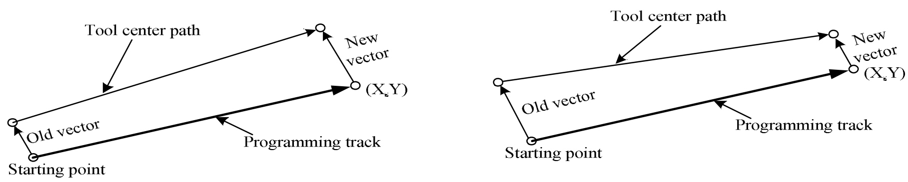


Figure 4-7-2-7 Change in offset

Volume I Programming instructions

(E) Positive and negative offset and tool center path

When the offset is set to a negative value, the workpiece machined is equivalent to the case where G41 and G42 on the program sheet are converted. Therefore, cutting or machining along the outer side of workpiece becomes cutting or machining along the inner side of workpiece.

As general programming shown below, assume the offset is a positive value:

When the tool path is programmed as shown in Figure (A), if the offset is set to a negative value, the tool movement path is as shown in Figure (B); similarly, when the tool path is programmed as shown in Figure (B), if the offset is set negative, the tool movement path is as shown in Figure (A).

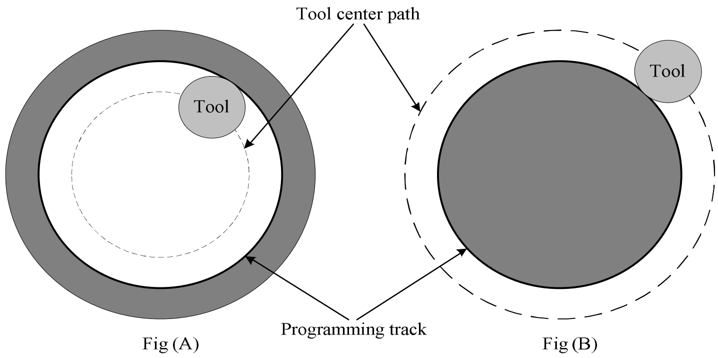


Fig. 4-7-2-8

Graphs with sharp corners are commonly used (patterns with sharp circular interpolation). However, when the offset is set to a negative value, the inner circle of the part cannot be machined. In case of cutting an inner sharp corner, insert an arc of appropriate radius there, and then perform cutting after smooth transition.

Selection of left or right compensation depends on whether the compensation direction is on the left or right side of the tool movement direction relative to the workpiece (the workpiece is considered motionless). G41 or G42 puts the system into compensation mode while G40 causes the system to cancel the compensation mode.

An example of compensation program is as follows:

Program Segment (1) is called start part, and in this segment, G41 changes the mode from compensation cancelled to compensation. At the end point of this segment, the tool center is compensated with the tool radius perpendicular to the next program path (from P1 to P2). The tool compensation quantity is specified by D07, that is, the compensation number is set to 7, and G41 represents the tool path left compensation.

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

Example of tool path compensation program

G92 X0 Y0 Z0;

(1) N1 G90 G17 G0 G41 D7 X250 Y550; (The compensation quantity must be preset with the compensation number)

(2) N2 G1 Y900 F150 ;

(3) N3 X450 ;

(4) N4 G3 X500 Y1150 R650 ;

(5) N5 G2 X900 R-250 ;

(6) N6 G3 X950 Y900 R650 ;

(7) N7 G1 X1150 ;

(8) N8 Y550 ;

(9) N9 X700 Y650 ;

(10) N10 X250 Y550 ;

(11) N11 G0 G40 X0 Y0 ;

Volume I Programming instructions

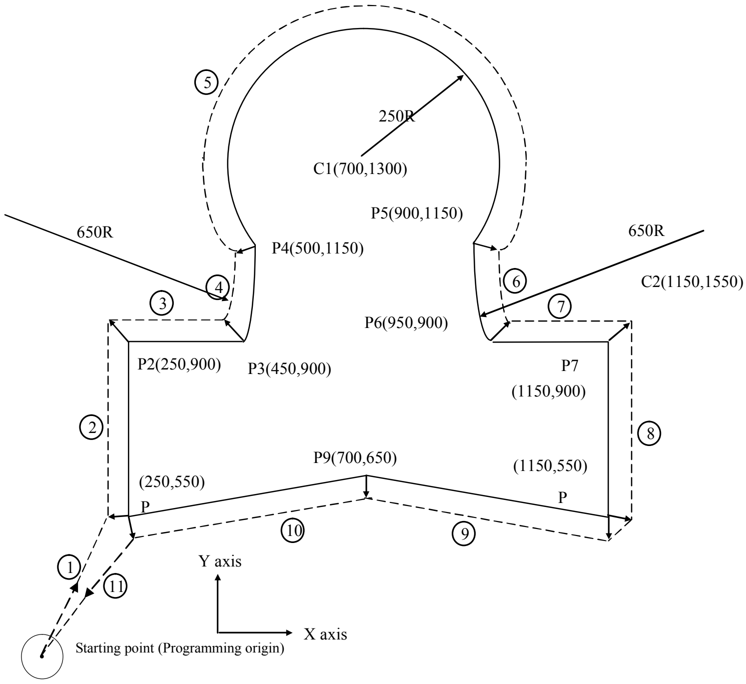


Fig. 4-7-2-9

4.7.3 Detailed Description of Tool Radius Compensation

Concept: Inner side and outer side: When the angle of the tool paths established by two program segments exceeds 180°, the path is called the inner side, and when the angle is between 0° and 180°, it is called the outer side.

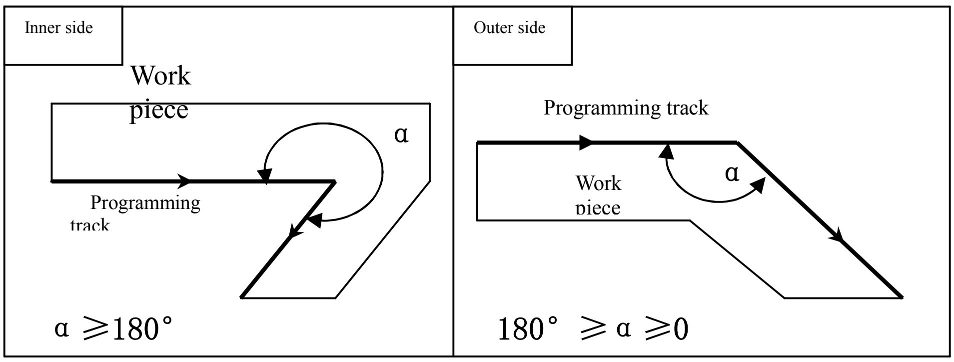


Fig. 4-7-3-1

Meaning of symbols:

Use the following symbols in the following figures:

——S indicates that a single program segment is executed once at this position;

——SS indicates that a single program segment is executed twice in this position;

——SSS indicates that a single program segment is executed three times in this position;

——L indicates that the tool moves in a straight line;

——C indicates that the tool moves along the arc;

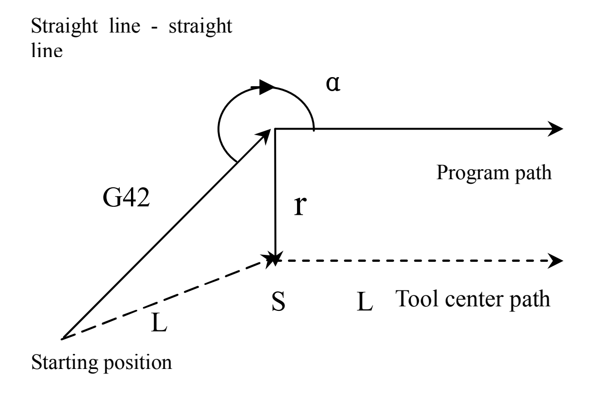
——r represents the tool radius compensation value;

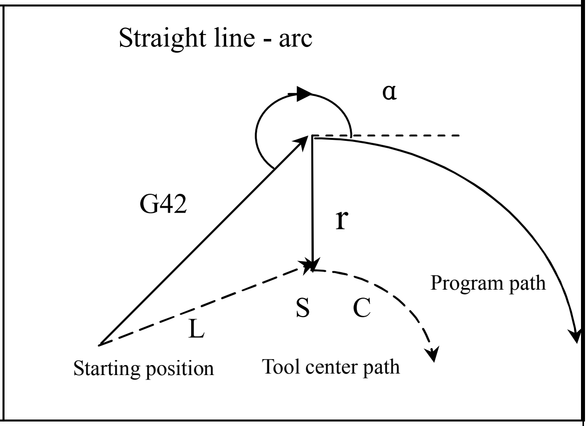
——Point of intersection is a position where two programming tracks intersect after they are offset by r;

——O represents the tool center.

1. **Tool movement at the start of cutting:** when the offset cancelled mode is changed to the offset mode, the tool moves as shown below (at the start of cutting):

(a) Move along the inner side of the corner (α≥180°)

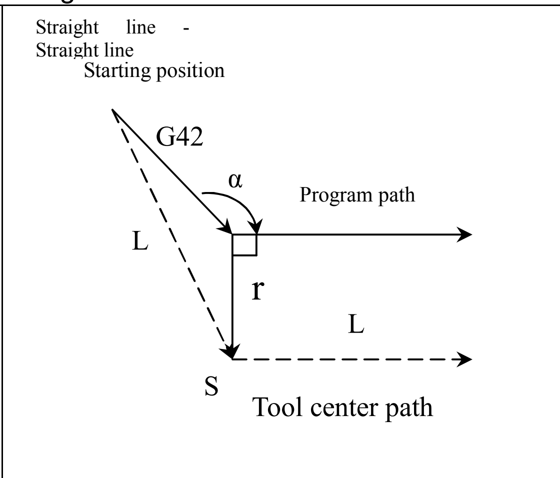


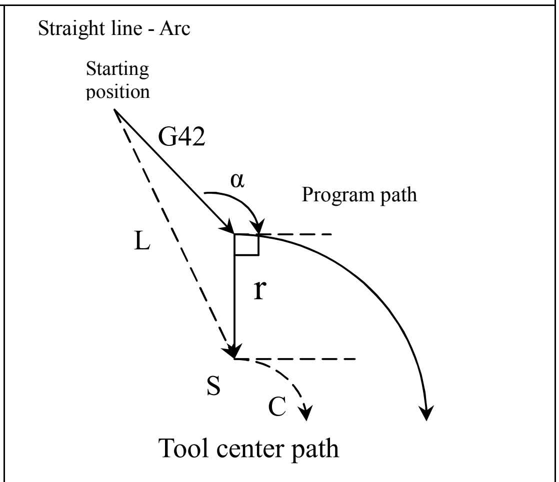


(b) Move along the outer side of the corner which is an obtuse angle (180°>α≥90°)

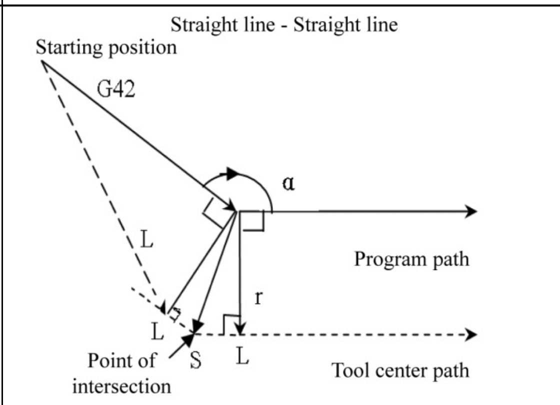
The tool path has 2 types (A and B) when the compensation is started or cancelled, depending on Position Parameter NO: 40#0:

A

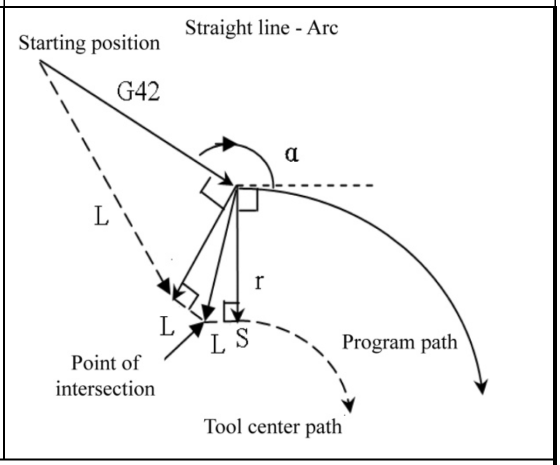




B

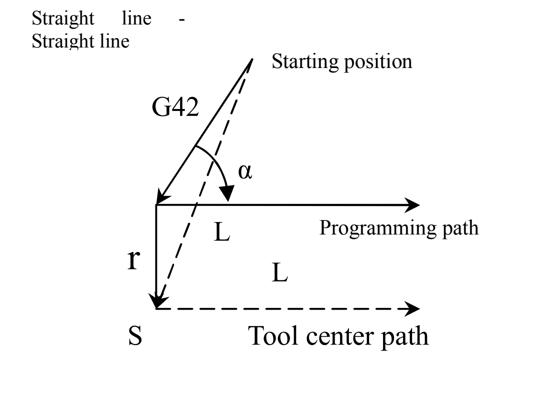
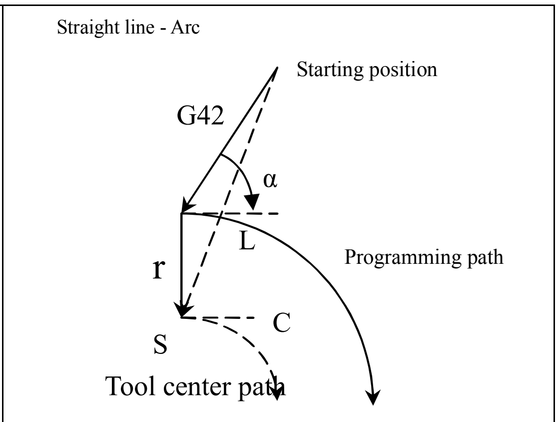
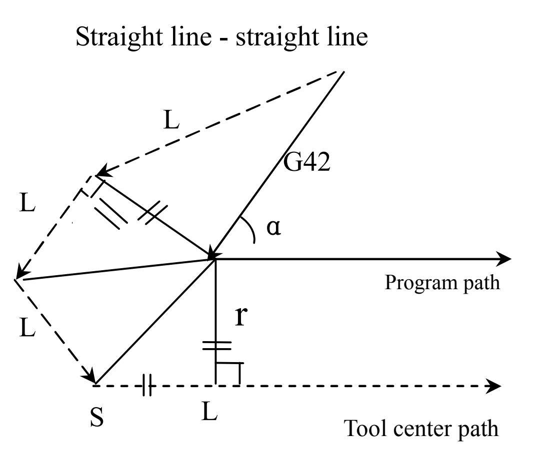
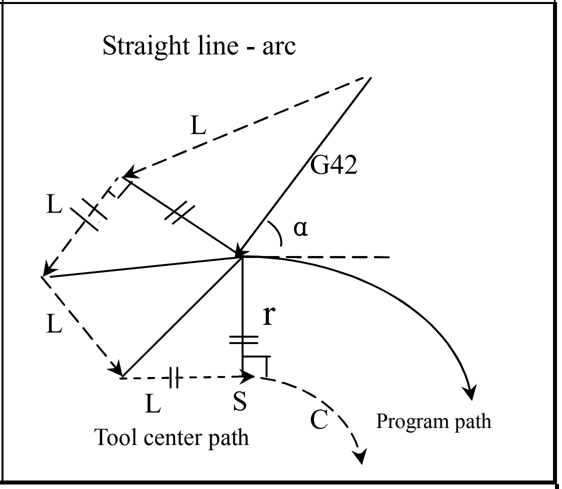


Note: The point of intersection means the position where two continual program compensation paths are intersected.



(C) Move along the outer side of the acute angle (α<90°)

The tool path has 2 types (A and B) when the compensation is started or cancelled, depending on Position Parameter NO: 40#.0:

A		
B		

(d) Move along the outer side of the corner which is an acute angle of less than 1°, straight line → straight line. (α<1°)

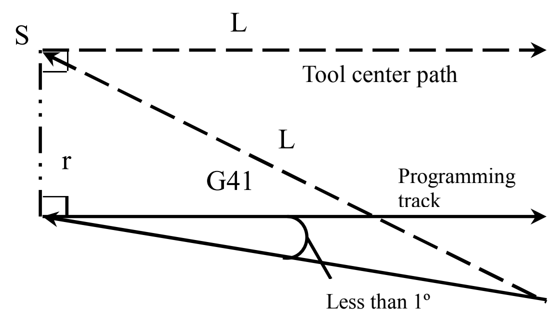


Fig. 4-7-3-2

2. Tool movement in offset mode

The compensation plane cannot be changed during execution of compensation mode, or otherwise an alarm will be generated and the tool will stop. In the offset mode, the tool moves as shown below:

(a) Move along the inner side of the corner (α≥180°)

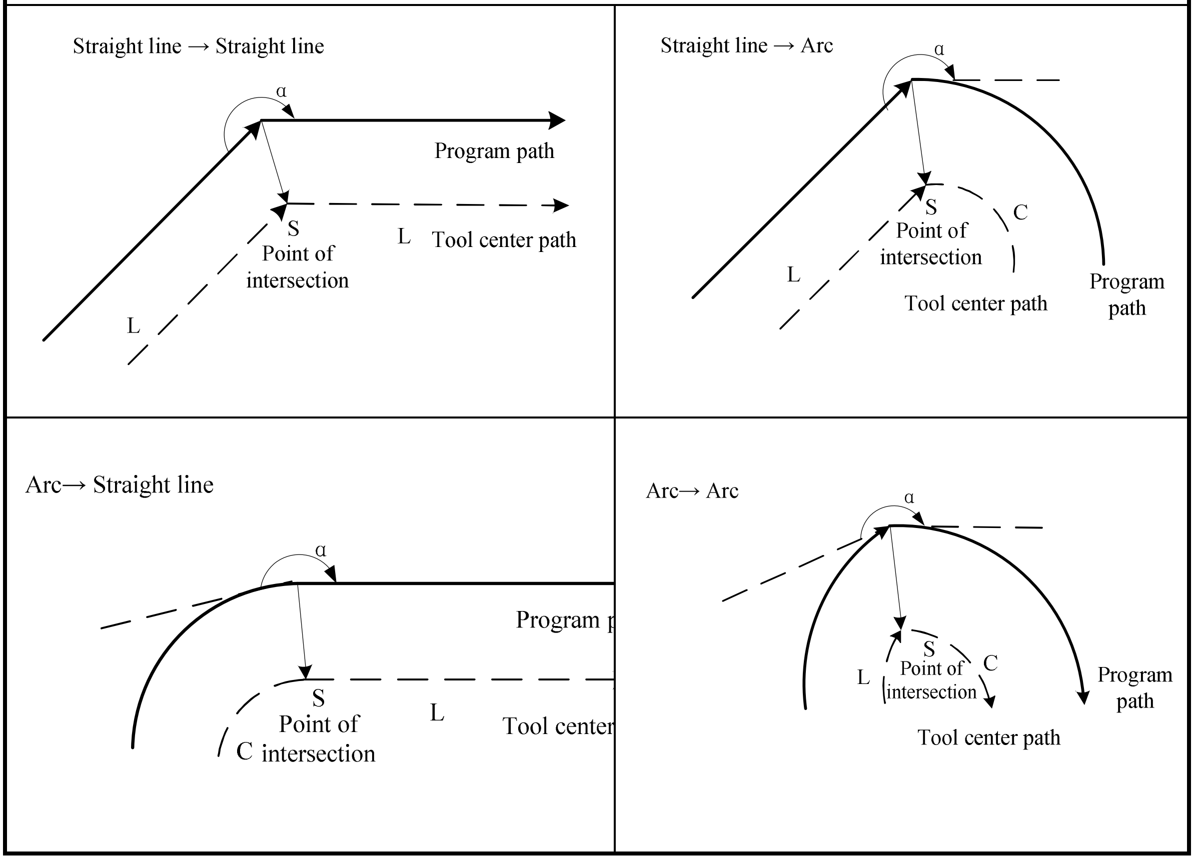


Fig. 4-7-3-3

3. Special circumstances

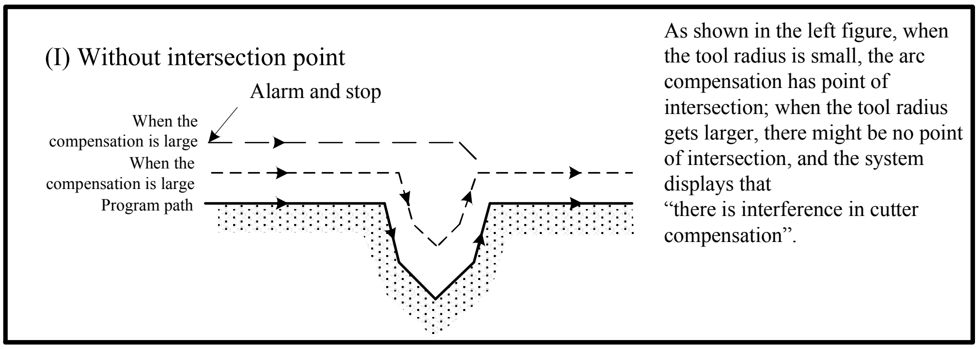


Fig. 4-7-3-4

4. Tool movement in offset cancelled mode

In the compensation mode, when a program segment that meets any of the following conditions is executed, the system will enter into the compensation cancelled mode, and the activity of this segment is called compensation cancellation.

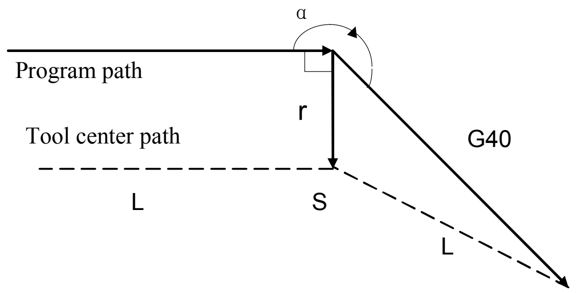
a) G40

b) The tool radius compensation number is 0.

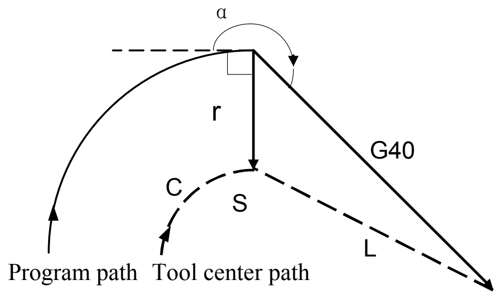
The arc code (G03 and G02) cannot be used to perform compensation cancellation. If a command arc is generated, an alarm will be generated and the tool will stop.

(a) Move along the inner side of the corner (α≥180°)

Straight line →
Straight line



Arc→ Straight line

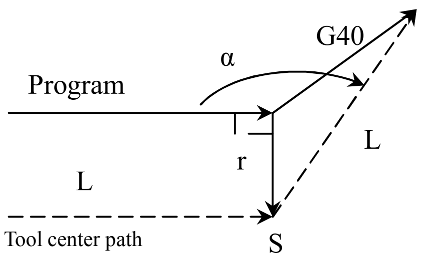


(b) Move along the outer side of the corner (90°≤α<180°)

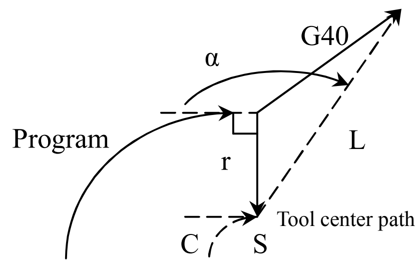
The tool path has 2 types (A and B) when the compensation is started or cancelled, depending on Position Parameter NO: 40#0:

A

Straight line - Straight
line

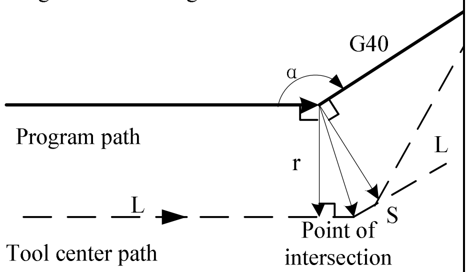


Arc - straight line

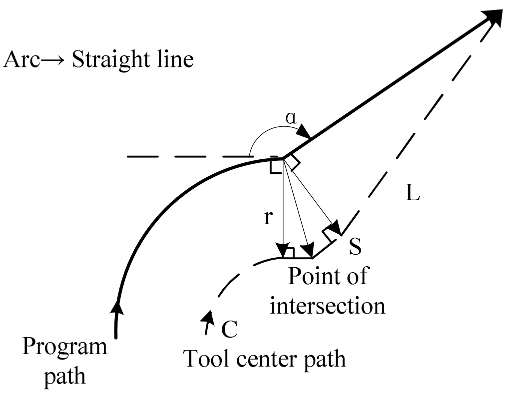


B

Straight line → Straight line



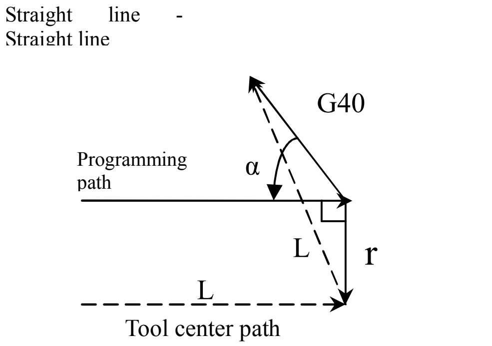
Arc→ Straight line

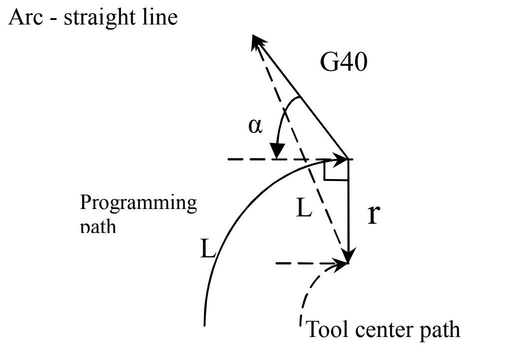


(c) Move along the outer side of the corner which is an acute angle (α<90°)

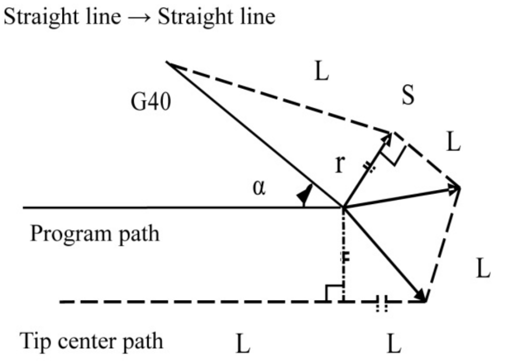
The tool path has 2 types (A and B) when the compensation is started or cancelled, depending on Position Parameter NO: 40#0:

A





B



v

(d) Move along the outer side of the corner which is an acute angle of less than 1°, straight line → straight line. (α<1°)

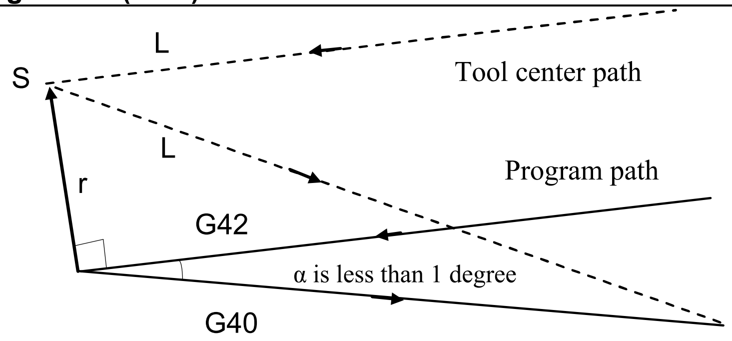


Fig. 4-7-3-5

5. Change of compensation direction in compensation mode

The tool radius compensation G code (G41 and G42) determines the compensation direction. Compensation symbols are shown as follows:

Table 4-7-3-1

Compensation symbols G code	+	-
G41	Left compensation	Right compensation
G42	Right compensation	Left compensation

In special cases, the compensation direction can be changed in the compensation mode. However, it cannot be changed in the start program segment and subsequent segments. For change of compensation direction, there is no difference between inner side and outer side. The following compensations are assumed to be positive.

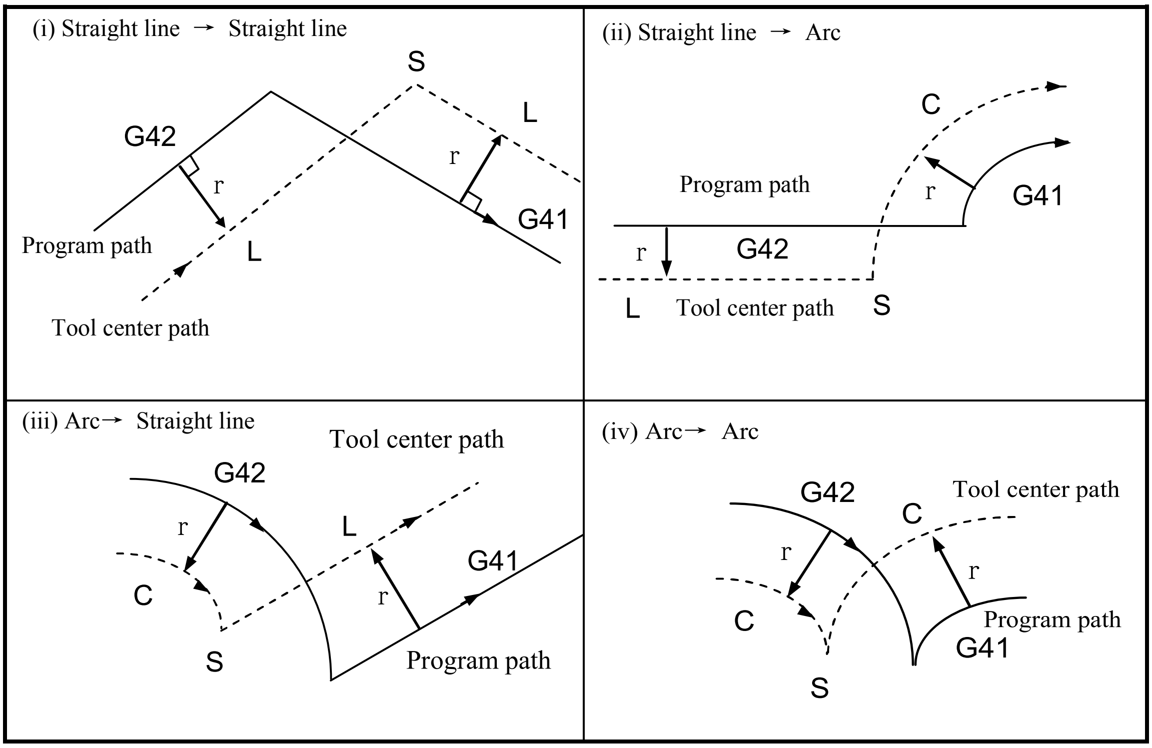


Fig. 4-7-3-6

(v) If the compensation is performed normally, but there is no intersection

When G41 and G42 are used to change the offset direction from Program Segment A to Program Segment B, if the intersection of compensation paths is not required, a vector perpendicular to Program Segment B will be made at the starting point of Program Segment B.

(1) Straight line - straight line

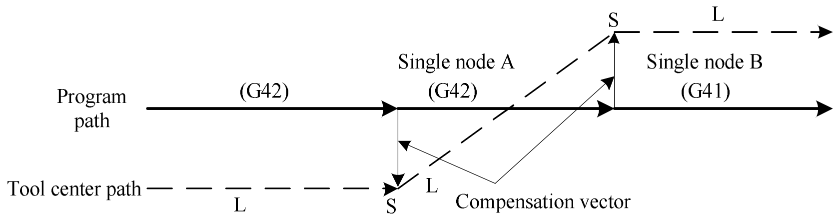


Fig. 4-7-3-7

(2) Straight line - arc

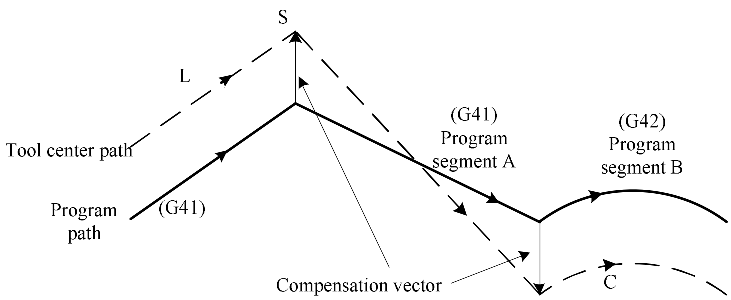


Fig. 4-7-3-8

(3) Arc - arc

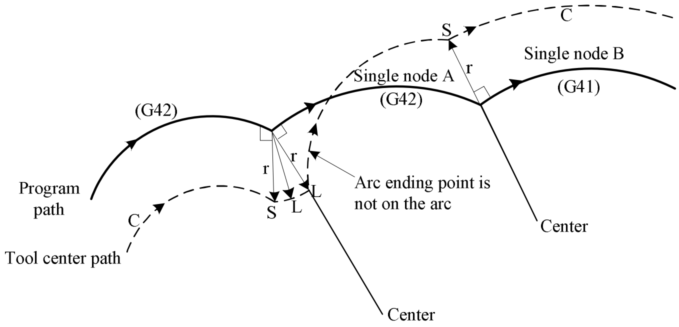


Fig. 4-7-3-9

(vi) When the tool radius compensation makes the tool center path longer than one week, the following conditions will usually not occur. However, when G41 and G42 are changed, the following conditions may occur:

Arc - arc (straight line - arc) The system will give an alarm when the tool compensation direction is changed. When the tool number is D0, the alarm prompts that the arc code cannot cancel the tool compensation!

Straight line - straight line can change the tool compensation direction.

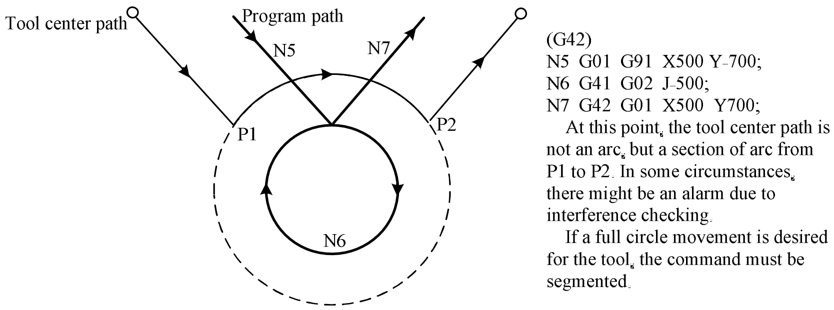


Fig. 4-7-3-10

6. Temporary compensation cancellation

In the compensation mode, Position Parameter NO: 40#2 can be used to determine whether the compensation will be temporarily cancelled at the intermediate point when G28 or G30 is specified.

Please see the detailed description of compensation cancellation and compensation start for detailed method of this operation.

a) Automatic return to reference point (G28)

In the compensation mode, if G28 is specified, the compensation will be cancelled at the intermediate point, and the compensation mode will be automatically restored upon return to the reference point.

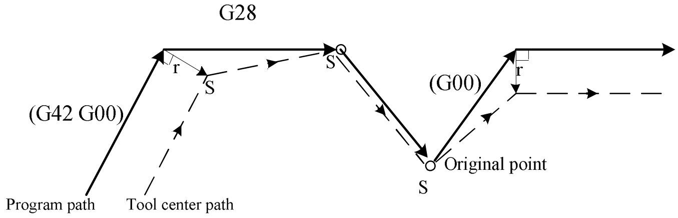


Fig. 4-7-3-11

b) Automatic return from reference origin (G29)

In the compensation mode, if G29 is specified, the compensation will be cancelled at the intermediate point, and the compensation mode will be automatically restored during return to the point specified by G29.

In case of immediate command after G28:

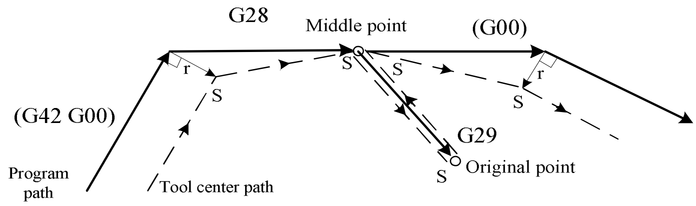


Fig. 4-7-3-12

In case of non-immediate command after G28:

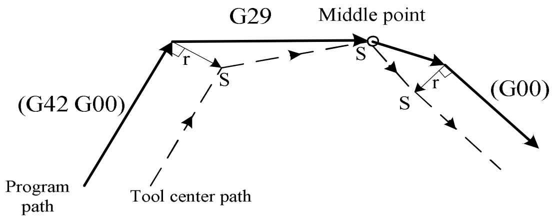


Fig. 4-7-3-13

7. Tool radius compensation in compensation mode (G code)

In the compensation mode, when the tool radius compensation G code (G41, G42) is specified, it will generate a vector at right angle to the front program segment with respect to the movement direction, regardless of the inner or outer side of machining. However, if such a G code is specified in the arc code, the correct arc cannot be obtained.

When changing the compensation direction with the tool radius compensation G code (G41, G42), please refer to (5).

Straight line - straight line

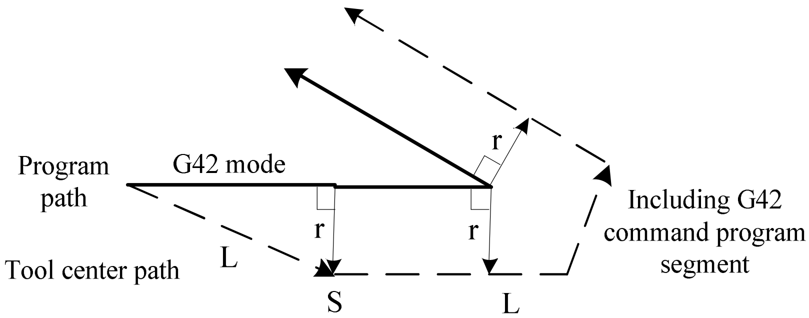


Fig. 4-7-3-14

Arc - straight line

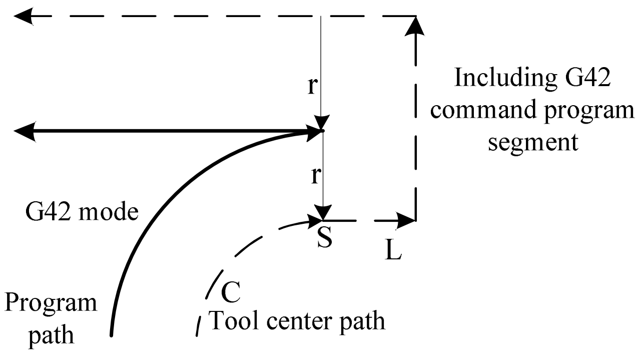


Fig. 4-7-3-15

8. Code segments where the tool does not move

The followings are program segments where the tool does not move. In these segments, the tool will not move even if the tool radius compensation mode is effective.

(1) M05; …………………… M code output
(2) S21; .... S code output
(3) G04 X10; .... Pause
(4) (G17) Z100;.................. No movement code in the compensation plane
(5) G90;...........................only G code
(6) G01 G91 X0;...............No movement

a) Code at the beginning of compensation

In case of no movement of the program segment for the start of cutting, the system will generate an activity to start cutting in the next segment of movement code.

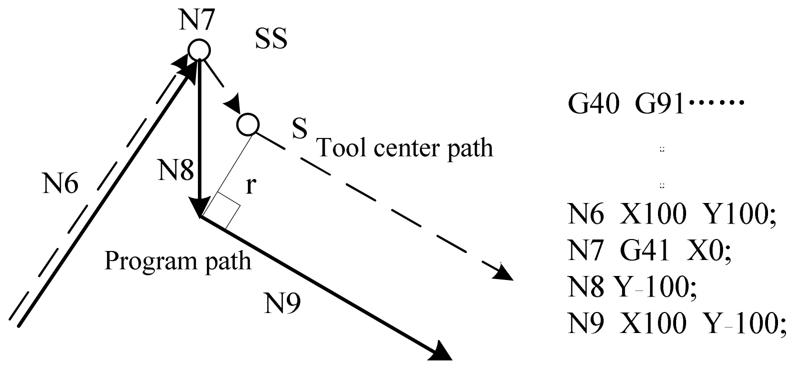


Fig. 4-7-3-16

b) When the compensation mode is specified

When only one program segment without tool movement is specified in the compensation mode, the vector and tool center path remain unchanged. (Refer to Item (3) for compensation mode) This program segment will be executed at the stop point of a single program segment.

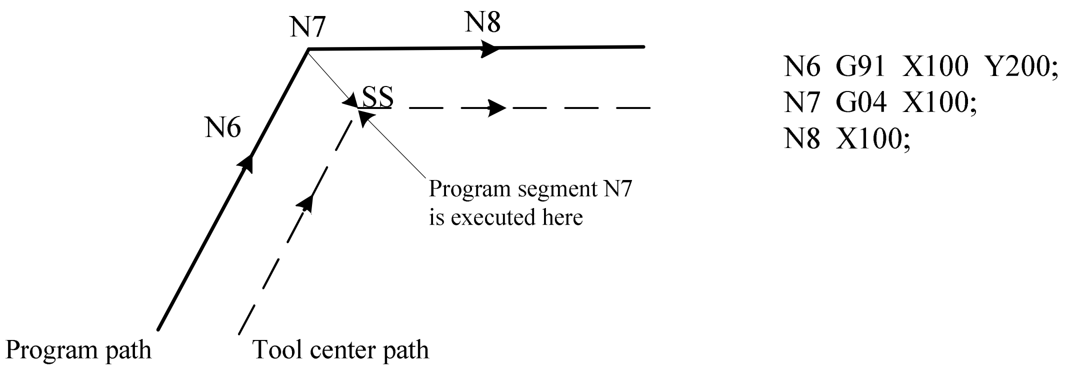
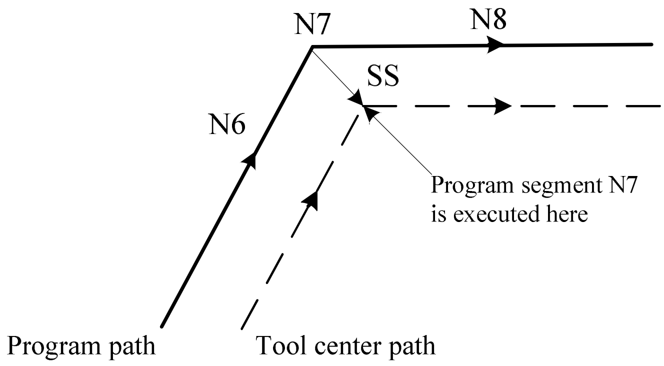


Fig. 4-7-3-17

However, in case of no movement of the program segment, even if only one program segment is specified, the tool moves in the same manner as two or more segments without tool movement.



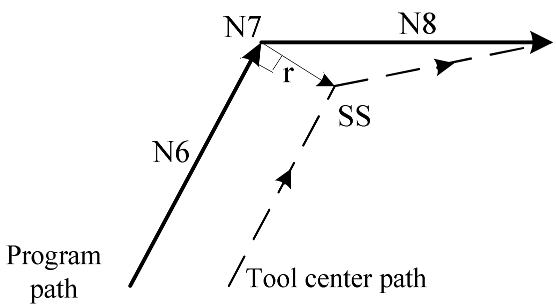
N6 G91 X100 Y200;
N7 X0;
N8 X100;

Fig. 4-7-3-18

Note: The above program segment runs under the condition of G1, G42, and the track in case of G0 is not consistent with the figure.

c) When specified together with compensation cancellation

In case that the program segment, specified together with compensation cancellation, features no tool movement, it will generate a vector with its length as compensation and perpendicular to the movement direction of the front program segment, and this vector will be cancelled by the next movement command.



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

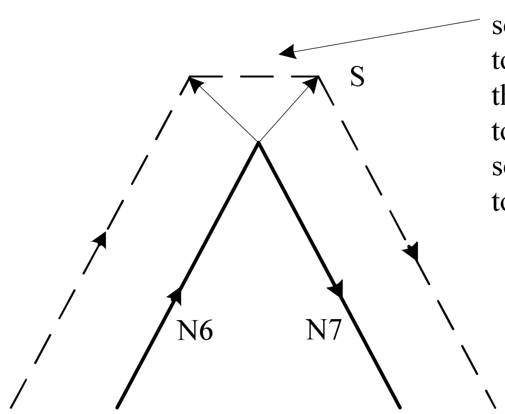
Fig. 4-7-3-19

9. Corner movement

If more than two vectors are generated at the end of a program segment, the tool moves from one vector line to another vector. Such movement is called corner movement.

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

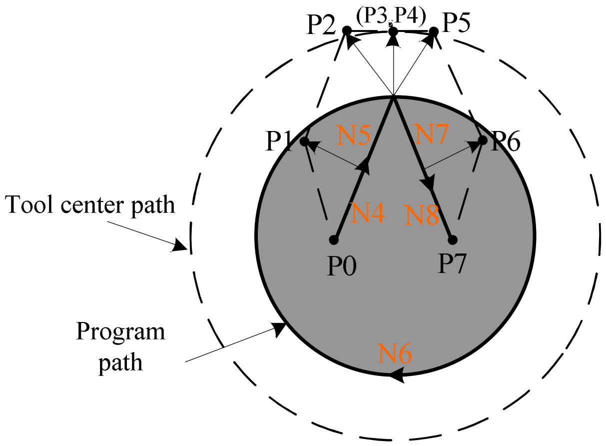
If these vectors are inconsistent, a movement along the corner is produced. This movement belongs to the front program segment.



These movements belong to the program segment N6. Therefore the feed rate equals to the one of the program segment N6. If the program segment N6 is G00 mode, the tool moves at fast feed rate; if the program segment N6 is G01, G02, G03 mode, the tool moves at cutting feed rate.

Fig. 4-7-3-20

However, if the path of the next segment exceeds a semicircle, the above functions are not executed. The reasons include:



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 shown as follows:
P0 → P1 → P2 → P3 (arc) → P4 → P5 → P6 → P7
However, if the distance between P2 and P3 is ignored, P3 will be ignored. The tool path is shown as follows:
P0 → P1 → P2 → P4 → P6 → P7 The arc cutting of Program Segment N6 is ignored.

10. Interference check

Excessive cutting of the tool is called "interference". Interference can be used to pre-check excessive tool cutting. The system will give an alarm if an interference is detected in the syntax check after the program is loaded in. It is set by Position Parameter **NO: 41#6** whether an interference check is performed during radius compensation.

Basic conditions for interference:

(1) The movement distance of the program segment for tool radius compensation is smaller than the tool radius.

(2) The tool path direction is different from the program path direction. (The angle between the paths is between 90° and 270°).

(3) During arc machining, in addition to the above conditions, the angle between the start point and the end point of the tool center path is greatly different from the angle between the start point and the end point of the program path (180° or more).

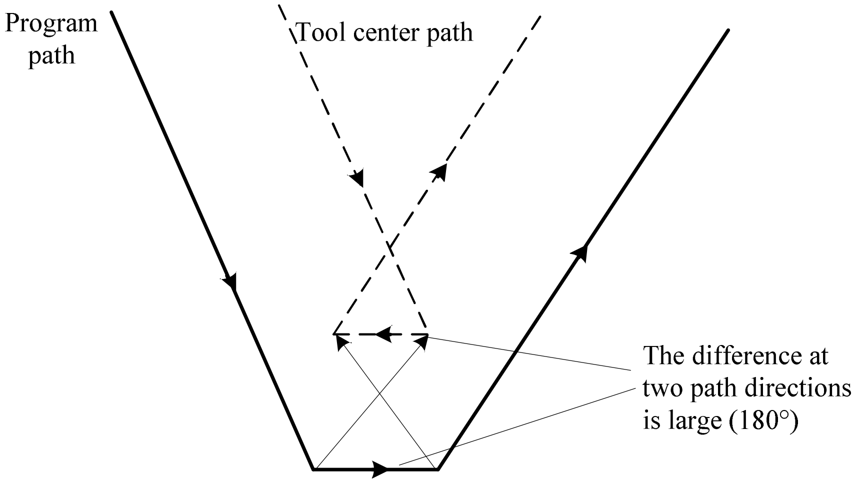


Fig. 4-7-3-22

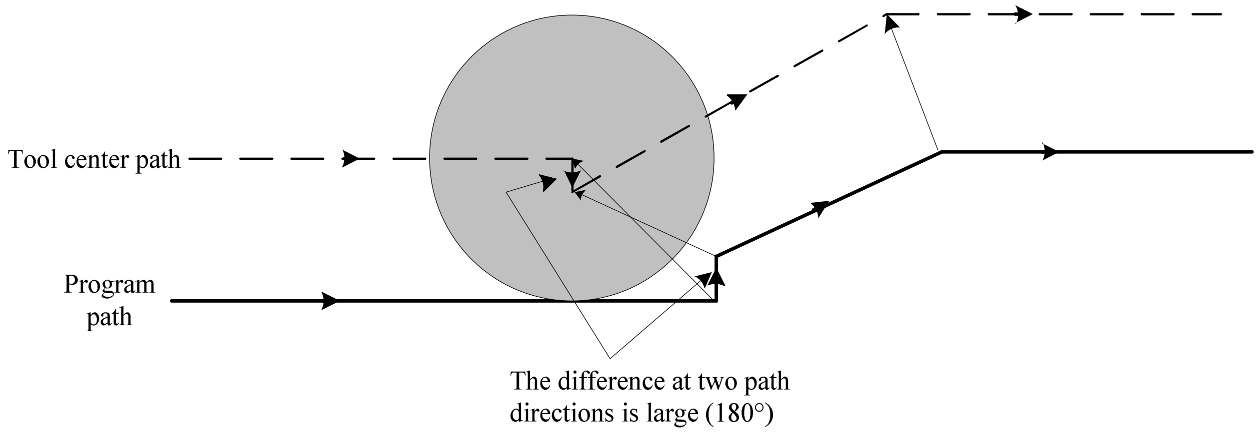


Fig. 4-7-3-23

11. Manual operation

Please see Part II Operation Instructions for manual operation related to the tool radius compensation.

12. General considerations about compensation

a) Specify the compensation quantity

The compensation quantity is specified by the D code. Once specified, the D code remains valid until another D code is specified, or the compensation is cancelled. The D code is not only used to specify the tool radius compensation quantity, but also used to specify the tool offset value.

b) Change the compensation quantity

Normally, when the tool is replaced, the compensation quantity must be changed in the compensation cancelled mode. If changed in the compensation mode, the new compensation quantity will be calculated at the end point of the program segment.

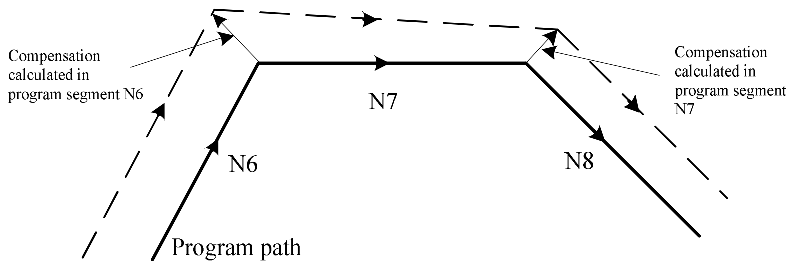


Fig. 4-7-3-24

c) Positive or negative compensation quantity and tool center path

If the compensation quantity is a negative value (-), G41 and G42 in the program will be interchanged. If the tool center moves along the outer side of the workpiece, it will move along the inner side and vice versa, as shown in the following example.

In general, the compensation quantity is positive (+) during programming. When the tool path is programmed as shown in Figure (a), if the compensation quantity is negative (-), the tool center will move as shown in Figure (b) and vice versa. Therefore, the same program can be cut into a male or female form, and the gap between them can be adjusted by selection of compensation quantity.

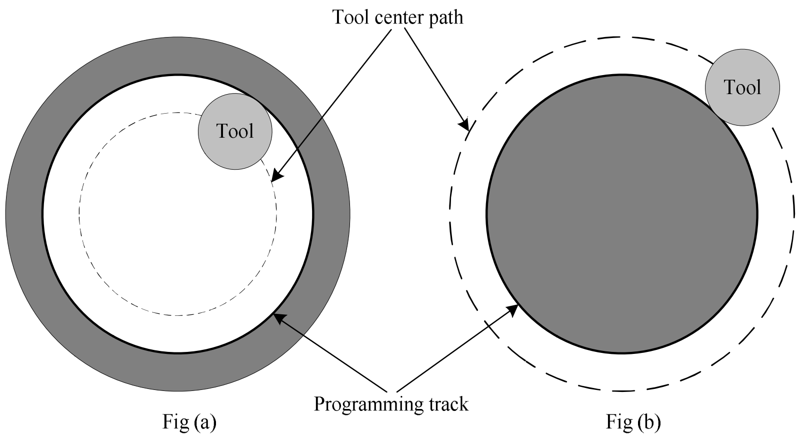


Fig. 4-7-3-25

d) Overcuting with tool radius compensation

(1) In case of inner machining with an arc smaller than the tool radius

When the corner radius is smaller than the tool radius, since inner compensation of the tool will cause excessive cutting, an interference alarm will be generated before the program is executed and the system will stop working.

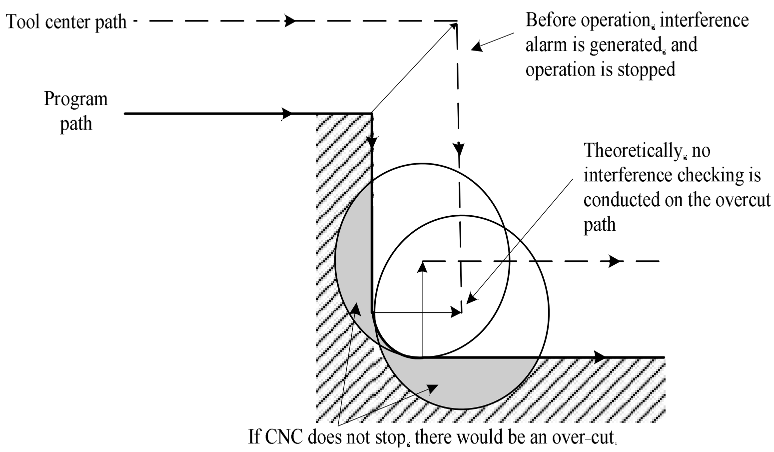


Fig. 4-7-3-26

(2) In case of machining with a groove smaller than the tool radius

When a groove smaller than the tool radius is used for machining, excessive cutting will occur because the tool radius compensation forces the tool center path to move in the opposite of the program path.

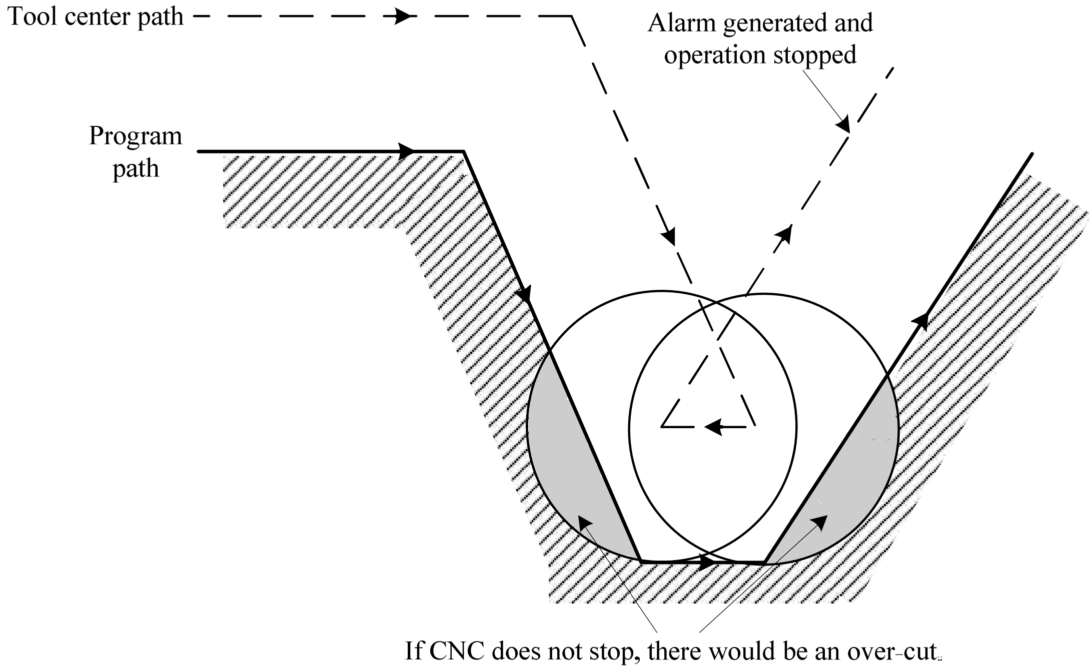


Fig. 4-7-3-27

(3) In case of machining with a segment gap smaller than the tool radius

If there is a segment gap smaller than the tool radius in the program, when arc machining is used to specify the machining in this segment gap, the tool center path of normal compensation will be in the opposite to the program direction. At this moment, the initial vector is ignored and the tool moves straight to the second vector. Single-segment execution stops here. In case of machining not in the single-segment mode, the automatic operation will continue. If the segment gap is a straight line, no alarm will be generated and correct cutting will be performed. However, there will be uncut portions.

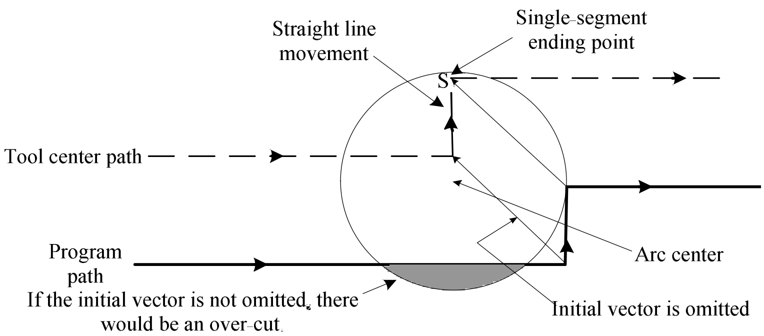


Fig. 4-7-3-28

Tool radius compensation starts and moves in the Z axis

Generally, at the beginning of machining, when the tool radius compensation becomes valid, the tool will move along the Z axis at a certain distance from the workpiece. In the above case, if you want to divide the movement along the Z axis into rapid feed and cutting feed, please refer to the following two programs:

In case Program Segment N3 (movement code in the Z axis)

Divided as 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;

When N3 is executed,
N6 also enters the buffer zone. Use the relationship between them. The correct compensation is shown in right figure.

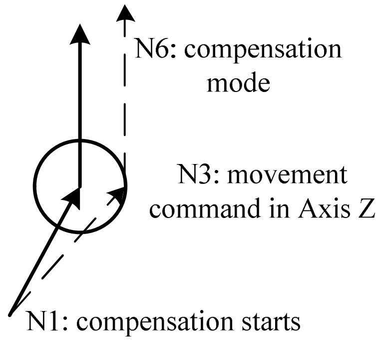


Fig. 4-7-3-29

4.7.4 Corner Offset Circular Interpolation (G39)

Format: G39

Function: During the tool radius compensation, G39 can be used to specify the corner offset circular interpolation. The radius of corner compensation is equal to the compensation value. Position Parameter **NO: 41#5** is used to determine whether the corner arc is valid in radius compensation.

Description:

1. When G39 is specified, a corner circular interpolation with its radius equal to the compensation value can be performed.
2. G41 or G42 before this code determines whether the arc is clockwise or counterclockwise, and G39 is a non-modal G code.
3. In case of programming with G39, an arc is formed at the corner, so the vector at the end point of the arc is perpendicular to the start point of the next program segment. As shown in the figure:

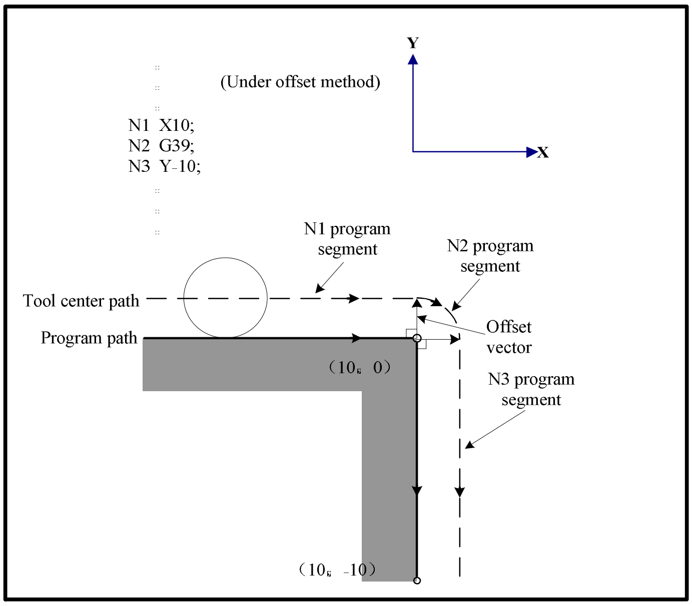


Figure 4-7-4-1 G39

4.7.5 Input of Tool Compensation Value and Compensation Number With 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 absolute value code (G90) mode.
Tool compensation value in incremental value code (G91) mode, which is added to the value of the specified tool compensation number (the sum is the tool compensation value).

Description: Effective input range of tool compensation values:
Geometric compensation: Input in mm -999.999 mm~+999.999 mm; input in inch -39.3700 inch~+39.3700 inch.
Wear compensation: Input in mm -400.000 mm~+400.000 mm (take the No. 291 data parameter setting value); input in inch -39.3700 inch~+39.3700 inch (take 1/25.4 of the No. 291 data parameter setting value).

Note: The maximum value of wear compensation is limited by Data Parameter P291.

4.8 Feed (G Code)

4.8.1 Feed Mode (G64/G61/G63)

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

Function:
Exact stop mode G61: once specified, this function remains valid until G62, G63 or G64 is specified. The tool decelerates in the end point of the program segment to perform in-position check and then executes the next segment.
Tapping mode G63: once specified, this function remains valid until G61, G62 or G64 is specified. The tool executes the next program segment without decelerating at the end point of the current segment. When G63 is specified, the feedrate override and feed hold are invalid.
Cutting mode G64: once specified, this function remains valid until G61, G62 or G63 is specified. The tool executes the next program segment without decelerating at the end point of the current segment.

Description:
1. No parameter format.
2. G64 is the default feed mode of the system. The tool does not decelerate at the end point of the program segment and directly executes the next segment.
3. The in-position check in the exact stop mode is designed to check if the servo motor is placed in within the specified range.
4. In exact stop mode, cutting mode and tapping mode, the tool moves along different paths. See Figure 4-8-1-1 below for details.

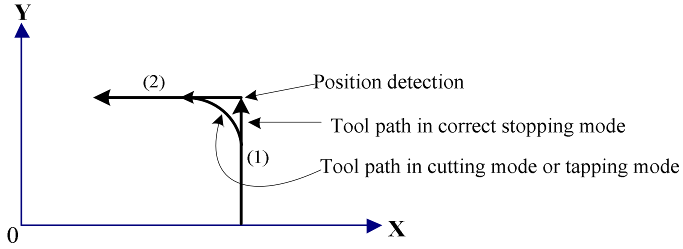


Figure 4-8-1-1 Tool path from Program Segment 1 to Program Segment 2

4.8.2 Automatic Corner Override (G62)

Format: G62

Function: Automatic corner override mode G62: once specified, this function remains valid until G61, G63 or G64 is specified. During tool radius compensation, when the tool moves along the inner corner, the cutting feed rate is multiplied to suppress the cutting output per unit time, so that good surface accuracy can be achieved.

Description:

1. During tool radius compensation, the tool automatically decelerates to reduce the load on the tool and outputs smooth surfaces when moving in the inner corner and inner arc area.
2. It is set by position parameters NO: **16#7** whether the automatic corner override function is valid. It is set by position parameters NO: **15#2** to control the automatic corner deceleration function (0: angle control, 1: speed difference control).
3. When G62 is specified and the tool radius compensation function is applied and the inner corner is machined, the feedrate is automatically adjusted at both ends of the corner. There are four types of inner corner as shown in Figure 4-8-2-1. In the figure: 2° ≤ θ ≤ θp ≤ 178°. θp is set by Data Parameter **P144**.

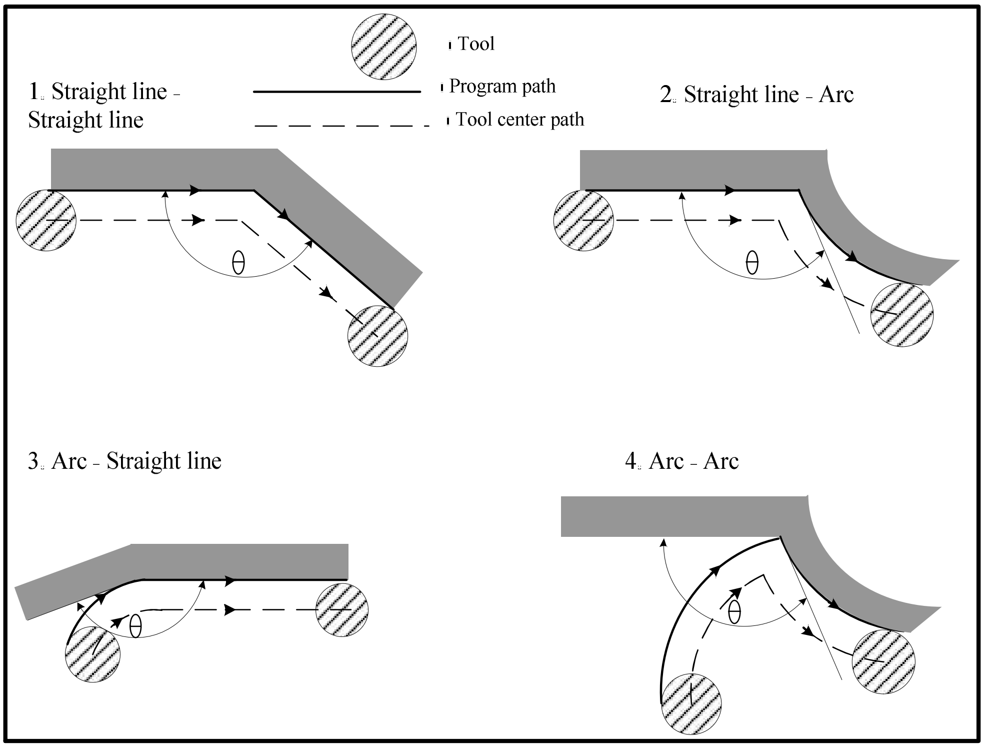


Fig. 4-8-2-1

4. When the corner is determined as inner corner, the feedrate override is performed before and after the inner corner. The distance at which the feedrate override is performed is Ls

and Le, which is the distance from the point on the tool center path to the corner. As shown in Figure 4-8-2-2, where Ls+Le≤2mm.

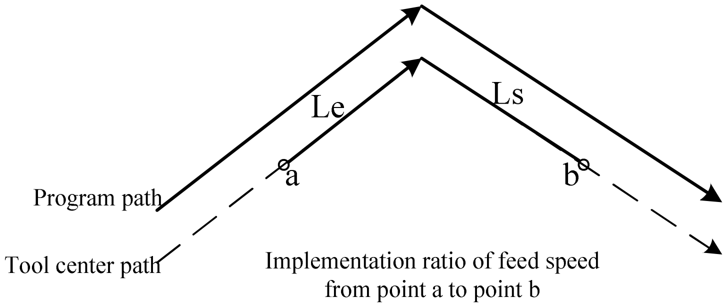


Figure 4-8-2-2 Straight line to straight line

5. When the programming track includes two arcs, if the start point and the end point are in the same quadrant or in adjacent quadrants, the feedrate is multiplied, and Data Parameter P145 is used to control the minimum feedrate of automatic corner deceleration, as shown in Figure 4- 8-2-3.

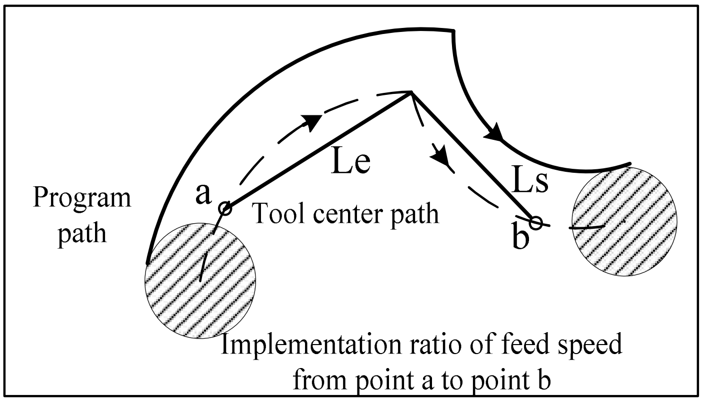


Figure 4-8-2-3 Arc to arc

6. Given that a program includes straight line to arc and also arc to straight line, as shown in Figure 4-8-2-4, the feedrate is multiplied from Point a to Point b and from Point c to Point d.

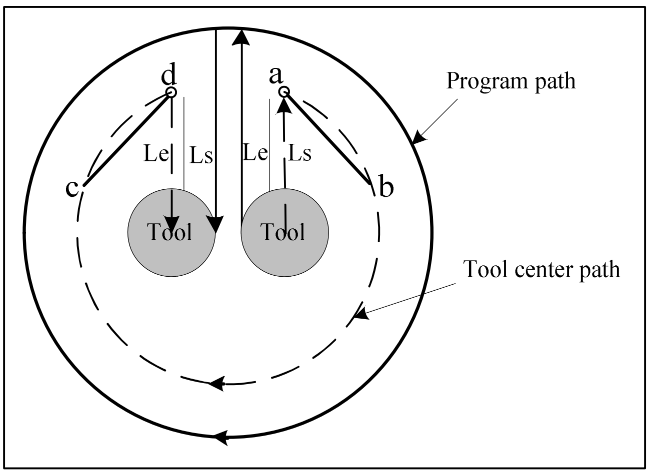


Figure 4-8-2-4 Straight line to arc, arc to straight line

Restrictions:

1. During acceleration/deceleration before interpolation, the inner corner is invalid.
2. If there is a program segment for start of cutting before the corner or a program segment

including G41 or G42 after the corner, the inner corner override is invalid.

3. If the offset is zero, the inner corner is not performed.

4.9 Macro Function (G Code)

4.9.1 User Macro Program

A certain function realized by a set of codes is pre-stored in the memory like subprograms, and a code is used to represent these functions. These functions can be realized just by writing the representative codes in the program. This set of codes is called user macro program itself, and the representative code is called “user macro code”. Sometimes, user macro program itself is also known as macro program, and user macro code as macro program calling code.

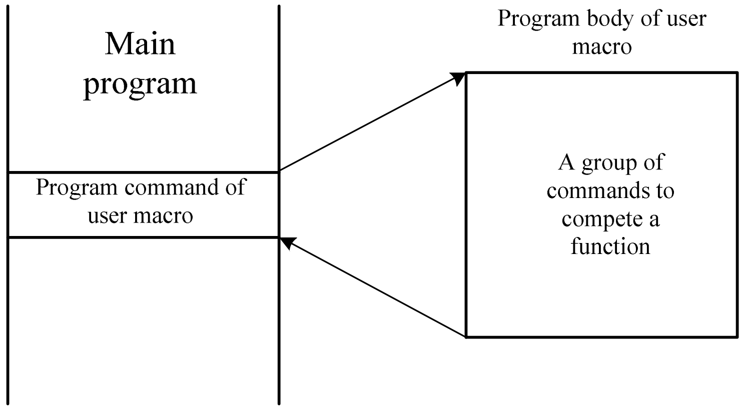


Fig. 4-9-1-1

Variables can be used in the user macro program itself. Operations can be performed among variables which can be assigned with macro code.

4.9.2 Macro Variables

In a user macro program, general CNC commands can be used, as well as variables, operation and jump codes.

A user macro program starts with program number and ends with M99.

```
O0066;                       Program No.
G65  H01  ……;                Operation command
G90  G00  X#101  ……;         Use variable CNC
                             command
……
……
……
G65  H82  ……;                Transferring command
……
……
M99;                         Program body of user
                             macro completed
```

Figure 4-9-2-1 Composition of user macro program itself

1. How to use variables

Variables can be used to specify the parameter values in user macro program itself. The value of a variable can be assigned by the main program or set by LCD/MDI, or assigned with the result figured out when the user macro program itself is executed.

Several variables can be used, which are distinguished by variable number.

(1) Representation of variables

Use # followed by a variable number to represent the variable; the format is as follows:

#i (i = 1， 2， 3， 4 ……)

(Example) #5， #109， #1005

(2) Reference to variables

A variable can replace the value after a parameter value.

(Example) F#103   In case of #103=15, it is the same as the F15 command.

G#130   In case of #130=3, it is the same as G3.

Note: 1. Reference to variables is not applicable to the parameters O and N (program number and sequence number). Programming with O#100, N#120 is not allowed.

2. When exceeding the maximum code value specified by the parameter, the variable cannot be used. In case of #30=120, M#30 exceeds the maximum code value.

3. Display and setting of variable values: The value of a variable can be displayed on the LCD screen, and can also be set in the MDI mode.

2. Classification of variables

Variables can be classified into empty variables, local variables, common variables, and system variables, which have different purposes and properties.

(1) Empty variables #0: (The variable is always empty and no value can be assigned to the variable)

(2) Local variables #1 - #50: local variables can only be used to store data in macro programs; Position Parameter NO: 52#7 can be used for reset setting or determine whether to delete data after emergency stop. When a macro program is called, the local variables are assigned by independent variables.

(3) Common variables #100 - #199, #500 - #999: Position parameter NO: 52#6 can be used for reset settings or determine whether to eliminate the common variables #100 - #199 after emergency stop.

Common variables can be shared in the main program and all user macro programs called by the main program. That is, the variable #i used in a certain user macro program is the same as #i used in other macro programs. Therefore, the common variable #i as a result of operation in a certain macro program can be used in other macro programs.

If there are no rules about the purpose of common variables in the system, these variables can be freely used by users.

Table 4-9-2-1

Variable number	Variable type	Function
＃100～＃199	Common variable	Cleared when the power is cut off and all reset to "empty" when power is on.
＃500～＃999		The data is saved in the file and will not be lost even in case of outage

(4) System variables: System variables are used to read and write changes to various data during CNC runtime, As shown below:

1) Interface input signal   #1000 --- #1015 (Read by bit the signal input by PLC to the system, i.e. G signal)

#1032 (Read by byte the signal input by PLC to the system, i.e. G signal)

2) Interface output signal   #1100 --- #1115 (Write by bit the signal output from the system to PLC, i.e. F signal)

#1132 (Write by byte the signal output from the system to PLC, i.e. F signal)

3)Tool length compensation value   #1500 --- #1755 (readable and writable)

4) Length wear compensation value   #1800 --- #2055 (readable and writable)

Volume I Programming instructions

5) Tool radius compensation value #2100 --- #2355 (readable and writable)
6) Radius wear compensation value #2400 --- #2655 (readable and writable)
7) Alarm #3000
8) User Data Table #3500 --- #3755 (read only, not writable)
9) Modal information #4000 --- #4030 (read only, not writable)
10) Position information #5001 --- #5030 (read only, not writable)
11) Workpiece zero offset #5201 --- #5235 (readable and writable)
12) Additional workpiece coordinate system #7001 --- #7250 (readable and writable)

3. Detailed description of system variables

1) Modal information

Table 4-9-2-2

Variable number	Function	Group number
#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	

Variable number	Function	Group number
#4028	N	
#4029	O	
#4030	P (additional workpiece coordinate system selected currently)	

Note 1: The **P** code is the currently selected additional workpiece coordinate system.
Note 2: When G#4002 is executed, the value obtained in #4002 is 17, 18, or 19.
Note 3: Modal information can be read only and cannot be written.

2) Current position information

Table 4-9-2-3

Variable number	Position information	Related coordinate system	Read operation during movement	Tool offset value
#5001	X-axis program segment end position (ABSIO)	Workpiece coordinate	OK	Tool tip position (which is specified by the program) not considered
#5002	Y-axis program segment end position (ABSIO)			
#5003	Z-axis program segment end position (ABSIO)			
#5004	4th-axis program segment end position (ABSIO)			
#5006	X-axis program segment end position (ABSMT)	Machine tool coordinate system	No.	Consider the tool reference point position (machine tool coordinate)
#5007	Y-axis program segment end position (ABSMT)			
#5008	Z-axis program segment end position (ABSMT)			
#5009	4th-axis program segment end position (ABSMT)			
#5011	X-axis program segment end position (ABSOT)	Workpiece coordinate		
#5012	Y-axis program segment end position (ABSOT)			
#5013	Z-axis program segment end position (ABSOT)			
#5014	4th-axis program segment end position (ABSOT)			
#5016	X-axis program segment end position (ABSKP)		OK	
#5017	Y-axis program segment end position (ABSKP)			
#5018	Z axis program segment end position (ABSKP)			
#5019	4th-axis program segment end position (ABSKP)			
#5021	X-axis tool length compensation value		No.	
#5022	Y-axis tool length compensation			

	value			
#5023	Z-axis tool length compensation value			
#5024	4th-axis tool length compensation value			
#5026	X-axis servo position compensation			
#5027	Y-axis servo position compensation			
#5028	Z-axis servo position compensation			
#5029	4th-axis servo position compensation			

Note 1: ABSIO: Coordinate values of the end point of the previous program segment in the workpiece coordinate system.
Note 2: ABSMT: Current position in the machine tool coordinate system.
Note 3: ABSOT: Current position in the workpiece coordinate system.
Note 4: ABSKP: In the workpiece coordinate system, the position where the skip signal is valid in Program Segment G31.

3) Workpiece zero offset and additional zero offset:

Table 4-9-2-4

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

4. Local variables

Correspondence between address and local variables:

Table 4-9-2-5

Address of independent variables	Local variable number	Address of independent variables	Local variable number
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: a variable is assigned in such a form of English letter followed by a value. Except for G, L, O, N, H and P, the rest of 20 English letters can be used to assign values to independent variables. Each letter is used for one assignment, from A-B-C-D… to X-Y-Z. Assignments do not have to be done in alphabetical order, and the addresses that are not assigned can be omitted.

Note 2: G65 must be specified before any independent variable is used.

5. Considerations about user macro program itself

1) Input with # Key
   Press down # Key to input # following G, X, Y, Z, R, I, J, K, F, H, M, S, T, P or Q.
2) In the MDI state, operation or jump codes can also be specified.
3) H, P, Q, and R of operation or jump code are used as parameters of the G65 command before and after G65.
   H02 G65 P#100 Q#101 R#102;   Correct.
   N100 G65 H01 P#100 Q10;   Correct
4) The input range of a variable cannot exceed fifteen significant digits. The operation result cannot exceed an integer of nine digits, and the manual input range of the variable is eight significant digits.
5) The result of variable value calculation can be a decimal with an accuracy of 0.0001. Only H11 (or operation), H12 (and operation), H13 (non-operation) and H23 (remainder operation) will ignore the fractional part of a variable in the calculation process, and other operations will not round off the decimal point for operation.
   Example:
   #100 ＝ 35， #101 ＝ 10， #102 ＝ 5
   #110 ＝ #100÷#101   (＝3.5)
   #111 ＝ #110×#102   (＝17.5)
   #120 ＝ #100×#102   (＝175)
   #121 ＝ #120÷#101   (＝17.5)
6) The execution time of operation and jump codes varies depending on specific conditions, generally 10 ms on average.
7) When a variable value is undefined, this variable will become an “empty” variable. Variable #0 is always an empty variable. It can be read only and cannot be written.
   a. Reference to variables
   When an undefined variable is referenced, the address itself is also ignored.
   For example:

When the value of Variable #1 is 0, and the value of Variable #2 is empty, the execution result of G00X#1 Y#2 is G00X0;

b. Operation

Except for assignments with <empty>, <empty> is the same as 0 in the other cases.

Table 4-9-2-6

When #1=<empty>	When #1=0
#2=#1 ↓ #2=<empty>	#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 expression

<Empty> in EQ and NE is different from 0.

Table 4-9-2-7

When #1=<empty>	When #1=0
#1 EQ #0 ↓ Correct	#1 EQ #0 ↓ Incorrect
#1 NE #0 ↓ Incorrect	#1 NE #0 ↓ Correct
#1 GE #0 ↓ Correct	#1 GE #0 ↓ Correct
#1 GT #0 ↓ Incorrect	#1 GT #0 ↓ Incorrect

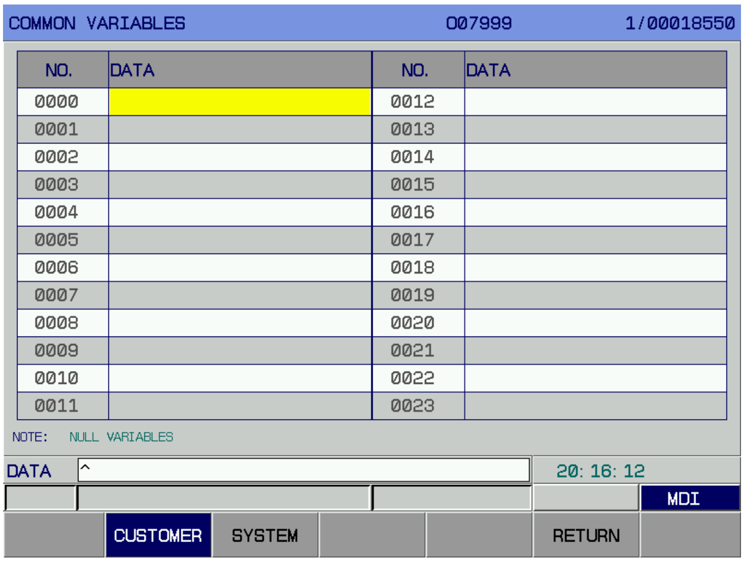


Fig. 4-9-2-2

When the variable value is blank, it is an empty variable.

4.9.3 User Macro Program Call

When G65 is specified, the user macro program specified by address P is called and the data is passed to the user macro program itself via independent variables.

The format is as follows:

G65 P □□□□□L□□□□ <specified independent variable>;

Number of calls

Program number of the called macro program itself

After G65, the program number of user macro program is specified by the address P, the number of calls is specified by L, and the data is passed to the macro program via independent variables.

When repeating is required, the number of repetitions from 1 to 9999 is specified after the address L, and when L is omitted, the default number is 1.

The independent variable can be used to assign its value to the corresponding local variable.

Note 1: An alarm is generated when the subprogram number specified by the address P cannot be retrieved (PS 078).

Note 2: The subprograms No. 90000~99999 are the system retained programs. When the user calls such subprograms, the system can execute the subprogram content, but the cursor will stay in the G65 program segment, and the program interface will always display the main program content. (The subprogram contents can be displayed by modifying Position Parameter N0: 27#4.)

Note 3: Macro programs cannot be called in DNC mode.

Note 4: Macro program calls can be nested up to five levels.

4.9.4 User Macro Program - Function A

1. General form:

G65 Hm P#i Q#j R#k;

m: 01 to 99 indicate the function of operation code or jump code.

#i: Variable name in which the operation result is stored.

#j: Variable Name 1 to perform the operation. It can also be a constant. Directly represented by the constant without #.

#k: Variable Name 2 to perform the operation. It can also be a constant.

Meaning: #i = #j ○ #k

Operational symbol, specified by Hm

(Example) P#100 Q#101 R#102……#100 = #101 ○ #102 ;

P#100 Q#101 R15 ……#100 = #101 ○ 15 ;

P#100 Q-100 R#102……#100 = -100 ○ #102

The H code specified by G65 has no effect on the selection of depth offset.

G code	H code	Function	Definitions
G65	H01	Assignment	#i = #j
G65	H02	Addition	#i = #j + #k
G65	H03	Subtraction	#i = #j - #k
G65	H04	Multiplication	#i = #j × #k
G65	H05	Division	#i = #j ÷ #k
G65	H11	Logical add (OR)	#i = #j OR #k
G65	H12	Logic multiplication (AND)	#i = #j AND #k
G65	H13	Exclusive OR	#i = #j XOR #k
G65	H21	Square root	$\#i = \sqrt{\#j}$
G65	H22	Absolute value	$\#i = \|\#j\|$
G65	H23	Take the remainder	#i=#j-trunc(#j ÷ #k)×#k
G65	H26	Compound multiplication and division	#i = (#i × #j) ÷ #k
G65	H27	Compound square root	$\#i = \sqrt{\#j^2+\#k^2}$
G65	H31	Sine	#i = #j×SIN(#k)
G65	H32	Cosine	#i = #j×COS(#k)
G65	H33	Tangent	#i = #j×TAN(#k)
G65	H34	Inverse tangent	#i = ATAN(#j/#k)
G65	H80	Unconditional transfer	转向N
G65	H81	Conditional transfer 1	IF #j = #k， GOTO N
G65	H82	Conditional transfer 2	IF #j ≠ #k， GOTO N
G65	H83	Conditional transfer 3	IF #j ＞ #k， GOTO N
G65	H84	Conditional transfer 4	IF #j ＜ #k， GOTO N
G65	H85	Conditional transfer 5	IF #j ＞ #k， GOTO N
G65	H86	Conditional transfer 6	IF #j ≤ #k， GOTO N
G65	H99	Alarm	

Fig. 4-9-4-1

2. Operation code:

1) Assignment of variables: # I = # J

G65 H01 P#I Q#J;

(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) Addition: # I = # J+# K

G65 H02 P#I Q#J R#K;

(Example) G65 H02 P#101 Q#102 R15; (#101 = #102+15)

3) Subtraction: # I = # J- #K

G65 H03 P#I Q#J R# K;

(Example) G65 H03 P#101 Q#102 R#103; (#101 = #102-#103)

4) Multiplication: # I = # J×# K

G65 H04 P#I Q#J R#K;

(Example) G65 H04 P#101 Q#102 R#103; (#101 = #102×#103)

5) Division: # I = # J÷# K

G65 H05 P#I Q#J R#K;

(Example) G65 H05 P#101 Q#102 R#103; (#101 = #102÷#103)

6) Logical Add (OR): # I = # J.OR. # K

G65 H11 P#I Q#J R#K;

(Example) G65 H11 P#101 Q#102 R#103; (#101 = #102.OR. #103)

7) Logical multiplication (AND): # I = # J.AND. # K

G65 H12 P#I Q#J R#K;

(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;

(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；

(Example) G65 H21 P#101 Q#102 ； （#101=$\sqrt{\#102}$ ）

10) Absolute value: # I = | # J |

G65 H22 P#I Q#J;

(Example) G65 H22 P#101 Q#102; (#101 = | #102 |)

11) Remainder operation: # I = # J-TRUNC(#J/#K)×# K, TRUNC: Round off the fractional part

G65 H23 P#I Q#J R#K;

(Example) G65 H23 P#101 Q#102 R#103; (#101 = #102- TRUNC (#102/#103)×#103)

12) Compound multiplication and division operation: # I =(# I×# J)÷# K

G65 H26 P#I Q#J R# k;

(Example) G65 H26 P#101 Q#102 R#103; (#101 =(#101×# 102)÷#103)

13) Compound square root: # I = $\sqrt{\#j^2+\#k^2}$

G65 H27 P#I Q#J R#K；

(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;

(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;

(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;

(Example) G65 H33 P#101 Q#102 R#103; (#101 = #102•TAN(#103))

17) Arctangent: # I = ATAN(# J /# K) (Unit: °)

G65 H34 P#I Q#J R# K;

(Example) G65 H34 P#101 Q#102 R#103; (#101 =ATAN(#102/#103))

Note 1: The angle variable is measured in degree (°).
Note 2: In each operation, when the required Q and R are not specified, the value is entered as zero.
Note 3: trunc: Rounding operation with the fractional part rounded off.

3. Jump command

1) Unconditional jump

G65 H80 Pn; n: Sequence No.

(Example) G65 H80 P120; (Jump to Program Segment N120)

2) Conditional jump 1 #J.EQ.# K (=)

G65 H81 Pn Q#J R# K; n: Sequence No.

(Example) G65 H81 P1000 Q#101 R#102;

When # 101 = #102, jump to Program Segment N1000, and when #101 ≠ #102, the program is executed in sequence.

3) Conditional jump 2 #J.NE.# K (≠)

G65 H82 Pn Q#J R# K; n: Sequence No.

(Example) G65 H82 P1000 Q#101 R#102;

When # 101 ≠ #102, jump to Program Segment N1000, and when #101 = #102, the program is executed in sequence.

4) Conditional jump 3 #J.GT.# K (>)

G65 H83 Pn Q#J R# K; n: Sequence No.

(Example) G65 H83 P1000 Q#101 R#102;

When #101 > #102, jump to Program Segment N1000, when #101 ≤ #102, the program is

Volume I Programming instructions

executed in sequence.

5) Conditional jump 4 #J.LT.# K (<)

G65 H84 Pn Q#J R# K; n: Sequence No.

(Example) G65 H84 P1000 Q#101 R#102;

When #101<#102, jump to Program Segment N1000, and when #101 ≥ #102, the program is executed in sequence.

6) Conditional jump 5 #J.GE.# K (≥)

G65 H85 Pn Q#J R# K; n: Sequence No.

(Example) G65 H85 P1000 Q#101 R#102;

When # 101 ≥ #102, jump to Program Segment N1000, and when #101<#102, the program is executed in sequence.

7) Conditional jump 6 #J.LE. # K (≤)

G65 H86 Pn Q#J R# K; n: Sequence No.

(Example) G65 H86 P1000 Q#101 R#102;

When #101≤#102, jump to Program Segment N1000, and when #101>#102, the program is executed in sequence.

Note: A variable can be used to specify the sequence number. For example: G65 H81 P#100 Q#101 R#102; when the condition is satisfied, the program will jump to the program segment with its sequence number specified by #100.

4. Logical AND, logical OR and logical negation codes

Example:

G65 H01 P#101 Q3;

G65 H01 P#102 Q5;

G65 H11 P#100 Q#101 Q#102;

In the binary system, 5 is represented as 101, 3 as 011, and the calculation result is #100=7;

G65 H12 P#100 Q#101 Q#102;

In the binary system, 5 is represented as 101, 3 as 011, and the calculation result is #100=1;

5. Macro variable alarm

Example:

G65 H99 P1; Macro Variable Alarm 3001

G65 H99 P124; Macro Variable Alarm 3124

User macro program examples

1. Bolt hole cycle

On the circumference where the circle center is the reference point (X0, Y0) and the radius is (R), the starting angle is (A), and N equant holes are machined.

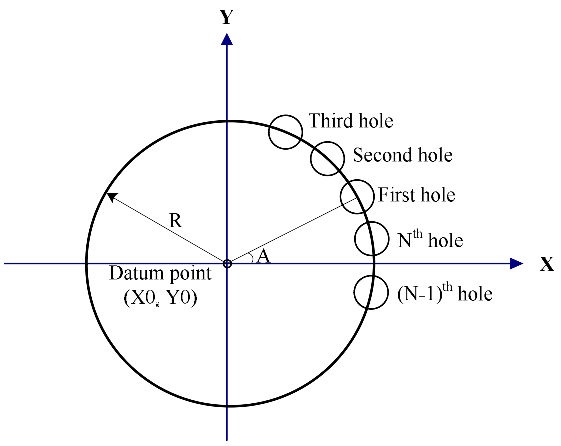


Fig. 4-9-4-2

X0, Y0 are the coordinate values of the bolt hole cycle reference point.

R: Radius, A: Starting angle, N: The number of holes. The following variables are used in the above parameters.
#500: Coordinate value of reference point in X axis (X0)
#501: Coordinate value of reference point in Y axis (Y0)
#502: Radius (R)
#503: Starting angle (A)
#504: N holes
When N>0, rotate counterclockwise, and the number of holes is N.
When N<0, rotate clockwise, and the number of holes is N.
The following variables are used for operations in macro programs.
#100: indicates the count of Hole I machining (I)
#101: final value of the count (=∣N∣)(IE)
#102: angle of Hole I (θI)
#103: coordinate value of Hole I in X axis (Xi)
#104: coordinate value of Hole I in Y axis (Yi)
The user macro program itself can be written in 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;
G65 H05 P#102 Q#102 R#504;          θI＝A＋360°×I/N
G65 H02 P#102 Q#503 R#102;
G65 H32 P#103 Q#502 R#102;          X I＝X I＋R·COS(θI)
G65 H02 P#103 Q#500 R#103;
G65 H31 P#104 Q#502 R#102;          Y I＝Y I＋R·SIN(θI)
G65 H02 P#104 Q#501 R#104;
G90 G00 X#103 Y#104; positioning of Hole I
G**;                                Specific G code for hole machining
G65 H02 P#100 Q#100 R1;             I＝I＋1
G65 H84 P200 Q#100 R#101;           When I < IE, go to N200 to machine IE holes.
M99;
```

The followings are examples of programs to call the above user macro program itself:

```
O0010;
G65 H01 P#500 Q100;   X0=100MM
G65 H01 P#501 Q-200;Y0=-200MM
G65 H01 P#502 Q100;   R=100MM
G65 H01 P#503 Q20;    A=20°
G65 H01 P#504 Q12;                  N=12, rotate counterclockwise
G92 X0 Y0 Z0;
M98 P9010;                          Call the user macro program
G80;
X0 Y0;
M30;
```

4.9.5 User Macro Program - Function B

1. Arithmetic and logical operations

The operations listed below can be performed in variables. The expression to the right of the operator can contain constants and/or variables consisting of functions or operators. The variables #j and #k in the expression can be replaced with constants. Variables on the left can also be assigned with expressions.

Table 4-9-5-1 Arithmetic and logical operations

Function	Format	Remark
Definitions	#i = #j	

Addition Subtraction Multiplication Division	#i = #j + #k; #i = #j - #k; #i = #j * #k; #i = #j / #k;	
Sine Arc sine Cosine Arc cosine Tangent Inverse tangent	#i = SIN[#j]; #i = ASIN[#j]; #i = COS[#j]; #i = ACOS[#j]; #i = TAN[#j]; #i = ATAN[#j] / [#k];	The angle is specified in degree, and thus 90°30' is expressed as 90.5°.
Square root Absolute value Rounding Rounding up Rounding down Natural logarithm Exponential function	#i = SQRT[#j]; #i = ABS[#j]; #i = ROUND[#j]; #i = FUP[#j]; #i = FIX [#j]; #i = LN[#j]; #i = EXP[#j];	
Or Exclusive OR AND	#i = #j OR #k; #i = #j XOR #k; #i = #j AND #k;	Logical operations are performed bit by bit in binary numbers.
Convert from BCD to BIN Convert from BIN to BCD	#i = BIN[#j]; #i = BCD[#j];	Used for signal exchange with PMC.

Description:

(1) **Angle unit**

The angle unit in SIN, COS, ASIN, ACOS, TAN and ATAN is degree (°). For example, 90°30′ is expressed as 90.5°.

(2) **ARCSIN #i = ASIN [#j]**

The value ranges from -90° to 90°.
When #j exceeds the range of -1 to 1, the system will give an alarm.
The variable #j can be substituted by a constant.

(3) **ARCCOS #i = ACOS [#j]**

The value ranges from 180° to 0°.
When #j exceeds the range of -1 to 1, the system will give an alarm.
The variable #j can be substituted by a constant.

(4) **ARCTAN #i = ATAN [#j] / [#k]**

Specify the length of the two sides and separate them with a slash (/).
The value ranges from 0° to 360°.
[Example] When #1 = ATAN [-1] / [-1]; is specified, #1=225°.
The variable #j can be substituted by a constant.

(5) **Natural logarithm #i = LN [#j]**

When the anti-logarithm (# j) is 0 or less than 0, the system will give an alarm.
The variable #j can be substituted by a constant.

(6) **Exponential function #i = EXP [#j]**

When the operation result exceeds 99997.453535 (#j is approximately 11.5129), an overflow occurs and the will give an alarm.
The variable #j can be substituted by a constant.

(7) **ROUND function**

The ROUND function is used for rounding off at the first decimal place.
Example:
When #1=ROUND[#2]; is executed, #2=1.2345 and the value of Variable 1 is 1.0.

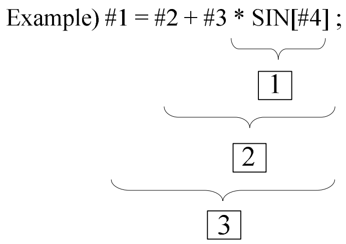
(8) **Rounding up and down**

When CNC is used to deal with numerical operation, if the absolute value of the integer generated after the operation is greater than the absolute value of the original number, it is rounded up; if it is smaller than the absolute value of the original one, it is rounded

down. Attention should be paid to handling of negative numbers.
For example:
Assume 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 for arithmetic and logical 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 Table 4-9-5-1 for details)
For example:
ROUND→RO
FIX→FI

(10) **Operation order**
①Function
②Multiplication and division (* / AND)
③Addition and subtraction (+ - OR XOR)

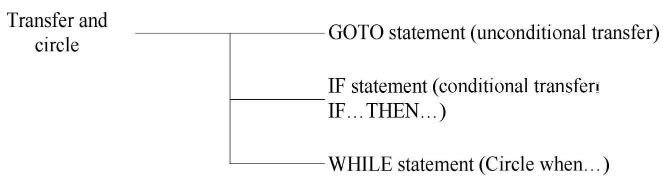


1, 2 and 3 mean order of operation.

(11) **Restrictions**
[,] is used for closed expressions.
The system will give an alarm when a divisor of 0 or TAN[90] is specified in division.

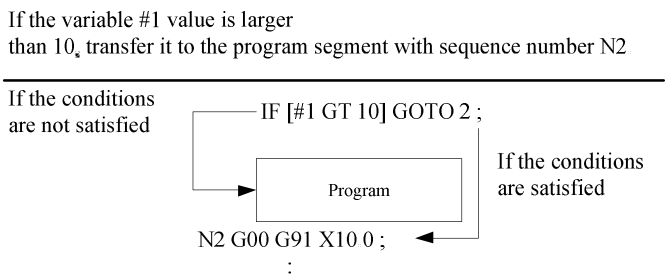
2. Jump and loop

1) Jump and loop
In a program, the GOTO statement and the IF statement can be used to change the flow direction of control. There are three jump and loop operations available:



2) Unconditional jump
- GOTO statement
Jump to the program segment with Sequence Number n The sequence number can be specified by an expression.
GOTOn; n: Sequence number (1 to 99999)
Example:
GOTO 1;
GOTO #10;

3) Conditional jump (IF statement) [<conditional expression>]
IF [<conditional expression>] GOTO n
If the specified conditional expression is satisfied, jump to the program segment with Sequence Number n; if not, the next program segment will be executed.



IF [<conditional expression>] THEN

If the conditional expression is satisfied, execute the preset macro program statement. Only one macro program statement will be executed.

If #1 and #2 share the same value, 0 is assigned to #3.
IF[#1 EQ #2] THEN #3=0;

Description:

- Conditional expression
  Conditional expressions must include operators. An operator is inserted between two variables or between a variable and a constant and enclosed in parentheses ([,]). Variables can be substituted by expressions.
- Operator
  An operator consists of 2 letters and is used to compare two values to determine if they are equal or if one value is less than or greater than the other.

Table 4-9-5-2 Operator

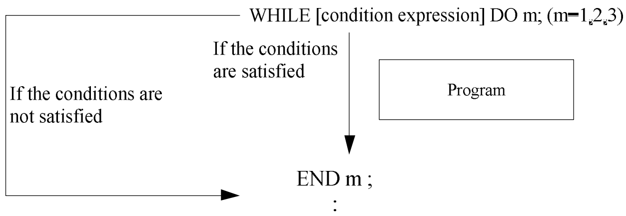
Operator	**Meaning**
EQ	Equal to (=)
NE	Not equal to (≠)
GT	Greater than (>)
GE	Greater than or equal to (≥)
LT	Less than (<)
LE	Less Than or Equal To (≤)

- Typical program
  The following program is used to calculate the sum of the values 1 - 10.

O9500；	
#1=0；	Initial value of storage sum variable
#2=1；	Initial value of augend variable
N1 IF[#2 GT 10]GOTO 2；	When the augend is > 10, transfer to N2
#1=#1+#2；	Calculate the sum
#2=#2+1；	The next augend
GOTO 1；	Transferred to N1
N2 M30；	Completion of program

4) Loop (WHILE statement)

Specify a conditional expression after WHILE. When the specified condition is satisfied, execute the program from DO to END, or otherwise jump to the program segment after END.

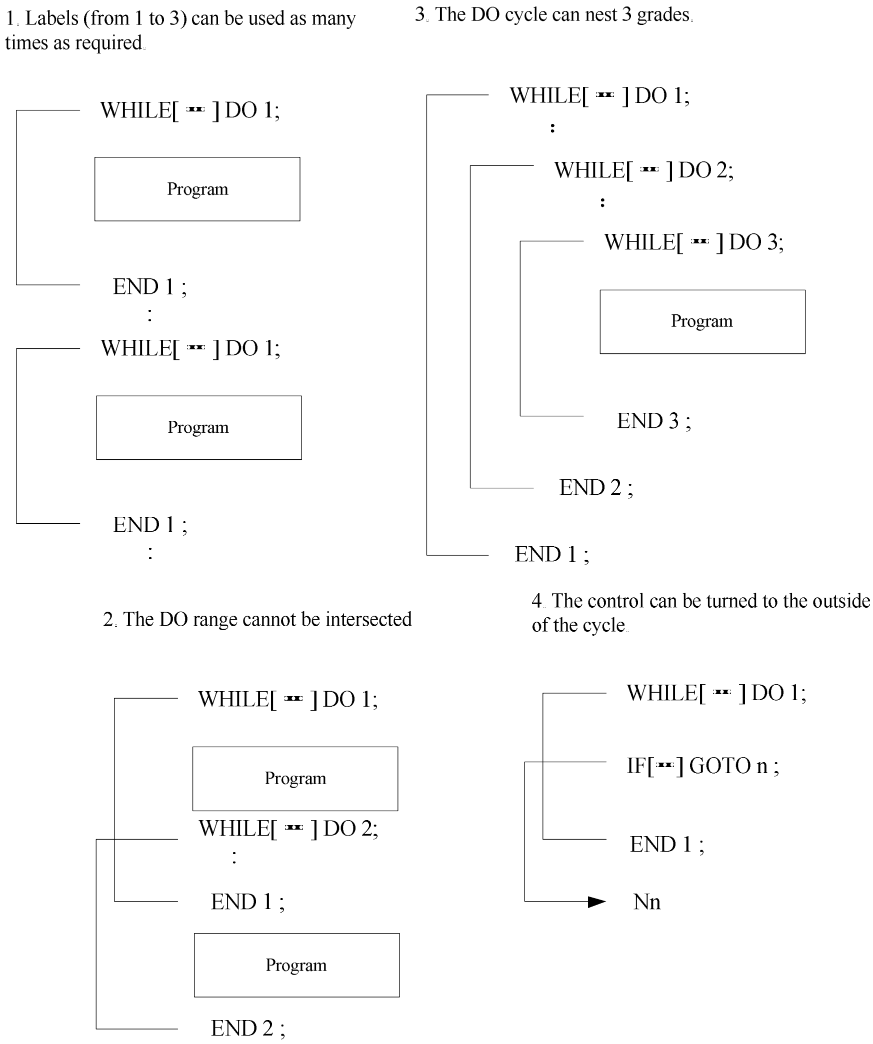


When the specified condition is satisfied, execute the program between WHILE and DO. Otherwise, execute the program segment after END. Such command format applies to IF statements. The mark numbers after DO and END are to specify the execution range of the program. Their values are 1, 2 and 3, and if other values are used, an alarm will be generated.

Description:

- Nesting

  The mark numbers (1 to 3) in the DO-END loop can be used as many times as needed. However, an alarm will be generated when the program is found overlap with the crossover loop DO.



- Infinite loop

  When DO is specified without specifying a WHILE statement, an infinite loop from DO to END is generated.

- Treatment time

  When a GOTO statement with jump to other mark numbers is treated, perform the sequence number retrieval. Reverse retrieval takes longer than forward retrieval. Implementing loops with WHILE statements can reduce treatment time.

- Undefined variables

  In conditional expressions using EQ or NE, <empty> and zero have different effects. In conditional expressions in other forms, <empty> is treated as zero.

- Typical program

  The following program is used to calculate the sum of the values from 1 to 10.

```
O0001 ;
#1=0;
#2=1;
WHILE [#2 LE 10] DO 1;
#1=#1+#2;
#2=#2+1;
END 1;
M30;
```


Notice:

- When G65 is used to call a macro program and F is used for reference to variables, the system will execute according to the variable value.
- Loop and jump commands cannot be used in DNC.
- When a GOTO statement is used, the system will search down from the current program segment, and if no corresponding sequence number is searched, it will return to the program header for re-search. Try to avoid using the same N code in a program.
- When the variable number is represented by a decimal, the system will directly round off the fractional part without considering carry.
- Local variables will be maintained until the main program ends and can be shared in subprograms.

Chapter V Auxiliary Function M Code

The M codes available to user of this machine tool are listed as follows:

Table 5-1

	M code	Function
M code for program control	M30	The program ends and returns to the program header, with the number of machined workpieces increased by 1.
	M02	The program ends and returns to the program header, with the number of machined workpieces increased by 1.
	M98	Call subprogram
	M99	End and return of subprogram / repeat execution
	M00	Pause of program
	M01	Selective halt of program
Control the M code with PLC	M03	The spindle rotates CW
	M04	The spindle rotates CCW
	M05	The spindle stops
	M06	Replace the tool
	M07	Blowing on
	M08	Cooling on
	M09	Cooling off/blowing off
	M10	Unclamping of A axis
	M11	Clamping of A axis
	M16	Tool control - unclamping
	M17	Tool control - clamping
	M18	Cancel spindle orientation
	M19	Spindle orientation
	M20	Spindle neutral command
	M21	Tool search code when returning the tool
	M22	Tool search code when clamping the new tool
	M26	Start chip flush valve
	M27	Close chip flush valve
	M28	Cancel rigid tapping
	M29	Rigid tapping command
	M35	Start chip removal lifting conveyor
	M36	Close chip removal lifting conveyor
	M50	Start automatic tool change
	M51	End automatic tool change

When the movement code and the auxiliary function are specified in the same program segment, they will be executed simultaneously.

When a value is specified after the address M, the code signal and the gating signal are sent to the machine tool, which uses these signals to turn these functions on or off. Usually, only one M code can be specified in one program segment. Position Parameter **N0: 33#7** can be set to allow up to three M codes specified in one program segment. However, due to restrictions on mechanical operation, some M codes cannot be specified at the same time. For restrictions on specifying several M codes in the same program segment by mechanical operation, see the machine tool manufacturer's manual.

5.1 M Code Controlled By PLC

When the M code controlled by PLC shares the same segment with the movement code, the M code will be executed simultaneously with the movement code.

5.1.1 Spindle Rotation Cw and Ccw Commands (M03, M04)

Code: M03 (M04) Sx x x;
Description: According to applicable standards, the spindle rotation CCW is defined as forward rotation and the rotation CW as reverse rotation.
M03 means rotation CW while M04 means rotation CCW.
The Sx x x code refers to the speed of the spindle, or the gear position in case of gear control.
Unit: r/min
When controlled by frequency converter, Sx x x refers to the actual speed, for example: S1000 means that the spindle rotates at a speed of 1000 r/min.

5.1.2 Spindle Stop Code Command (M05)

Code: M05. When M05 is executed automatically, the spindle will stop rotating. However, the speed specified by the S code will be reserved. The deceleration mode in which the spindle stops rotating depends on the machine tool manufacturer's settings. Usually, it is dynamic braking.

5.1.3 Cooling On and Off (M08, M09)

Code: M08, make the cooling water pump on. M09, make the cooling water pump off. In the automatic mode, if the auxiliary function lock is used, the pump control code is not executed.

5.1.4 A-Axis Unclamping and Clamping (M10, M11)

Code: M10, unclamping of A axis M11, clamping of A axis

5.1.5 Tool Control - Unclamping and Clamping (M16, M17)

Code: M16, tool control - unclamping M17, tool control - clamping

5.1.6 Spindle Orientation and Cancellation (M18, M19)

Code: M18, cancel the spindle orientation M19, spindle orientation for tool replacement positioning

5.1.7 Tool Search Code Command (M21, M22)

Code: M21, tool search code when returning the tool; M22, tool search code when clamping the new tool.

5.1.8 Tool Magazine Return Code Command (M23, M24)

Code: M23, which makes the tool magazine placed in the spindle position; M24, which make the tool magazine return to the original position.

5.1.9 Rigid Tapping (M28, M29)

Code: M28, cancel rigid tapping M29, rigid tapping

5.1.10 Spiral Chip Conveyor On and Off (M35, M36)

Code: M35, make the spiral chip conveyor on. M36, make the spiral chip conveyor off.

5.1.11 Chip Flush Valve On and Off (M26, M27)

Code: M26 to start the chip flush valve; M27 to close the chip flush valve.

5.1.12 Spindle Blowing On and Off (M07, M09)

Code: M07, make the spindle blowing on. M09, make the spindle blowing off.

5.1.13 Start and End of Automatic Tool Replacement(M50, M51)

Code: M50, control the start of automatic tool replacement. M51, control the end of automatic tool replacement.

5.2 M Code For Program Control

The M code for program control is classified into main program control and macro program control. When the M code used for program control shares the same segment with the movement code, first executed is the movement code and then the M code.

Note: 1. M00, M01, M02, M06, M30, M98, M99 codes cannot be specified together with other M codes, or otherwise the system will give an alarm. When these M codes share the same segment with other non-M codes, first executed are the non-M codes and then the M codes.

2. Such M codes include those which cause CNC to send the M codes themselves to the machine tool and to perform internal operations, such as M codes that invalidate the read-ahead function of a program segment. In addition, the M codes, which only allow CNC to send the M codes themselves to the machine tool without performing internal operations, can be specified in the same program segment.

5.2.1 Program End and Return (M30, M02)

In the automatic operation mode, when the program runs to M30 (M02), the automatic operation will stop. After that, if there is any program that will not be executed and the spindle and cooling operation will stop, the number of machined workpieces will be increased by 1. M30 can use Position Parameter **N0: 33#4** to control whether to return to the program header, and M02 can use Position Parameter **N0: 33#2** to control whether to return to the program header. If M02 and M30 are at the end of a subprogram, it will return the program calling the subprogram and continue to execute the following program segment.

5.2.2 Program Halt (M00)

In the automatic operation mode, this mode will be temporarily halted when the program runs to M00, and at the moment, the previous modal information will be saved. When the loop start button is pressed, it will continue running. Its function is equivalent to pressing down the feed hold button.

5.2.3 Selective Halt of Program (M01)

In the automatic operation mode, this mode can be selectively halted when the program runs to M01. When the “Selection-stop” switch is placed on, M01 and M00 have the same effect; if the “Selection-stop” switch is off, M01 has no effect. Please refer to Part II Operation Instructions for details.

5.2.4 Code Command For Program Calling Subprogram (M98)

In a main program, M98 can be programmed to call a subprogram. Specific format:

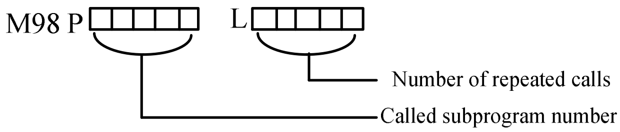


Fig. 5-2-4-1

5.2.5 Program End and Return (M99)

1. In the automatic operation mode, if M99 is used at the end of a main program segment, it will return to the beginning of the program for automatic execution when the program runs to M99. After that, if there is any program that will not be executed, the number of machined workpieces will not increase accumulatively.
2. When M99 is used at the end of a subprogram, it will return to the main program when running to this segment, and the program following the subprogram segment is called to continue execution.
3. In the DNC mode, M99 is used as M30, and the cursor stays at the end of the program.

Chapter VI Spindle Function S Code

Through the S codes and the values behind, the code signal is converted into an analog signal and sent to the machine tool for spindle control of the machine tool. S is a modal value.

6.1 Spindle Analog Control

When **SPT=0** in Position Parameter **N0: 1#2**, the address S and the values behind are used to control the spindle speed with analog voltage. Please see Part II Operation Instructions for details.

Code format: S_

Description:

1. One S code can be specified in one program segment.
2. The address S and the values behind can directly specify the spindle speed (unit: r/min). For example: M3 S300 indicates that the spindle runs at 300 r/min.
3. When the movement code and S code share the same program segment, the movement code will be executed simultaneously with the S functional code.
4. The spindle speed is controlled via the S code and the values behind it.

6.2 Spindle Switching Value Control

When **SPT=1** in Position Parameter **N0: 1#2**, the address S and the two-digit switching value behind are used to control the spindle speed.

When the switching value is selected to control the spindle speed, the system can provide 3-lelve mechanical gear shift for the spindle. Please see the machine tool manufacturer's manual for the correspondence between the S code and the spindle speed and how many levels of gear shift are provided for the machine tool.

Code format:

S01(S1);
S02 (S2);
S03 (S3);

Description:

1. At present, the software provides 8 gear-shift positions, and the gear shift of only 3 levels is shown in the ladder diagram When an S code other than the above ones is specified in the program, the system will display "**Auxiliary function under execution**".
2. If the S command has four digits, the last two digits are valid.

6.3 Constant Surface Cutting Speed Control G96/G97

Code format:

Constant surface cutting speed control code G96 S_ Surface speed (mm/min or inch/min)
Constant surface cutting speed control cancellation code G97 S_ Spindle speed (r/min)
Controlled shaft code of constant surface cutting speed control G96 P_ P1 X-axis; P2 Y-axis; P3 Z-axis; P4 4th axis
Maximum spindle speed clamp G92 S_ S specified maximum spindle speed (r/min)

Function: Specify the surface speed (relative speed between the tool and the workpiece) after S to rotate the spindle and keep the surface cutting speed constant, regardless of the tool position.

Description:

1. G96 is the modal code. After the instruction G96, the program enters the constant speed control mode, and the value of S is the surface speed.

2. G96 code must specify the axis around which the constant speed control is adopted. . G97 code cancels G96 means.
3. In order to implement the constant surface cutting speed control, the workpiece coordinate system should be set so that the center coordinate of the rotation axis becomes zero.

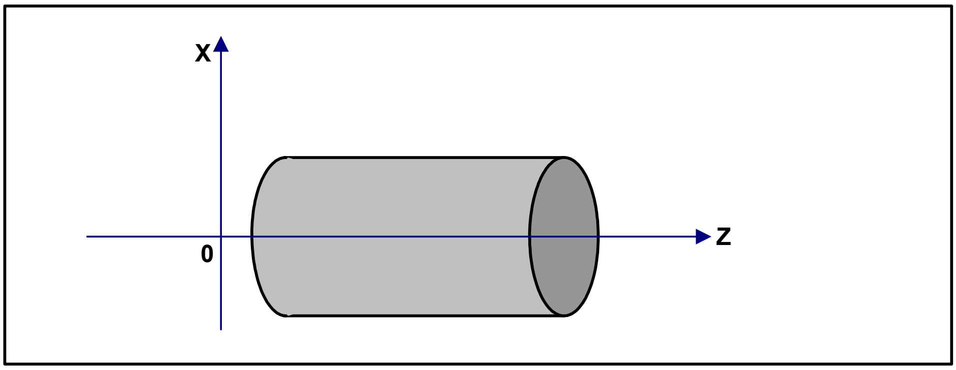


Fig. 6-3-1 Workpiece coordinate system of constant surface cutting speed control

4. When using constant surface cutting speed control, higher than G92 S_ setting, clamp on the highest spindle speed. When the power is switched on and the maximum spindle speed is not set, S in G96 code is treated as S=0 until M3 or M4 appears in the program.

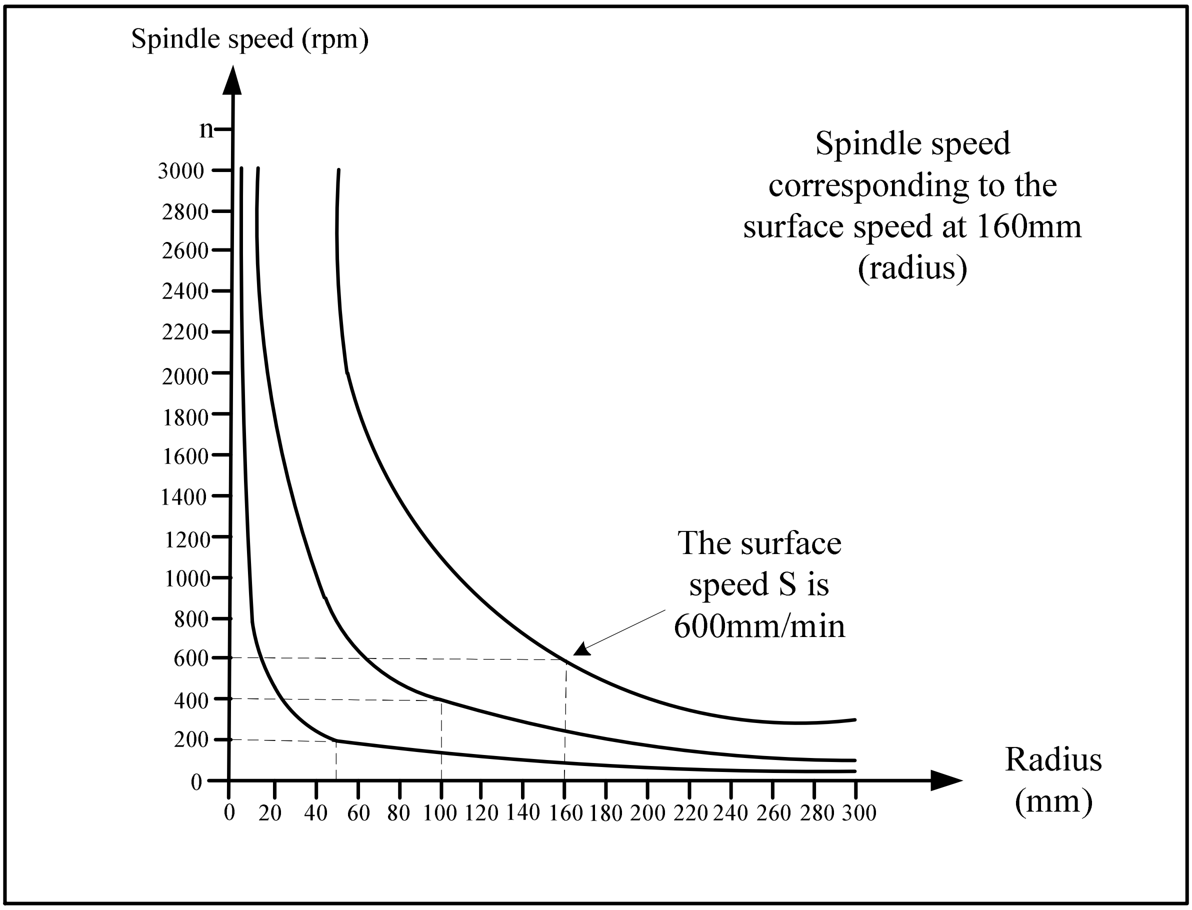


Fig. 6-3-2 Relationship between workpiece radius spindle speed and surface speed

5. Specify the surface cutting speed in G96 way:

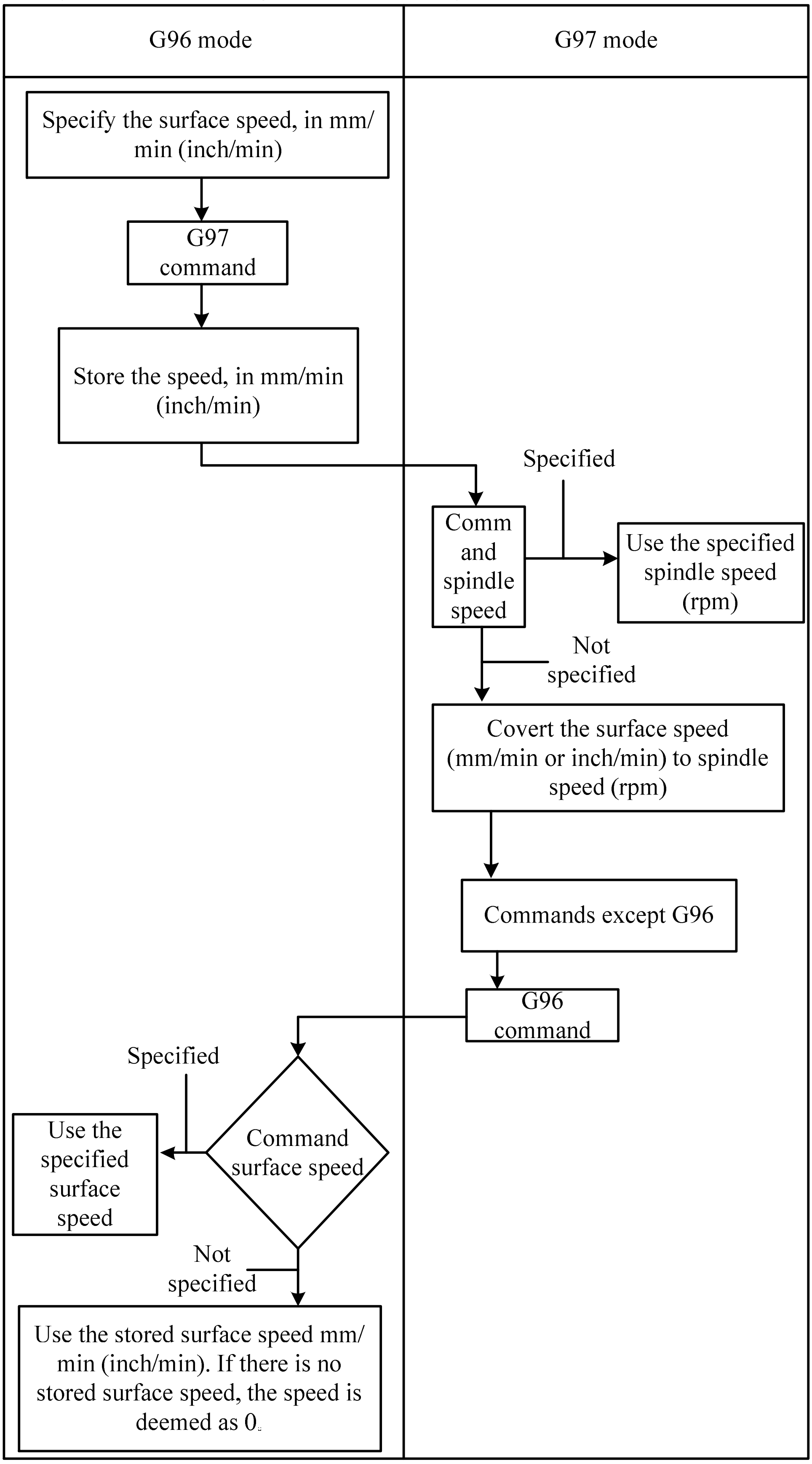


Fig. 6-3-3

6. G96 related parameter setting: Bit parameter No.37#2 is set to calculate the reference coordinate of G96 spindle rotating speed when G0 is positioned fast (0: end point, 1: current point); bit parameter No.37#3 set G96 spindle speed clamping (0: before spindle override, 1: after spindle override), and bit parameter No.61#0 set to determine whether constant cycle speed control is used.

Restrictions:

1. Because the response problem in the servo system is not considered when the spindle speed changes, and constant surface cutting speed control is also effective during thread cutting, G97 is used to cancel the constant surface cutting speed control before thread processing.
2. In the fast moving program segment specified by G00, the constant surface speed control is not calculated based on the instantaneous change in the tool position, but based on the end point of the segment; because the fast movement will not lead to cutting, no constant surface cutting speed is needed.
3. In the process of soft tapping, rigid tapping or deep-hole rigid tapping, it is necessary to cancel the constant surface cutting speed with G97 first, or otherwise there will be incorrect thread or broken tap.

Chapter VII Feed Function F code

The feed function controls the feeding speed of the tool. The feed function and control mode are as follows:

7.1 Fast Movement

Use code (G00) for fast positioning. Fast feed speed is set by the data parameters **P88-P92**. The override adjustment key on the operation panel can be used for the following override adjustment:

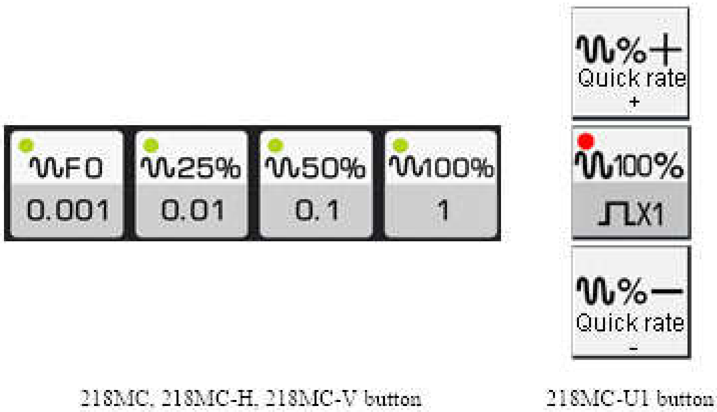


Fig. 7-1-1 Fast feed override key

Where, F0 is set by the data parameter **P93.**

The acceleration of fast positioning (G0) can be set by the data parameters **P110-P114**, and the acceleration / deceleration time constant can be set by the data parameters **P115-P124**. According to the response characteristics of the machine tool and the motor, it can be set reasonably.

Note: In the G00 program section, even if the feed speed F code is specified, it is invalid. The system is positioned at G0 speed.

7.2 Cutting Speed

In linear interpolation (G01) and circular interpolation (G02, G03), the number following F code is used to command the feed speed of the tool. The unit is mm/min. The tool moves at the cutting feed rate programmed. Use the feed override key on the operation panel of the machine tool to implement the feed override of cutting (the range of the feed override adjustment is: 0％-200%).

In order to prevent mechanical vibration, acceleration can be set by data parameters **P125-P128** when the tool movement starts and ends with automatic acceleration / deceleration.

The maximum cutting speed is set by the data parameter **P96**, and the minimum cutting speed is set by the data parameter **P97**. If the cutting speed is higher than the maximum limit, it is limited to the upper limit; if it is lower than the minimum limit, it is limited to the lower limit.

The cutting feed speed in the automatic mode when the power is switched on is set by the data parameter **P87**.

The cutting speed can be specified in two ways;

A) Feed per minute (G94): specify the tool feed per minute After F.

B) Feed per revolution (G95): Specify the spindle feed per revolution after F.

7.2.1 Feed Per Minute (G94)

Code format: G94 F_
Function: Tool feed per minute. Unit: mm/min or inch/min.
Description:

1. After G94 (feed per minute) is specified, the tool feed per minute is directly specified by the value following F.
2. G94 is modal code and, once specified, is valid until G95 is specified. When starting up, the feed mode per minute is in default, and the default cutting feed speed is set by the data parameter P87.
3. The **feed per minute** can be adjusted by the override adjustment key or the band switch on the panel, with the override from 0% to 200%.

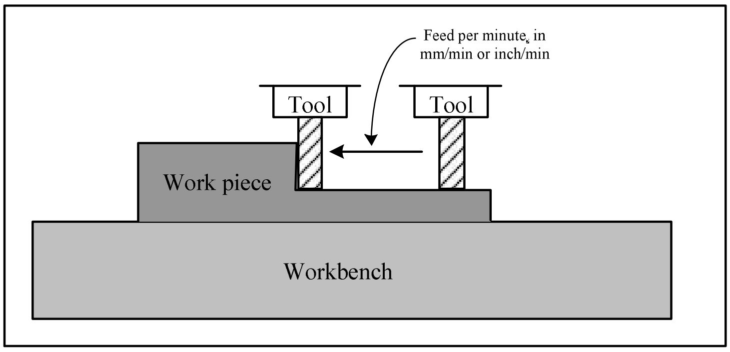


Fig. 7-2-1-1 Feed per minute

7.2.2 Feed Per Revolution (G95)

Code format: G95 F_
Function: Tool feed per revolution. Unit: mm/r or inch/r.
Description:

1. Machine tools must be installed with spindle encoder to use this function.
2. After G95 (feed mode per revolution) is specified, the feed amount per revolution is directly specified by the value following F.
3. G95 is modal code and, once specified, is valid until G94 is specified. The feed speed per revolution in initialization is zero by default.
4. The feed per revolution can be adjusted by the override adjustment key or the band switch on the panel, with the override from 0% to 240%.

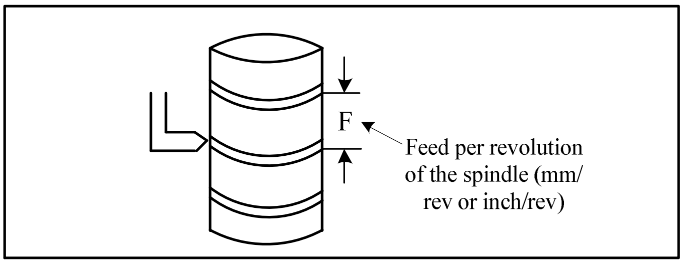


Fig. 7-2-2-1 Feed per revolution

Note: When the spindle speed is low, the feed speed may fluctuate. The lower the spindle speed is, the more frequently the feed amount fluctuates.
Note: For the feed mode per revolution of G95, the maximum speed of each feed handled by the system is F500. If it exceeds F500, an alarm will appear.

7.3 Tangential Speed Control

Generally, the cutting feed is to control the speed of the tangent direction of the contour track so as to reach the instructed speed value.

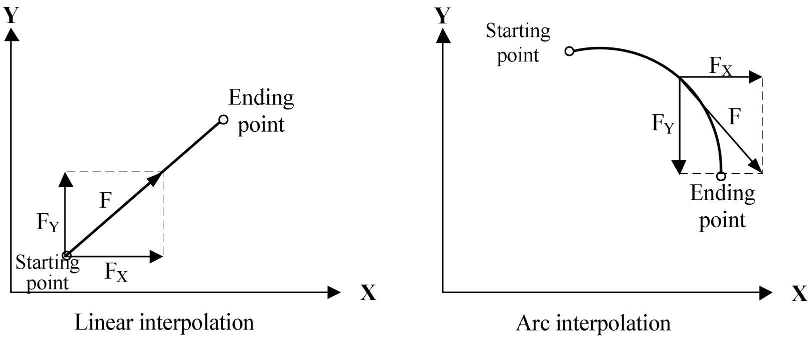


Fig. 7-3-1

$$F=\sqrt{F_X^2+F_Y^2+F_Z^2}$$

F: Speed in tangent direction
Fx: Speed in X axis direction
Fy: Speed in Y axis direction
Fz: Speed in Z axis direction

7.4 Feed Speed Override Key

The feed override under manual mode and automatic mode can be adjusted through the override adjustment key on the operation panel, with the range of 0-200% (10% for each gear, 21 gears in total). Under automatic mode, when the override adjustment key is set to zero, the system will stop feeding, showing the cutting override is zero; then the override adjustment key should be adjusted to keep the program running.

7.5 Automatic Acceleration and Deceleration

The system drives the motor to automatically accelerate and decelerate at the beginning and end of movement; so it can start and stop smoothly. It also automatically accelerates and decelerates when the movement speed changes; so the speed change can proceed smoothly. Therefore, there is no need to consider acceleration and deceleration when programming.

Fast feed: Forward acceleration and deceleration (0: linear; 1: S-type) Post acceleration and deceleration (0: linear; 1: exponential type).

Cutting feed: Forward acceleration and deceleration (0: linear; 1: S-type) Post acceleration and deceleration (0: linear; 1: exponential type).

Manual feed: Post acceleration and deceleration (0: linear; 1: exponential type).

(Use parameters to set the universal time constant of each axis)

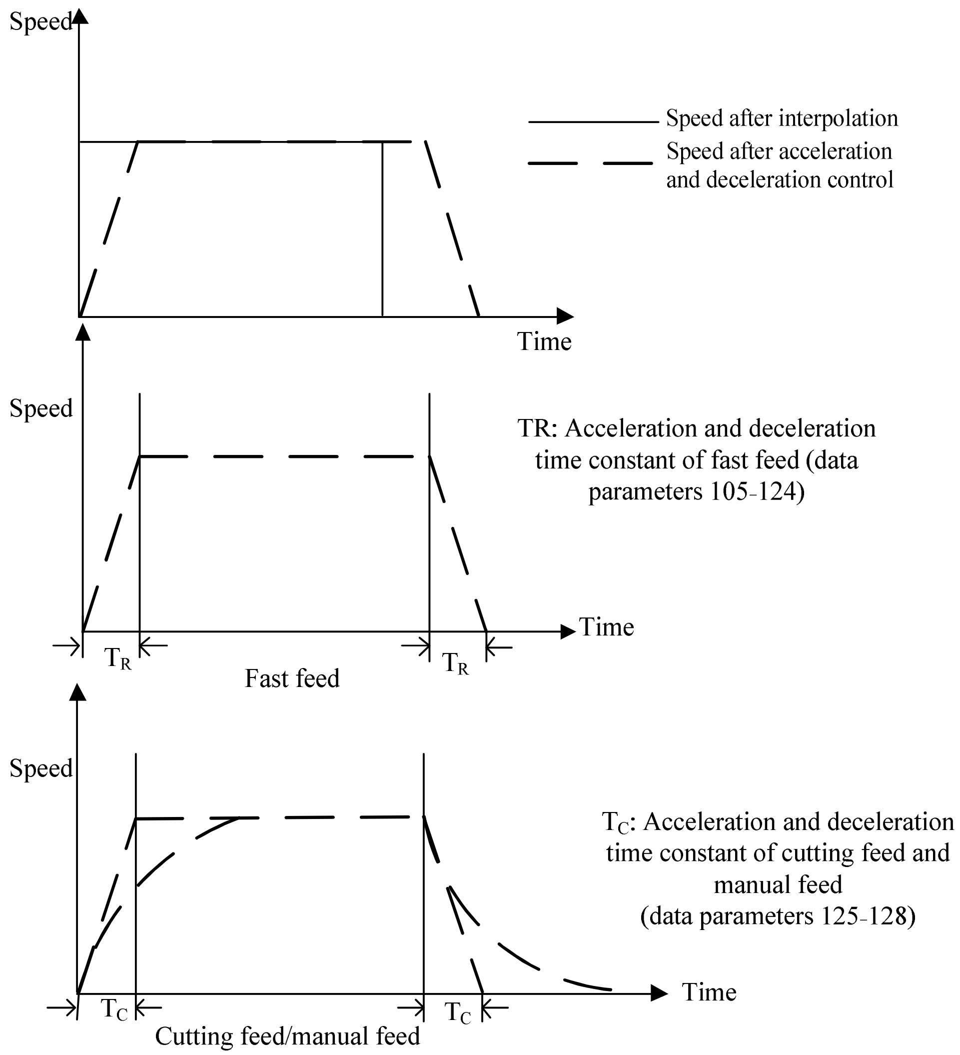


Fig. 7-5-1

7.6 Acceleration and Deceleration Processing At Program Segment Corner

For example: In the previous program segment, only Y moves, while in the next program segment, only X moves. When Y decelerates, X accelerates, and the path of the tool is as follows:

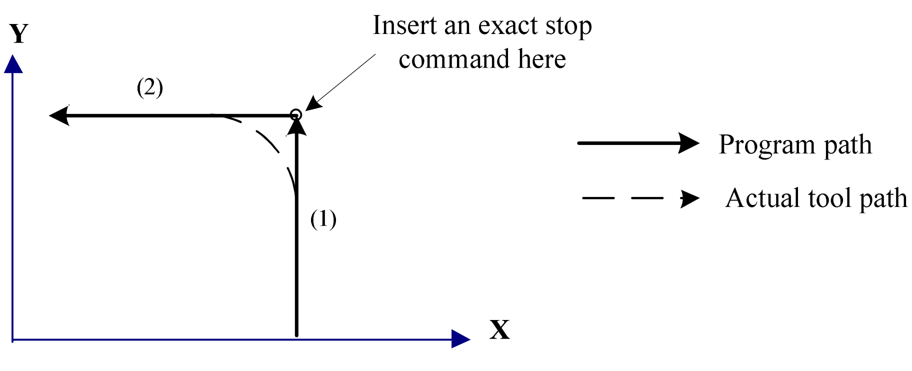


Fig. 7-6-1

If the exact-stop code is added, the tool will move according to the program instructions as shown in the solid line above. Otherwise, the greater the cutting feed speed, or the longer the acceleration and deceleration time constant, the greater the radiant of the corner is. In arc command, the arc radius of the actual tool path is smaller than that given by the program. To reduce the corner error, the acceleration and deceleration time constant should be reduced as far as possible if the mechanical system allows.

Chapter VIII Tool Functions

8.1 Tool Functions

Specify a value (up to 8 digits) after address T to select the tool on the machine.

In principle, two or more T codes cannot be instructed in the same program segment. If the same group of codes are set in the same segment, no alarm will appear. Please refer to the T code that appears later. For the number of digits that can be specified by address T and the machine tool action corresponding to T code, please refer to the operation manual of the machine tool plant.

When the movement code and T code are specified in the same program segment, the movement code and T code are implemented simultaneously.

When T code and tool change code M06 are in the same segment, T code will be implemented first, followed by the tool change code. When T code and tool change code M06 are in different segments, it is necessary to check whether the spindle tool number is consistent with code T tool during tool code change. If it is consistent, tool change will not be implemented.

As shown in the following procedure example:

```
O00010;
N10 T2M6;        The tool on the spindle is T2
N20 M6T3;        The tool on the spindle is T3
N30 T4;          The tool on the spindle is T3
N40 M6;          The tool on the spindle is T4
N50 T5;          The tool on the spindle is T4
N60 M30
%
```

After the tool change, the tool on the spindle is T4.

VOLUME II OPERATING DESCRIPTION

Chapter I Operating Panel

1.1 Panel Division

GSK218MC series includes GSK218MC, GSK218MC-H and GSK218MC-V, among which **GSK218MC** numerical control system adopts the combination structure, **GSK218MC-H and GSK218MC-V** adopt the horizontal and vertical structure respectively, and the panel is divided into LCD (liquid crystal display) area, editing keyboard area, soft key function area and machine control area, as shown below:

Fig. 1-1-1 GSK218MC panel

Fig. 1-1-2 GSK218MC-H panel

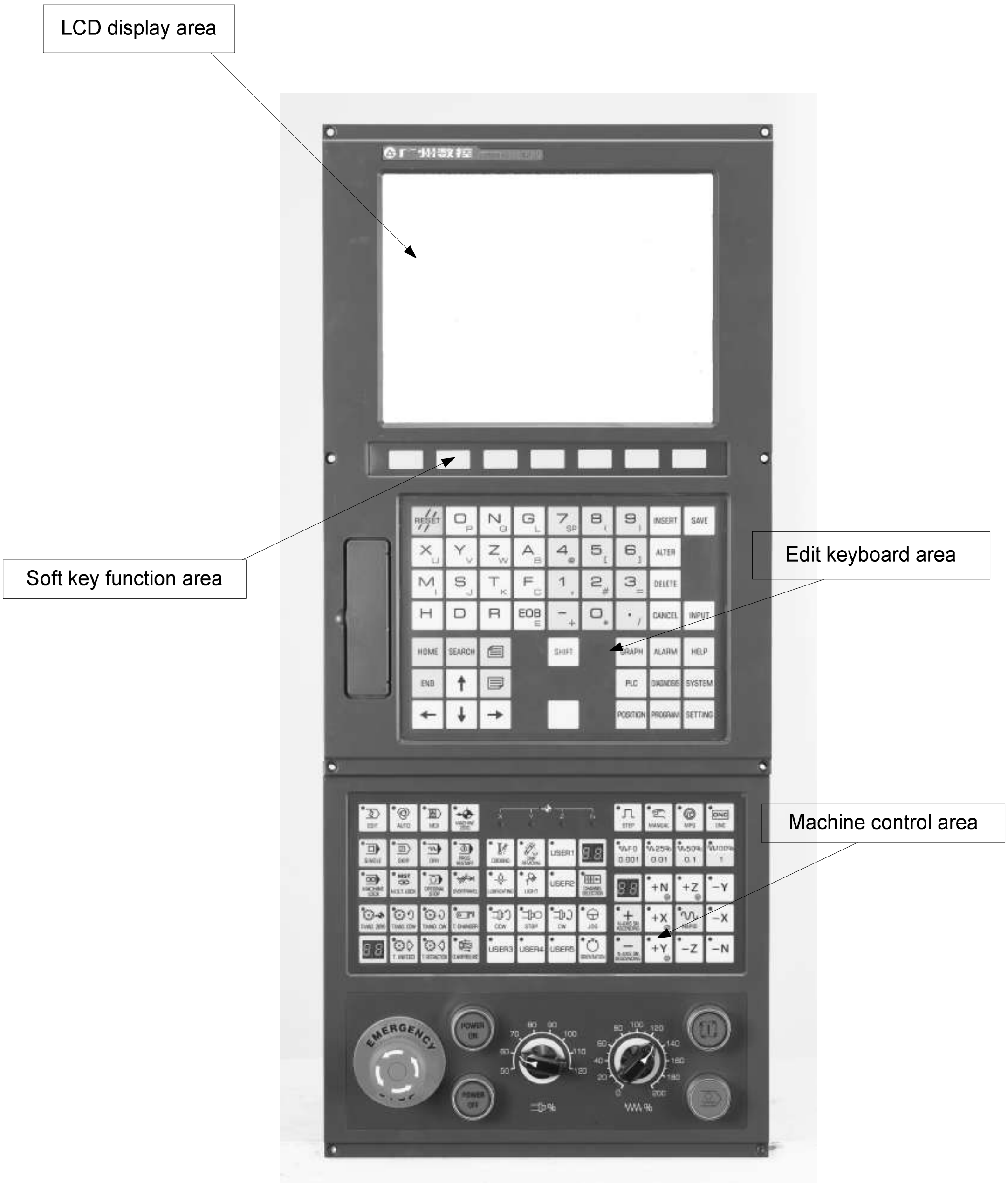


Fig. 1-1-3 GSK218MC-V

1.2 Panel Function Description

1.2.1 Lcd (Liquid Crystal Display) Area

GSK 218MC and GSK 218MC-V systems adopt 10.4-inch color LCD with resolution of 800×600.
GSK 218MC-H system adopts 8.4-inch color LCD with resolution of 800×600.

1.2.2 Editing Keyboard Area

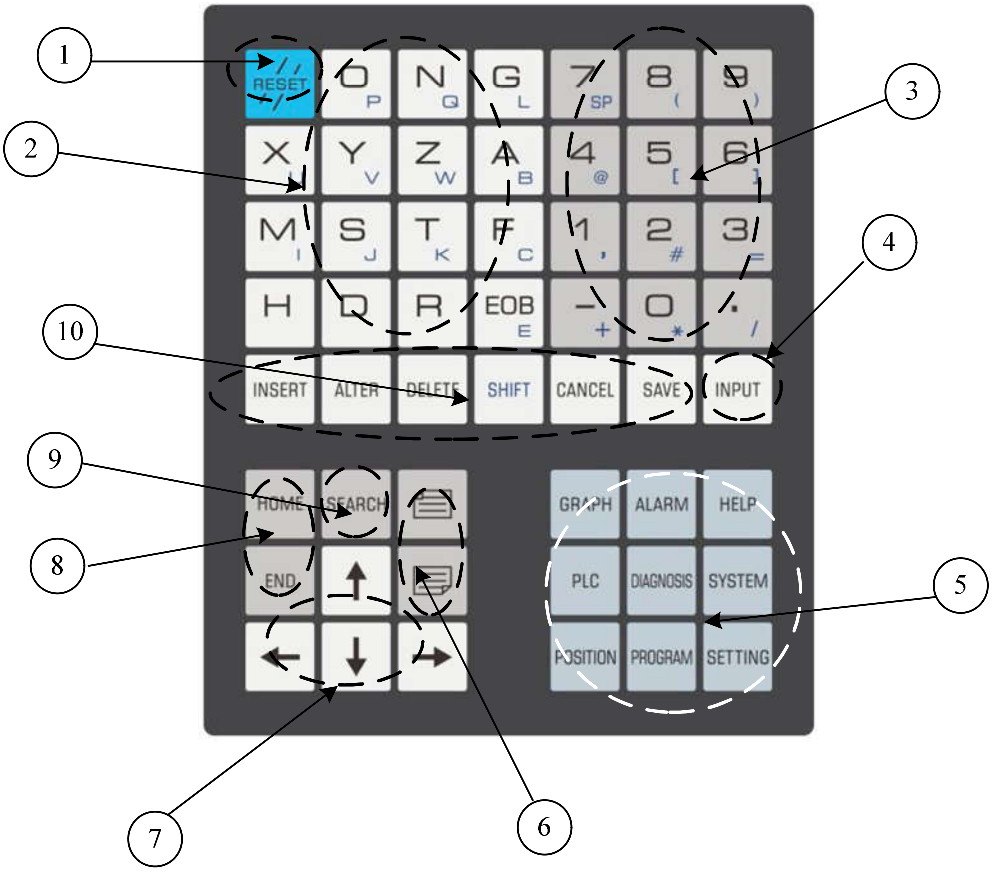


Fig. 1-2-2-1 Editing keyboard area of GSK218MC and GSK218MC-H

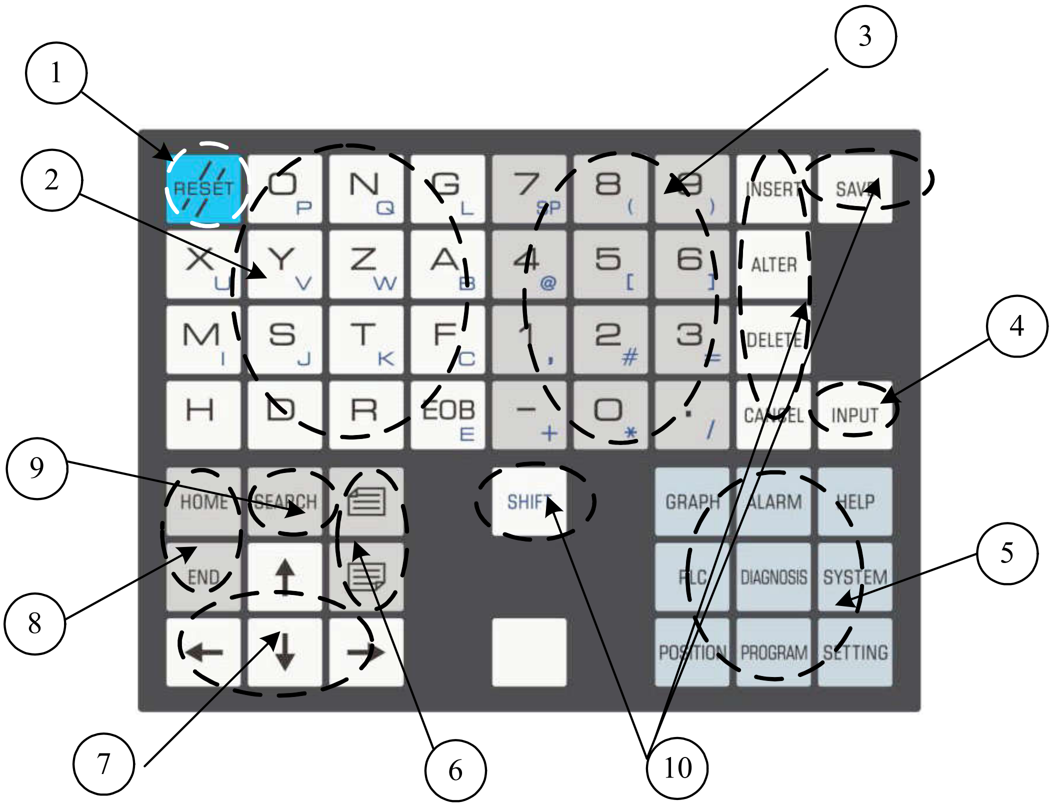


Fig. 1-2-2-2 Editing keyboard area of GSK218MC-V

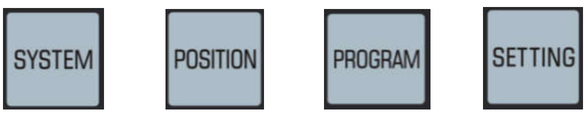
In the editing keyboard area, the functions of keys are further subdivided into 10 subareas. The specific instructions for each subarea are as follows:

S/N	Name	Function description
1	Reset key	System reset, feed and output stop
2	Address key	Address MDI entry
3	Number key	Digital MDI entry
4	Input key	Enter numbers, addresses, or data into the buffer area; confirm the operation result
5	Screen operation key	Press any key to enter the corresponding interface display. (Detailed introduction in Chapter III)
6	Page-Up and Page-Down keys	For page conversion, program page-up and page-down on the same display
7	Cursor moving key	Up/down and left/right movement of cursor
8	Editing key	Enable the cursor to move to the beginning or end of a program line or a program
9	Search key	Used to search for data, address for viewing or modification
10	Editing key	Used for program and field insertion, modification deletion operations during program editing, and as compound key.

1.2.3 Introduction To Screen Operation Key

This system has 8 operation page display keys and 1 help page display key on the operation panel, as shown below:

Graphic page Alarm page Help page PLC page Diagnosis page



System page Position page Program page Setting page

Name	Function description	Remark
Graphics page	Enter the graphics page (GRA)	Through the corresponding soft key conversion, display the page of graph parameters and graph display, Graph parameters display the graphics center, size and scale settings
Alarm page	Enter the alarm page (ALM)	Through the corresponding soft key conversion, check various alarm information pages
Help page	Enter the help page (HELP)	Through the corresponding soft key conversion, check various help information related to the system
Program control page	Enter the program control page (PLC)	Through the corresponding soft key conversion, check the version information of PLC ladder diagram and the configuration of system I/O port, and alter the PLC ladder diagram in the input mode
Diagnosis page	Enter the diagnosis page (DGN)	Through the corresponding soft key conversion, check the signal status at I/O ports at different sides of the system
System interface	Enter the system page	Through the corresponding soft key conversion, display the tool offset, parameter, macro-variable and pitch error compensation

Position page	Enter the position page (POS)	Through the corresponding soft key conversion, display relative coordinates absolute coordinate and comprehensive coordinate of the current point and program monitoring
Program page	Enter the program page (PRG)	Through the corresponding soft key conversion, display the program, **MDI**, present/module, present/times, and directory; program names on multiple pages are available through the Page-Up and Page-Down keys on the content interface
Settings page	Enter the setting page (SET)	Including four interfaces; through the corresponding soft key conversion, display setting, workpiece coordinate, data and password setting interfaces

Note: By setting the bit parameters NO:25#0 - 25#7, NO:26#6 - 26#7, the above soft key conversion interfaces can also be realized by pressing the corresponding function keys continuously. For detailed instructions on each page, please refer to Chapter 3 of the Manual "Part II Operation instructions".

1.2.4 GSK218mc Machine Control Area

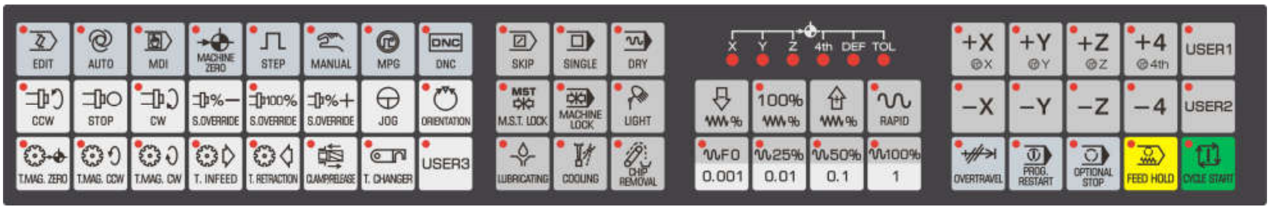
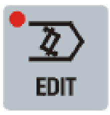
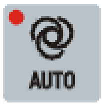
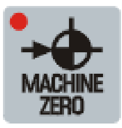
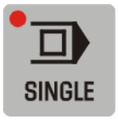
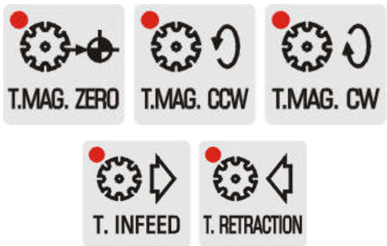
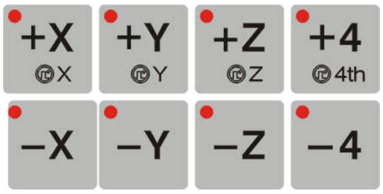
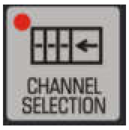


Fig. 1-2-4-1 GSK218MC machine control area

Keys	Name	Function description	Remarks and operation instruction
	Edit mode selection key	Enter the edit operation mode	Switch to edit mode when running in automatic mode, MDI and DNC mode, and the system will slow down and stop after running the current program segment
	Automatic mode selection key	Enter automatic operation mode	When automatic mode is selected, the system will select internal memory program
	MDI mode selection key	Enter entry (MDI) operation mode	Switch to MDI mode when running in automatic mode, and the system will slow down and stop after running the current program segment
	Mechanical zeroing mode selection key	Enter mechanical zeroing operation mode	Switch to zeroing mode when running in automatic mode, and the system will slow down and stop
	Manual single-step mode selection key	Enter manual single-step operation mode	Switch to single-step mode when running in automatic mode, and the system will slow down and stop
	Manual mode selection key	Enter manual operation mode	Switch to manual mode when running in automatic mode, and the system will slow down and stop

Keys	Name	Function description	Remarks and operation instruction
	Pulse mode selection key	Enter pulse operation mode	Switch to pulse mode when running in automatic mode, and the system will slow down and stop
	DNC mode selection key	Enter DNC operation mode	Switch to DNC mode when running in automatic mode, and the system will slow down and stop after running the current program segment
	Program segment selective-tripping switch	Whether the program segment marked with the header “/” symbol trips; when opened, the indicator light is on and the program is skipped	Automatic mode, entry mode and DNC
	Single-segment switch	Single program segment/continuous running state switching; when the indicator light is on, it runs in a single segment	Automatic mode, entry mode and DNC
	Dry running switch	When dry running is effective, the indicator light will light up	Automatic mode, entry mode and DNC
	Auxiliary function switch	The indicator light will light up when the auxiliary function is started; M, S, T and functional output become invalid	Automatic mode, entry mode and DNC
	Machine locking switch	The indicator light will light up when the machine locking is started; axis action output becomes invalid	Automatic mode, entry mode, mechanical zeroing, pulse mode, single-step mode, manual mode, DNC
	Machine working light switch	Machine working light on/off	Any mode
	lubrication on-off key	Machine lubrication on-off	Any mode
	Coolant on/off key	Coolant on/off	Any mode
	Chip removal on/off key	Chip removal on/off	Any mode

Keys	Name	Function description	Remarks and operation instruction
	Spindle control key	Positive rotation of spindle The spindle stops Negative rotation of spindle	Pulse mode, single-step mode and manual mode
	Spindle override key	Spindle speed adjustment (control mode of spindle rotation speed analog is effective)	Any mode
	Spindle inching switch	Spindle inching status on/off	Manual mode, single-step mode and pulse mode
	Spindle exact-stop key	Spindle exact-stop on/off	Manual mode, single-step mode and pulse mode
	Tool magazine action key	Tool magazine action on/off	Manual mode
	Manual tool releasing/tightening switch	Manual tool releasing/tightening switch	Manual mode
	Manual tool change	Complete manual tool change	Manual mode
	Overstroke release key	After the machine moves and presses the upper hard limit, the machine will alarm. Press the overstroke release key below, and the indicator light will be on. Move the machine in reverse until the indicator light is off.	Manual mode, pulse mode
	Program restart key	Exit the program being processed or restore to the processing state before power failure after the sudden power failure on site	Automatic mode (the residual movement here is the linear distance from the current point to the breakpoint)

Keys	Name	Function description	Remarks and operation instruction
	Selection on/off key	Stop or not when "M01" is in the program	Automatic mode, entry mode and DNC
	Movement override key	Rapid movement on/off	Any mode
	Rapid movement key	Rapid movement on/off	Manual mode
	Fast override, manual single-step and pulse override selection keys	Fast override, manual single-step and pulse override selection keys	Automatic mode, entry mode, mechanical zeroing, pulse mode, single-step mode, manual mode, DNC
	Manual feed key	Positive/negative movement in X-axis, Y-axis, Z-axis and N-axis under manual, single-step operation modes; positive axis is the selected axis of the pulse	Mechanical zeroing, single-step mode, manual mode and pulse mode
	Channel selection key	Processing channel switching (this function is unavailable temporarily)	Any mode
	Feed hold key	When pressing this key, the system will stop running automatically	Automatic mode, entry mode and DNC
	Cycle start button	When pressing this key, the program will run automatically	Automatic mode, entry mode and DNC

Note: When the number of the first symbol "/" at the program segment is greater than 1, the system will skip the program segment even if the segment skipping function is not enabled.

1.2.5 GSK218MC-H and GSK218MC-V Machine Control Areas

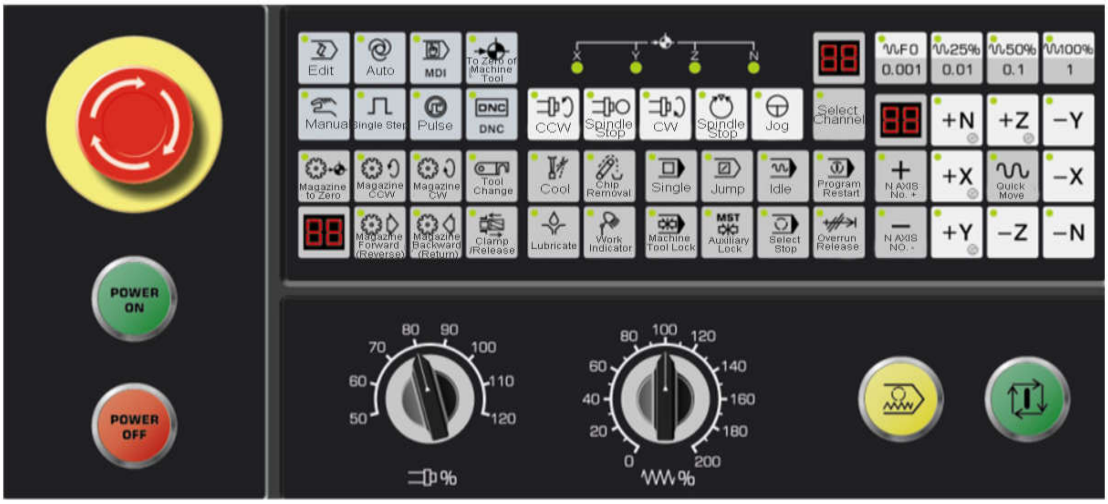
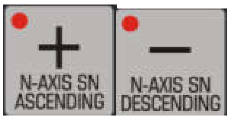
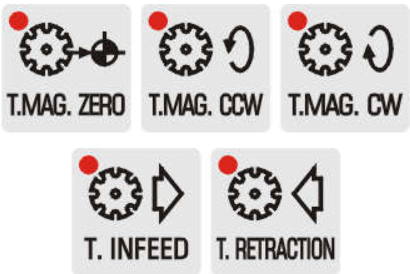
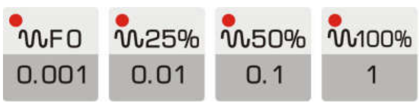
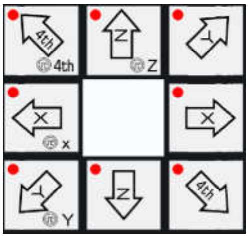


Fig. 1-2-5-1 GSK218MC-Hmachine control area

Fig. 1-2-5-2 GSK218MC-V machine control area

The use and function definition of the basic keys in GSK218MC-H and GSK218MC-V machine control areas are completely consistent with that of GSK218MC; now only the use of the increased key is described as follows:

Keys	Name	Function description	Remarks and operation instruction
	E-stop button	Make the system enter into the emergency stop state	Any mode
	N-axis selection key	Multi-axis switching	Manual mode, single-step mode and pulse mode
	Spindle override switch	Spindle speed adjustment (control mode of spindle rotation speed analog is effective)	Any mode
	Feed override switch	Feed speed adjustment	Automatic mode, entry mode, manual mode and DNC
	Tool magazine action key	Tool magazine action on/off	2018MC-U1 does not have any tool magazine-related keys
	Manual tool releasing/tightening switch	Manual tool releasing/tightening switch	2018MC-U1 does not have any tool releasing-related keys
	Manual tool change	Complete manual tool change	2018MC-U1 does not have any manual tool change-related keys
	Channel selection key	Processing channel switching (this function is unavailable temporarily)	2018MC-U1 does not have such function key
	Fast override, manual single-step and pulse override selection keys	Fast override, manual single-step and pulse override selection keys	Automatic mode, entry mode, mechanical zeroing, pulse mode, single-step mode, manual mode, DNC. The key function of 218MC-U1 is the same as that of 218MC, but the key icon is different.

	Movement override key	Rapid movement on/off	Any mode The key function of 218MC-U1 is the same as that of 218MC, but the key icon is different.
	Manual feed key	Positive/negative movement in X-axis, Y-axis, Z-axis and N-axis under manual, single-step operation modes; positive axis is the selected axis of the pulse	Mechanical zeroing, single-step mode, manual mode and pulse mode. The key function of 218MC-U1 is the same as that of 218MC, but the key icon is different.

Note 1: The feed hold key and  cycle start key  of GSK218MC are of the same effect as that of the  key and  key of GSK218MC-H and GSK218MC-V. The following is an example of the key of 218MC.

Note 2: in the manual mode, in the case that the rapid movement key is not pressed, the manual speed override is adjusted by the feed override switch.

Note 3: the key within < > in the following description is the panel key; the key in [ ] is the soft key under the screen; [ ] is the interface corresponding to the current soft key; ⊞indicates that the menu has sub-menus.

Chapter II System Power-on, Shutdown and Safe Operations

2.1 System Power-On

Before GSK218MC CNC system is powered on, the following should be confirmed:
1. The machine is in normal condition.
2. The supply voltage meets the requirements.
3. Wires are connected properly and bound securely.

After the system is normal through self-check and the initialization is completed, the current position (relative coordinates) page is displayed.

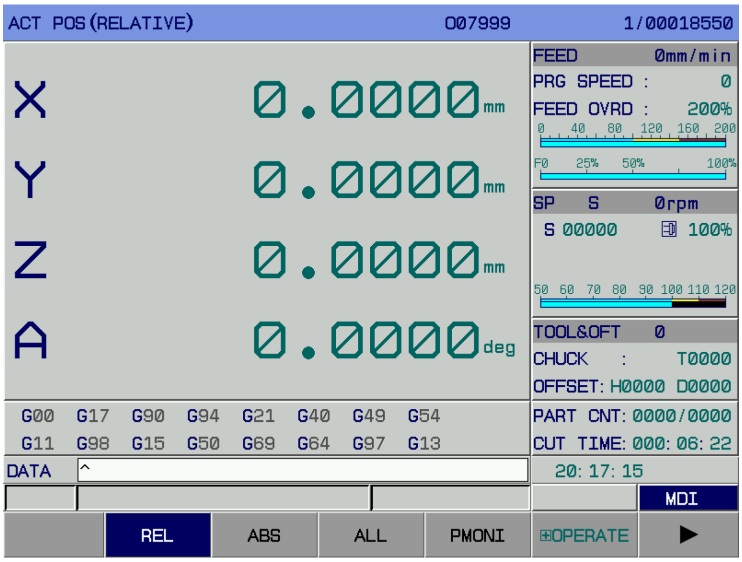


Fig. 2-1-1

2.2 Power-Off

Before power-off, the following should be confirmed:
1. X-axis, Y-axis and Z-axis of CNC are in a stop state;
2. Auxiliary functions (such as spindle and water pump) are closed;
3. First cut off the CNC power, then the machine power.

When powering off, check as follows:
1. Check the LED on the operation panel indicating that the cycle start is in a stop state;
2. Check that all movable parts of **CNC** machine are in a stop state;
3. Press POWER OFF (Power Off) button to shut down.

Power off in emergency

In emergency, the power supply of the machine can be cut off immediately in case of accident. However, it must be noted that there may be deviation between the system coordinates and the actual position after the power supply is cut off, and operations such as reset to zero and tool alignment must be carried out.

Note: Please refer to the machine manual of the machine manufacturer for cutting off the power supply of the machine.

2.3 Safe Operation

2.3.1 Reset Operation

Press 

key to reset the system:

1. All axial movements stop.
2. M function stops.
3. Modify the bit parameters NO. 35#1 - NO. 35#7 and NO. 36#0 - NO. 36#7. Decide whether to retain each group G code after resetting.
4. Modify the bit parameter NO:34#7. Decide whether to clear F, H and D codes after resetting.
5. Modify the bit parameter NO:28#7. Decide whether to delete the programming after resetting under MDI mode.
6. Modify the bit parameter NO:10#3. Decide whether to cancel the relative coordinate system after resetting.
7. Modify the bit parameter NO:10#7. Decide whether to return to the program header after resetting cursor in non-edit mode
8. Modify the bit parameter NO:52#7. Decide whether to empty macroprogram local variables #1 - #50 after resetting.
9. Modify the bit parameter NO:52#6. Decide whether to empty macroprogram public variables #100 - #199 after resetting.
10. Can be used for abnormal output of system or and abnormal action of coordinate axis.

2.3.2 Emergency Stop

Press the emergency stop button during the machine operation, and the system enters the emergency stop state. Then, the machine immediately stops movement. Release the emergency stop button (although it is different from machine manufacturers, but it usually automatically jumps by turning the button left) to release emergency stop.

Note 1: before releasing the emergency stop button, confirm whether the cause of fault has been eliminated.

Note 2: after the emergency stop button is released, the operation of returning to the reference point should be performed again to ensure the correctness of the coordinate position.

Generally, the emergency stop signal is the normally closed contact signal. When the contact disconnects, the system will enter the emergency stop state and make the machine stop urgently. The circuit connection of emergency stop signal is as follows:

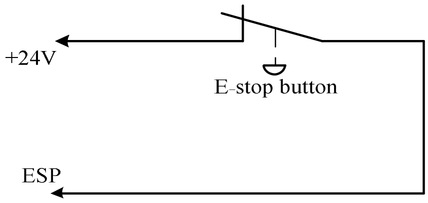


Fig. 2-3-2-1

2.3.3 Feed Hold

During machine operation process, press 

key to suspend the operation; what calls for special attention is that in the rigid tapping and loop code operation, it will suspend after the operation of the current code.

2.4 Cycle Start and Feed Hold

The  key and  key in the control panel are used to start and stop the operation of the program in automatic mode, entry mode and DNC mode. Change PLC address **K5.1** to set whether to use external start and stop.

Note 1: switch among automatic, MDI and DNC modes. Before executing the current program segment, the cycle start is valid; press <Feed hold> to make feed hold invalid.

Note 2: switch automatic, MDI, DNC modes to edit mode. Before executing the current program segment, the cycle start is invalid; press <Feed hold> to make feed hold invalid.

Note 3: switch from automatic, MDI and DNC modes back to machine zeroing, one-step, manual and pulse modes. Press the feed hold button to make the feed hold function invalid.

Note 4: when the cycle start becomes valid, and automatic, MDI and DNC modes are switched or switch to the edit mode, press the feed hold button before executing the current program segment, and then make the feed hold function invalid.

2.5 Overstroke Protection

In order to avoid the damage of the machine caused by the overstroke of X, Y and Z axes, the machine must take overstroke protection measures.

2.5.1 Software Overstroke Protection

Install stroke limit switches at the maximum stroke of X axis, Y axis and Z axis of the machine respectively. When overstroke occurs, the running axis will slow down and stop when it touches the limit switch, and the system will prompt the overstroke alarm information.

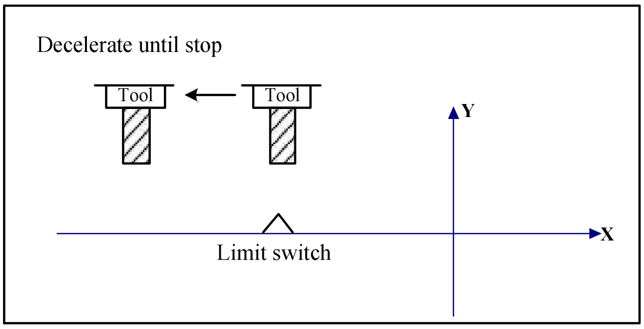


Fig. 2-5-1-1

Detailed description:

Overstroke during automatic operation

In the automatic operation mode, when the tool touches the limit switch in moving along a certain axis, all the axes will slow down and finally stop. At the same time, the overstroke alarm will be displayed and the program will stop at the overshoot segment.

Overstroke during manual operation

In the process of manual operation, as long as a certain axis of the machine touches the limit switch, the corresponding axis will immediately slow down and stop moving.

2.5.2 Software Overstroke Protection

The software stroke range is set by the data parameters **P66-P73**, and the coordinate value of the machine is taken as the reference value. If the moving axis exceeds the soft limit parameter setting, the overdrive alarm will appear. From setting connection power by the bit parameter **N0:11#6** to the manual return to reference point, whether stroke detection is carried out (0: No, 1: Yes). When the soft limit overstroke is set by the bit parameter **N0:11#7**, the overstroke (0: Before, 1: After) will alarm. After overstroke alarm, move the axis in the opposite direction under <Manual> mode. After moving out of the overstroke range, the alarm is removed.

2.5.3 Overstroke Alarm Removal

Removal method for hard limit overstroke alarm is as follows: In manual or pulse mode, move out the axis in the opposite direction (in case of positive overstroke, move out in the negative direction; in case of negative overstroke, move out in the positive direction).

2.6 Stroke Inspection

Use stored stroke inspection 1 and stroke inspection 2 to specify the areas where 2 tools are unaccessible to.

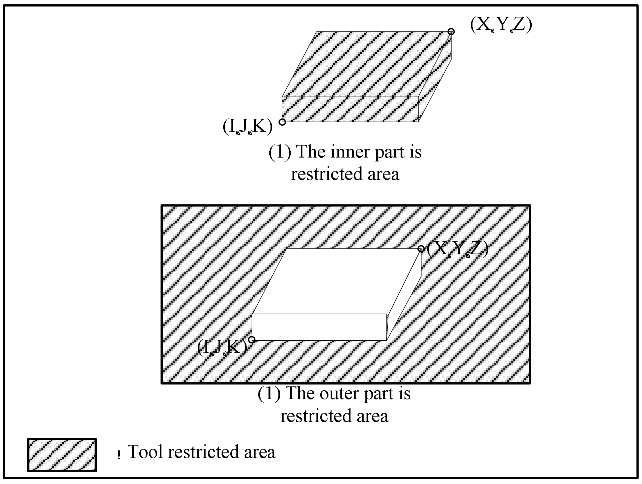


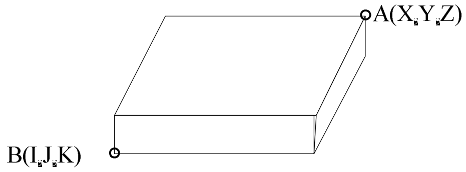
Fig. 2-6-1 Stroke inspection

When the tool exceeds the stored stroke limit, an alarm is displayed and the machine slows down and stops.

When the tool enters the restricted area and brings an alarm, the tool can move in the opposite direction as the tool enters.

Detailed description:

1. Stored stroke inspection **1**: set the boundary by the data parameters **P66-P73** beyond which lies the restricted area which is generally set as the maximum stroke of the machine by the machine manufacturer.
2. Stored stroke inspection **2**: set the boundary by the data parameters **P76-P83** or the program code within or beyond which can be set as the restricted area which is set by the bit parameter **NO:11#0** (0: inside of restricted area; 1: outside of restricted area).
   1) When using parameters to set the restricted area: points A and B in the figure below must be set.



X>I,Y>J,Z>K

X-I> ζ (Minimum command increment unit)

Y-J> ζ (Minimum command increment unit)

Z-K> ζ (Minimum command increment unit)

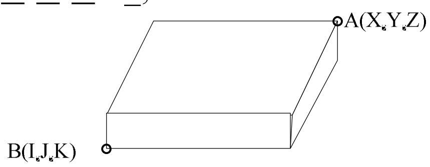
Fig. 2-6-2 Restricted area creation or change with parameter

When the restricted area is set by the data parameters P76-P83, the data must give the distance in the machine coordinate system (output increment) in the smallest command increment unit.

2) When using program instructions: G12 prohibits the tool from entering the restricted area; G13 allows the tool to enter it.

Each G12 in the program must have a separate program segment instruction. The following commands are used to create or change the restricted area.

G12 X_Y_Z_I_J_K _;



X>I,Y>J,Z>K

X-I> ζ (Minimum command increment unit)

Y-J> ζ (Minimum command increment unit)

Z-K> ζ (Minimum command increment unit)

Figure 2-6-3 Restricted area creation or change with program

Specify the distance in the machine coordinate system (input increment) in the smallest input increment if it is set by G12.

The programmed data is converted into a numeric value of the smallest code unit in the smallest increment, and such value is set in the parameter.

Example 1: Inside of restricted area (bit parameter **NO:11#0**=0)

N1 G12 X50 Y40 Z30 I20 J10 K15;	Set point A (50, 40, 30) and point B (20, 10, 15) in the restricted area for tools.
N2 G01 X30 Y30 Z20;	Linear interpolation to (30, 30, 20)
N3 G13;	Storage stroke detection function cancellation
N4 G01 X50;	

Example 2: Outside of restricted area (bit parameter **NO:11#0**=1)

N1 G12 X50 Y40 Z30 I20 J10 K15;	Set point A (50, 40, 30) and point B (20, 10, 15) in the restricted area for tools.
N2 G01 X10 Y-10 Z-10;	Linear interpolation to (10, -10, -10)
N3 G13;	Storage stroke detection function cancellation
N4 G01 X50;	

3) Checkpoint of restricted area: Confirm the checkpoint location (tip or top of the tool case) before programming the restricted area. As shown in Fig. 2-6-4, if the checkpoint is A (tool tip), the distance "A" should be set to the data in the storage function inspection; if checkpoint is B (tool case), distance "B" should be set as the data in the storage function inspection. When the checkpoint is A (tip) and the length of the tool varies with the tool, the restricted area should be set according to the longest tool, so as to ensure safe operation.

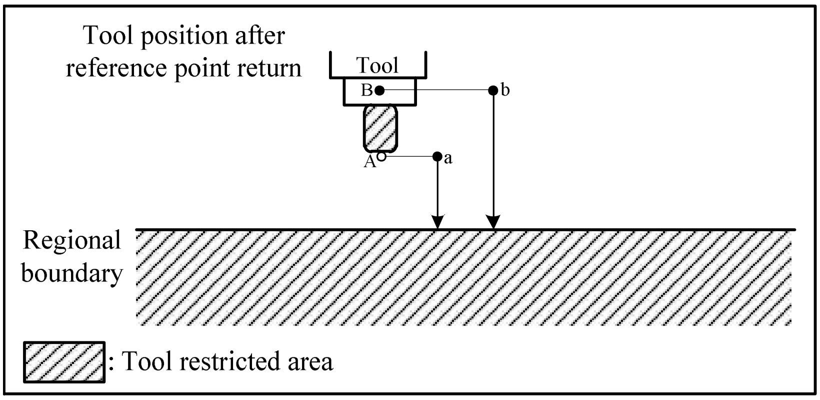


Fig. 2-6-4 Restricted area setting

4) Overlap of restricted area for tools: Restricted area can be set in an overlapping manner. As shown in the figure 2-6-5.

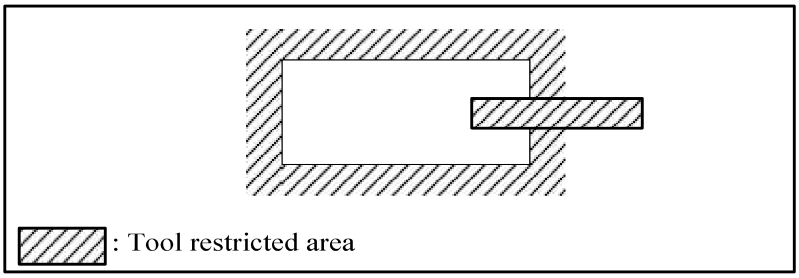


Fig. 2-6-5 Restricted area overlapping setting

Unnecessary limits should be set outside the stroke of the machine.

5) In case of bit parameter **NO:11#6=0**, the effective time of the restricted area is: When the power is switched on and the manual reference point return or the automatic reference point return is performed by code G28, the restricted area boundary will take effect.
In case of bit parameter **NO:11#6=1** and the power is switched on, if the reference position is in the restricted area, an alarm will appear immediately (only effective in G12 mode of stored stroke limit 2).

6) Alarm removal: If the tool enters a restricted area and an alarm appears, the tool can only move in the opposite direction. To eliminate the alarm, move the tool in the opposite direction until it exits the restricted area and reset the system. After the alarm is removed, the tool can move forward or backward. Please refer to section 2.5.2 of "Part II Operation instructions" of this Manual for details.

7) An alarm will immediately appear when G13 changes to G12 in the restricted area.

8) Use position parameter **NO:10#1** to set whether stroke detection should be conducted

Volume II Operation Instructions

before movement. In case of position parameter **NO:10#1=0**, no stroke detection is performed before movement. In case of position parameter **NO:10#1=1**, stroke detection is performed before movement.

Chapter III Interface Display and Data Modification and Setup

3.1 Position Display

3.1.1 Five Ways of Position Page Display

Press 

key to enter the position page display which has five interfaces: [relative coordinate], [absolute coordinate], [comprehensive coordinate], [program monitoring] and [monitoring] to be checked by corresponding soft keys as follows:

1) Absolute coordinate: Press [relative coordinate] soft key to display the position of the current tool in relative coordinate (see Fig. 3-1-1-1).

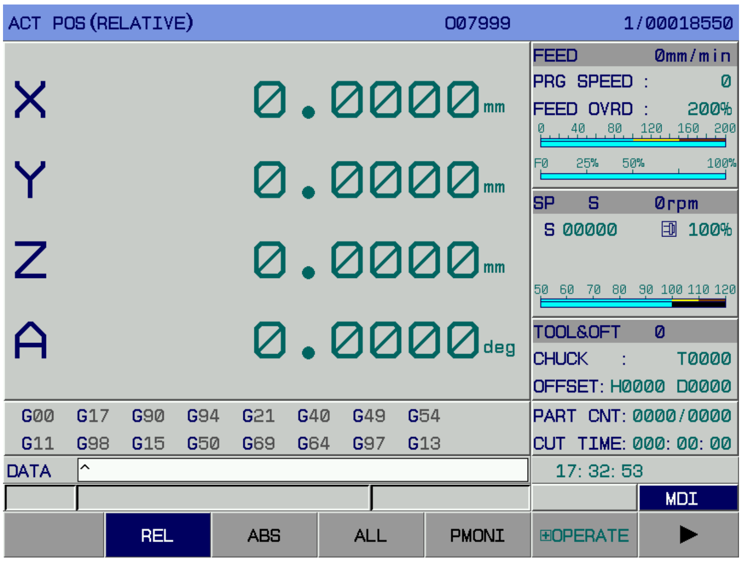
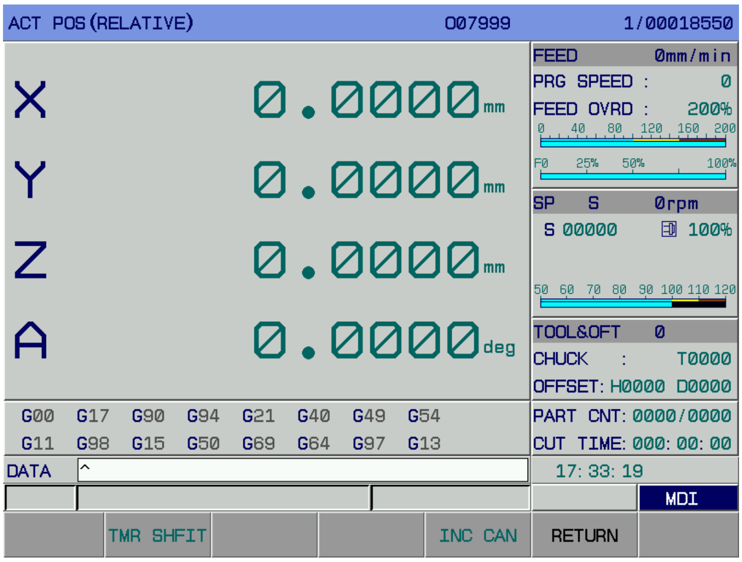


Fig. 3-1-1-1

A: Press [operation] soft key in the interface of relative coordinate, and [relative zeroing] to make the relative coordinates all return to zero.

B: Press [time switch] to switch display in [general time], [cutting time], [cycle time], [power-on time] and [running time].

Volume II Operation Instructions



2) Absolute coordinate: Press [Absolute coordinate] soft key to display the position of the current tool in absolute coordinate (see Fig. 3-1-1-2).

Fig. 3-1-1-2

3) Comprehensive coordinate: Press [comprehensive coordinate] soft key to enter the interface and display simultaneously:

(A) Position in relative coordinate;
(B) Position in absolute coordinate;
(C) Position in machine coordinate;
(D) Offset (displacement) from pulse interruption;
(E) Speed component;
(F) Residual movement amount (**displayed only in automatic, entry and DNC modes**).
The display page is as shown in Fig. 3-1-1-3.

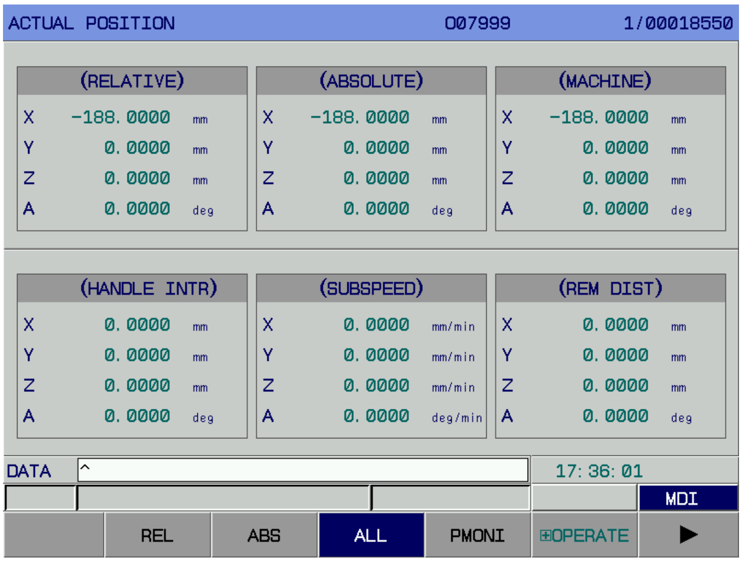
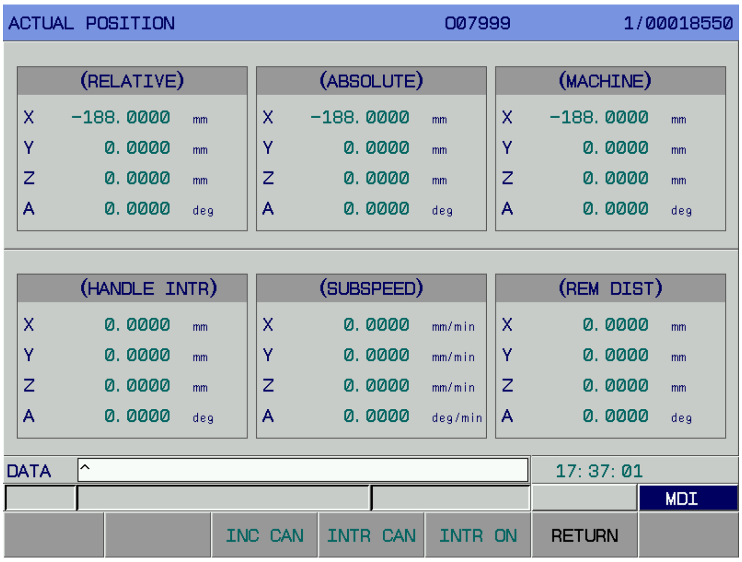


Fig. 3-1-1-3



1. Press [operation] soft key in the comprehensive interface, and press [relative zeroing] to make the relative coordinates all return to zero.

2. Press [operation] soft key in the comprehensive interface, and press [interruption zeroing] to make the pulse interruption coordinates all return to zero.

3. Press [operation] soft key in the comprehensive interface, and press [interruption on/off] to control the on/off state of the pulse interruption function.

4) Program monitoring

Press the soft key of [program monitoring] to enter the interface of [program monitoring]. This interface can simultaneously display the absolute coordinates and relative coordinates of the current position, modal information of the current running program and the running program segment, as shown in Fig. 3-1-1-4.

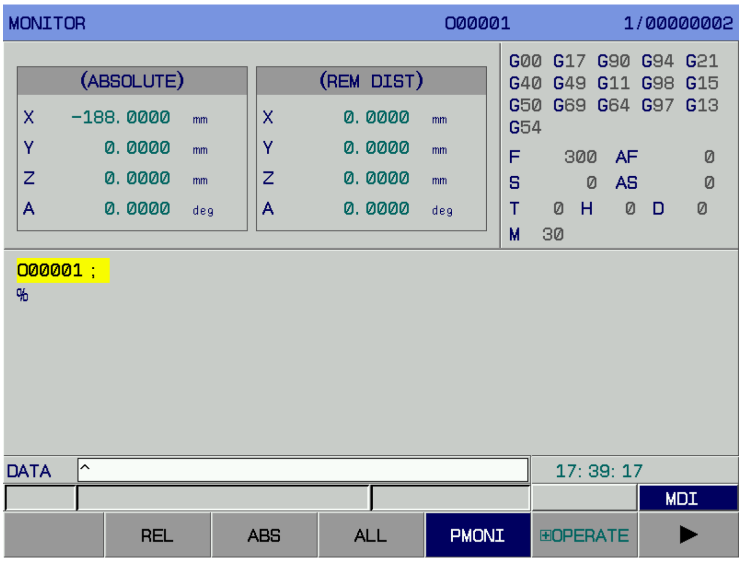
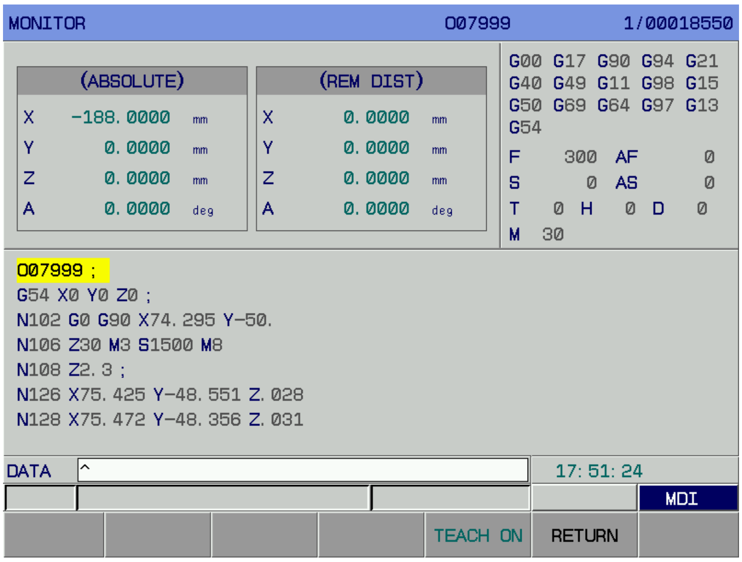


Fig. 3-1-1-4



Note 1: the modal display in the program monitoring display interface can be set by the parameter NO.: 23#6. In case of BIT6=0, the interface does not display the modal code, but display the machine coordinate value in the original position.

Note 2: the intermediate coordinate system is a relative coordinate system under four modes of <Zeroing>, <Single-step>, <Manual> and <MPG>; in three modes <Automatic>, <Entry> and <DNC>, the intermediate coordinate system is the residual movement.

Note 3: Press【Operation】soft key in the program monitoring interface, and press【Demonstration on/off】 to control the on/off state of the pulse demonstration function.

5) Monitoring interface

When the system selects the Ethernet bus communication mode, press  key to enter the position page display, and press the [monitoring] soft key to enter the [monitoring] interface; this interface can simultaneously display the machine coordinate in current position, multi-turn position, encoder value, grating position, motor speed and load (%, means the percentage of rated load); through this interface, it is easy to conduct machine debugging and real-time monitoring of current running state of the servo. The display page is as shown in Fig. 3-1-1-5.

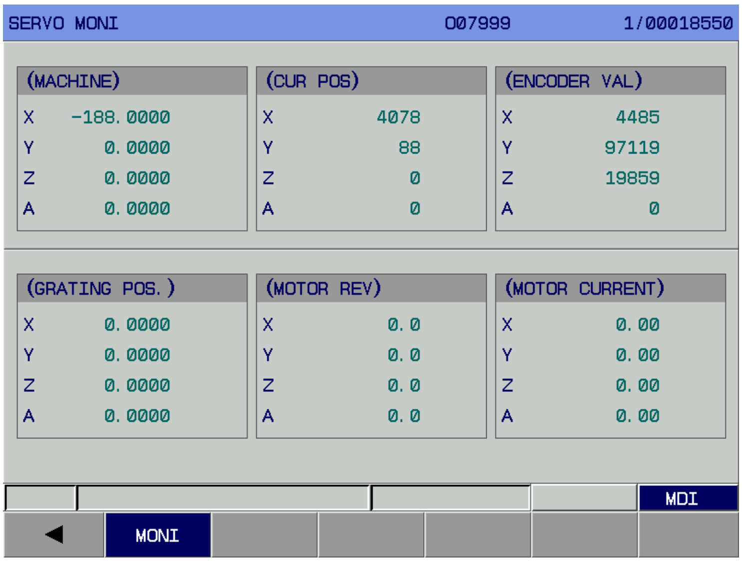


Fig. 3-1-1-5

Note 1: the modal display in the program monitoring display interface can be set by the parameter NO.: 23#6. In case of BIT6=0, the interface does not display the modal code, but display the machine coordinate value in the original position.

Note 2: the intermediate coordinate system is a relative coordinate system under four modes of <Zeroing>, <Single-step>, <Manual> and <MPG>; in three modes <Automatic>, <Entry> and <DNC>, the intermediate coordinate system is the residual movement.

3.1.2 Display Information Such As Processing Time, Number of Parts, Programming Speed, Override and Actual Speed

On the [absolute] and [relative] mode interfaces displayed at <Position>, the programming rate, actual rate, feed override, fast rate, G function code, tool offset, number of pieces processed, cutting time, spindle speed override, spindle speed and machining tool can be displayed, as shown in Fig. 3-1-2-1.

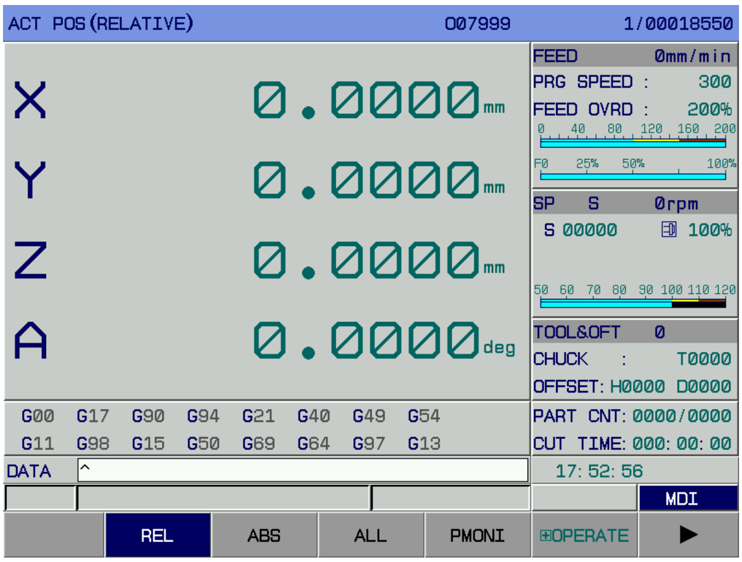


Fig. 3-1-2-1

Its specific meaning is as follows:

Rate: In processing, the actual rate of processing after override.

Programming rate: The rate specified by the F code in the program.

Feed override: The override selected by the feed override switch.

Fast override: The default panel control override in the system, or the override selected by the fast feed override band switch.

G function code: The value of the G code being executed currently.

Tool offset: H0000tool length compensation for current machining program; D0000 radius compensation for current program.

Number of pieces processed: When the program is executed to M30 or M02 in automatic or DNC mode, the number of processed pieces should be increased by 1. Otherwise, it will not be increased.

Cutting time: When the automatic operation starts, start timing in hours, minutes, seconds.

S00000: instruction speed. After pressing the direction key ↓ on the page of relative and absolute coordinates, position the cursor at “S 00000” (spindle speed); and then S value can be altered (the alteration range is 0-P258).

T0000: the tool number specified by T code in the program.

Note: Power-off memory of number of pieces processed.

Reset method for number of pieces processed and cutting time:

1) Switch to the location interface and select the entry mode.

2) Press the direction key ↓ to position the cursor to the column of number of pieces processed, input date to press INPUT key for confirmation; If press INPUT key directly, the number of pieces processes will be reset.

3) Use Page-up and Page-down keys to switch to cutting time.

4) Press INPUT key: Cutting time resetting.

Note 1: an encoder must be installed on the spindle when the actual speed of the spindle is displayed.

Note 2: actual rate = F value of programming rate × override. In case of G00, the running speed of each axis is set by the data parameters P88 - P92, which can be adjusted by the fast override. The dry running speed is set by the data parameter P86.

Note 3: programming rate display for each feed per revolution: It is displayed when the program segment of feed per revolution is being executed.

Note 4: the total number of processed parts can be set by the data parameter P356, and the total number of parts to be processed can be set by the data parameter P357.

3.1.3 Relative Coordinate Resetting and Centering

The relative coordinate position is reset as per the following operation steps:

1) Enter any interface with relative coordinates display (see Fig. 3-1-3-1);

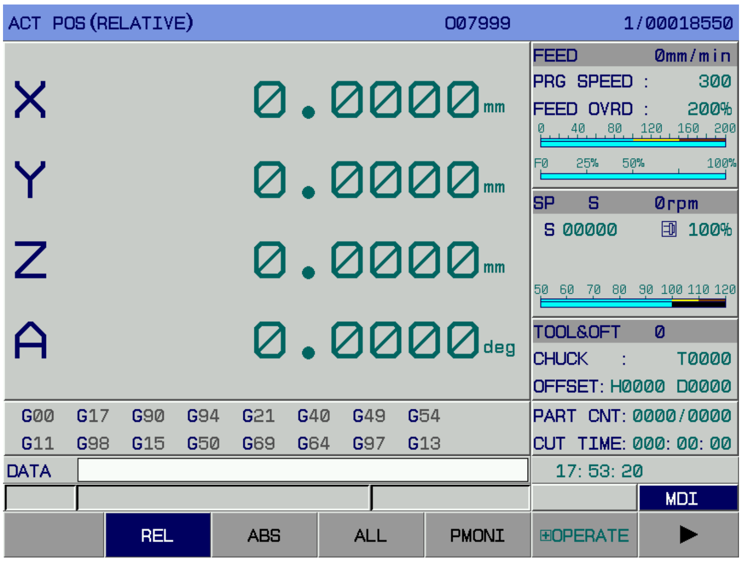
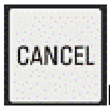


Fig. 3-1-3-1

2) Zeroing operation: Press the "X" key, and X flashes in the interface. Press  key, and it can be seen that the relative coordinate value of X direction has been reset (see Fig. 3-1-3-2).

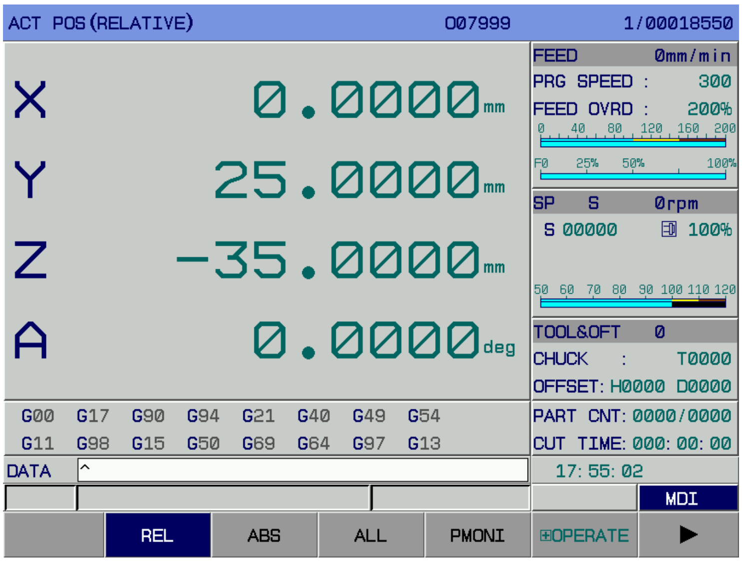
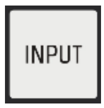
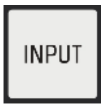


Fig. 3-1-3-2

3) Centering operation: Press the "X" key, and X flashes in the interface. Press  key, and it can be seen that the relative coordinate value of X direction has been divided (the relative coordinate value of this axis is divided by 2).

4) Coordinate setting: Press the "X" key, and X flashes in the interface. Press  key to confirm the data to be set which will be input into the coordinate system.

5) The resetting methods of Y axis and Z axis are the same as above.

3.2 Program Display

Press 

key on the panel to enter the program page, including five interfaces of [⊞program], **[MDI]**, [Present/module], [Present/times] and [Directory]. The corresponding soft key can be used to view and alter the interfaces, as shown in Fig. 3-2-1.The details are as follows:

1) Program display

Press the [⊞program] soft key to enter the program display interface, in which the program on the page of the program segment being executed in the memory is displayed (see Fig. 3-2-1).

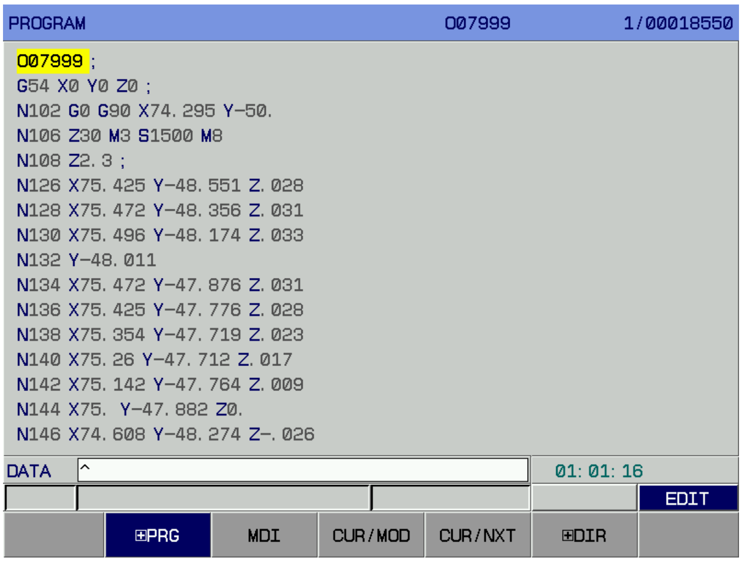


Fig. 3-2-1

After the soft key [⊞program] is pressed again, the interface enters the editing and alteration page of the program (see Fig. 3-2-2).

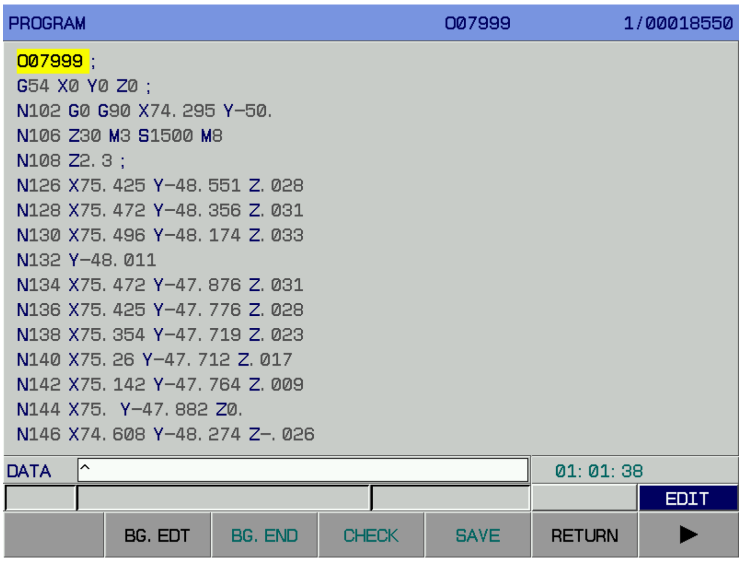
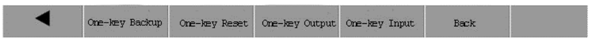


Fig.3-2-2

Press [▶] to go to the next page 

Press [▶] to go to the next page 

Press [◀] key to return to the previous page 

Note: The [Debugging] function can only be used in automatic mode.

[B edit] and [B end] functions can only be used under automatic mode and **DNC** mode (background editing function). [B edit] functions are the same as the editing program under the <**Edit**> mode; see Chapter 10 *Program editing operation* in "Part II Operation instructions"; after editing, press [B end] to store or [Back] to exit the editing interface.

2) MDI input display

Press [MDI] soft key to enter MDI display interface; it is allowed to edit and execute multiple segments of program under the entry mode; the program format and the editing program are the same. **MDI** running is applicable to simple test program operation (see Fig. 3-2-3).

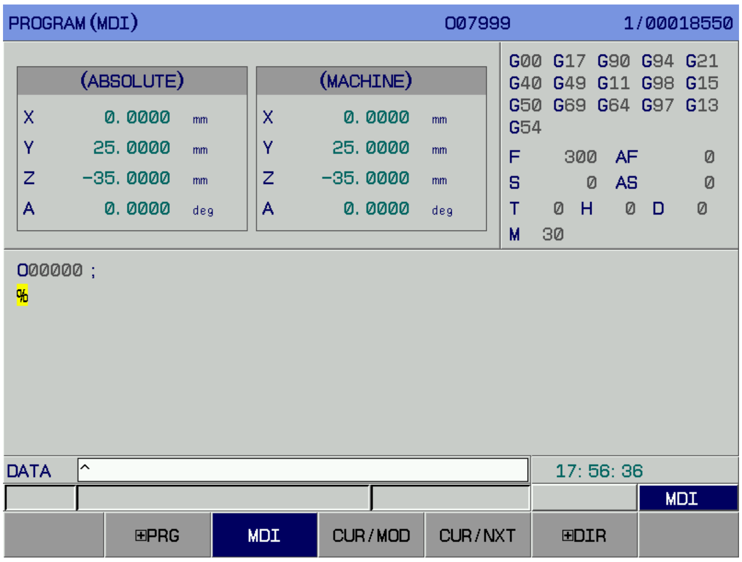


Fig. 3-2-3

3) Program (present/module) display

Press [Present/module] soft key to enter [Present/module] display interface and display the code value and current mode value of the program segment being executed. MDI data input and operation can be carried out under the entry operation mode (see Fig. 3-2-4).

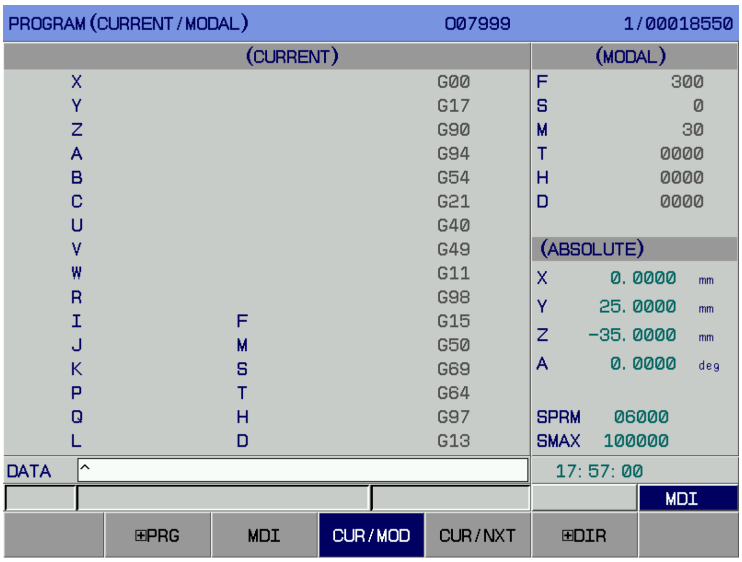


Fig. 3-2-4

4) Program (present/times) display

Press [Present/times] soft key to enter [Present/times] display interface and display the code value and of the program segment being executed and the program to be executed in next segment (see Fig. 3-2-5).

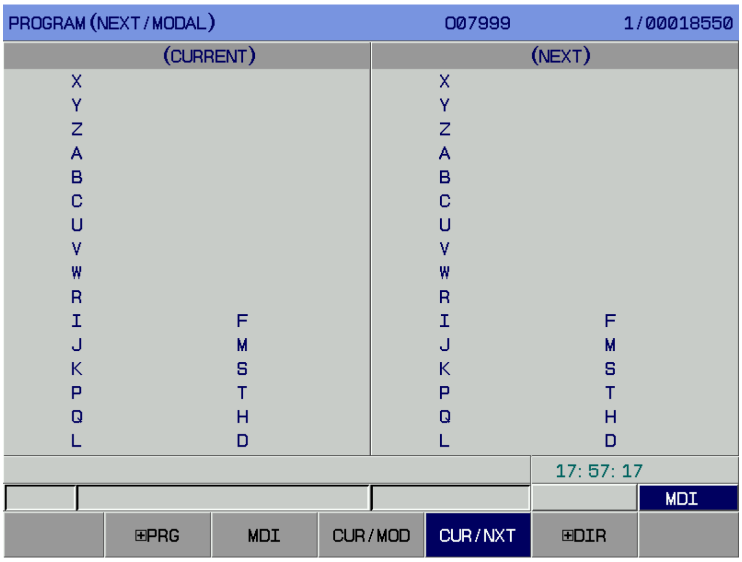


Fig. 3-2-5

5) Program directory display

I. Press [Directory] soft key to enter the program (system directory) display interface, to display the contents as follows (see Fig. 3-2-6).

(a) Program number: Number of programs stored (including subprograms) / maximum number of programs stored.

(b) Storage capacity: Storage capacity of stored programs/total program storage capacity.

(c) Program directory list: Display the program number stored in the program in sequence.

(d) Preview the program in which the current cursor lies.

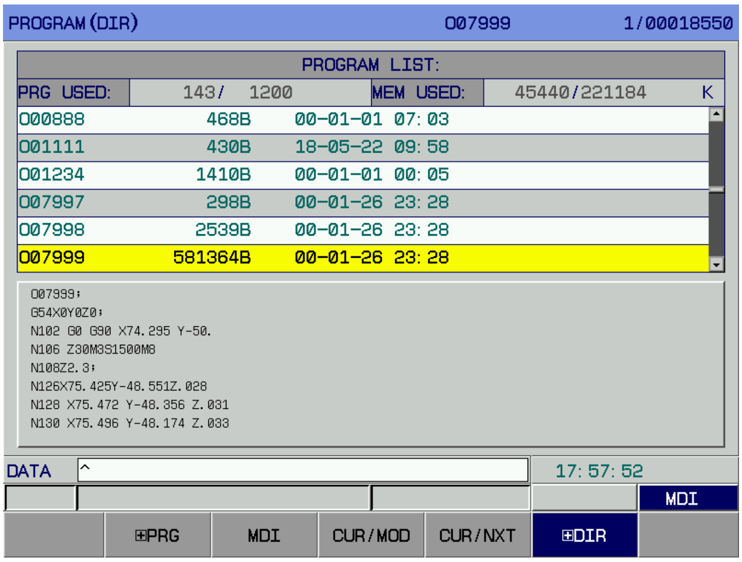


Fig. 3-2-6

II. Press [Directory] soft key again to enter the program (USB directory) display interface, to display the contents as follows (see Fig. 3-2-7).

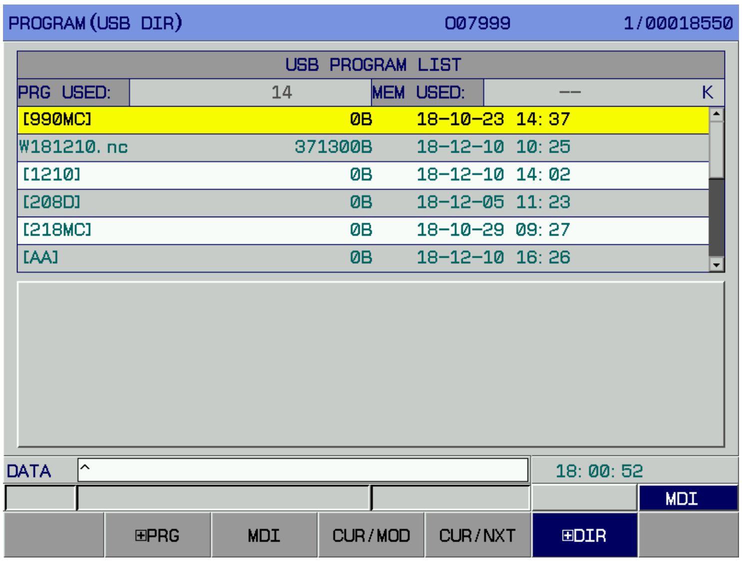
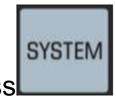


Fig. 3-2-7

Description: Show all program numbers in memory by the Page-Up and Page-Down keys. If the program name exceeds six digits or is non-conforming, the program name cannot be previewed.

3.3 System Display

Press 

key to enter the system page including five interfaces such as [⊞ offset], [⊞ parameter], [⊞ macro variable], [Pitch error compensation] and [⊞ bus configuration], which can be switched and displayed through the corresponding soft keys. Details are shown in Fig. 3-3-1-1-1.

3.3.1 Offset Display, Alteration and Setting

3.3.1.1 Offset display

Press [⊞ offset] soft key to enter the offset display page to display as follows (see Fig. 3-3-1-1-1).

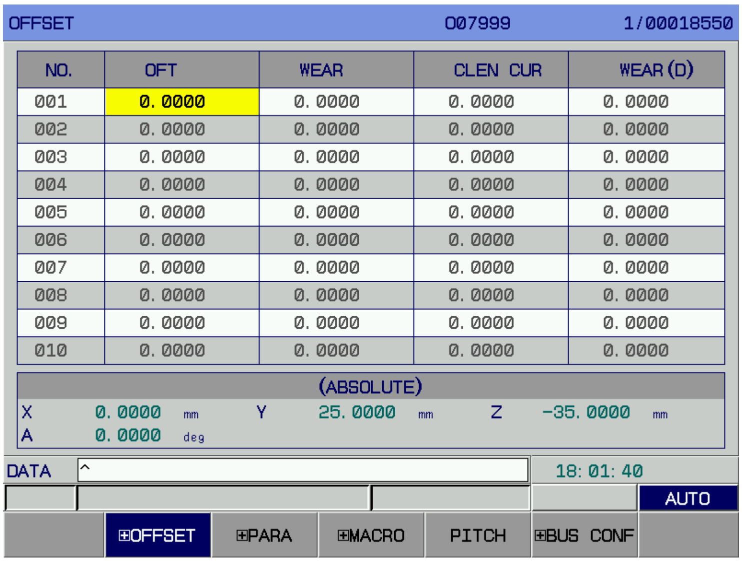


Fig. 3-3-1-1-1

Press the [⊞ offset] soft key in the figure above to enter the offset operation interface (see Fig. 3-3-1-1-2 and Fig. 3-3-1-1-3).

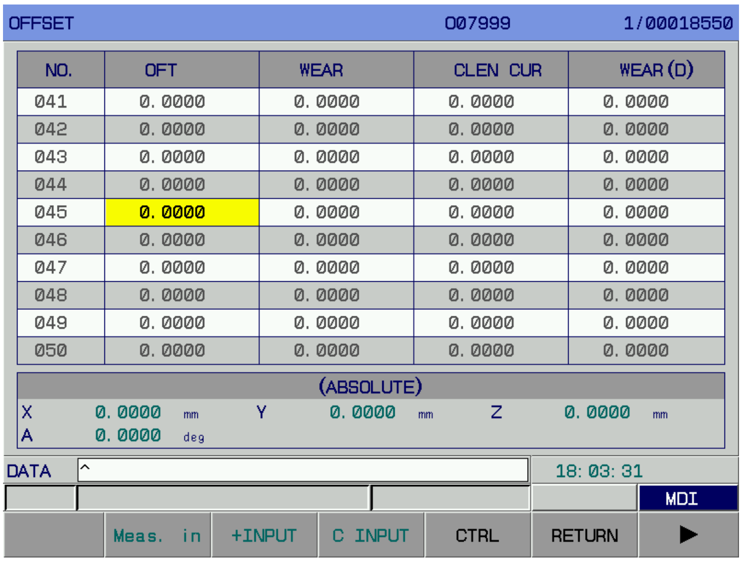


Fig. 3-3-1-1-2

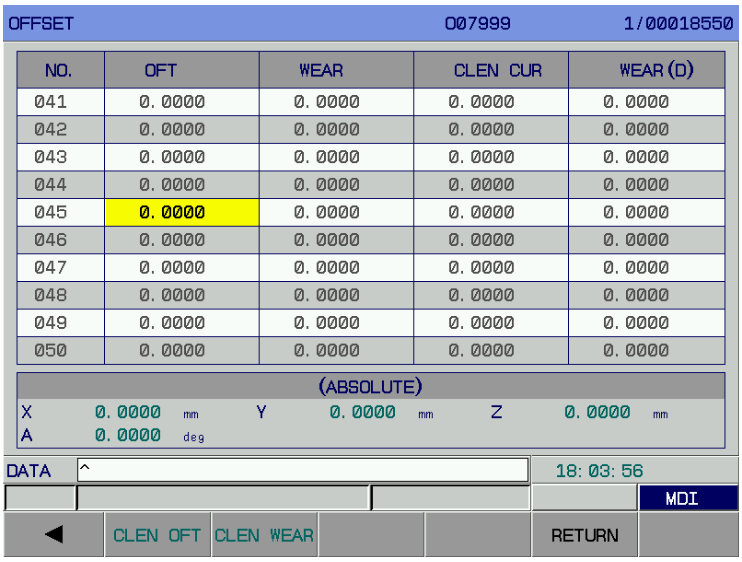


Fig. 3-3-1-1-3

The compensation quantity can be entered directly or added or subtracted from the value at the current position. Shape (H) represents tool length compensation, wear (H) represents tool length wear, shape (D) represents tool radius compensation, and wear (D) represents tool radius wear. Press [Clear shape] and [Clear wear] to clear all tool length compensation, wear, tool radius compensation and wear values.

3.3.1.2 Alteration and setting of offset value

The method of setting tool offset at the offset interface is as follows:

1) Press [⊞ offset] soft key to enter the offset display page.

2) **Move** the cursor to the compensation number to be entered.

Method 1: press the Page-Up and Page-Down keys to display the page where the compensation amount needs to be altered; press the direction key to move the cursor to locate the compensation number to be altered.

Method 2: after inputting the compensation number, press SEARCH key to locate.

3) In any mode, enter the compensation amount. Press INPUT key or [Input] soft key to confirm.

4) In any mode, input the compensation amount and press the [+Input] or [-Input] soft key. The system will automatically calculate and display the compensation amount.

Note 1: when the tool offset is changed, the new offset cannot take effect immediately. It only takes effect after H code or D code of the its compensation number in the instruction is executed.

Note 2: the user can alter the tool compensation value at any time during program running, but must ensure the alteration is completed before such tool compensation number running to make it take effect in time during the program running.

Note 3: if the length compensation quantity needs to be added with the relative coordinate value of Z axis, just write the compensation quantity after Z axis, and the system will superpose it automatically.
For example: Input Z 10, and the compensation is the current relative coordinate value of the Z axis plus 10.

3.3.2 Parameter Display, Alteration and Setting

3.3.2.1 Parameter setting

Press the [⊞ parameter] soft key to enter the parameter page display including two interfaces such as [Position parameter] and [Number parameter], which can be viewed or altered by the corresponding soft keys, as follows:

1) Press the [Position parameter] soft key to enter the position parameter interface (see Fig. 3-3-2-1-1).

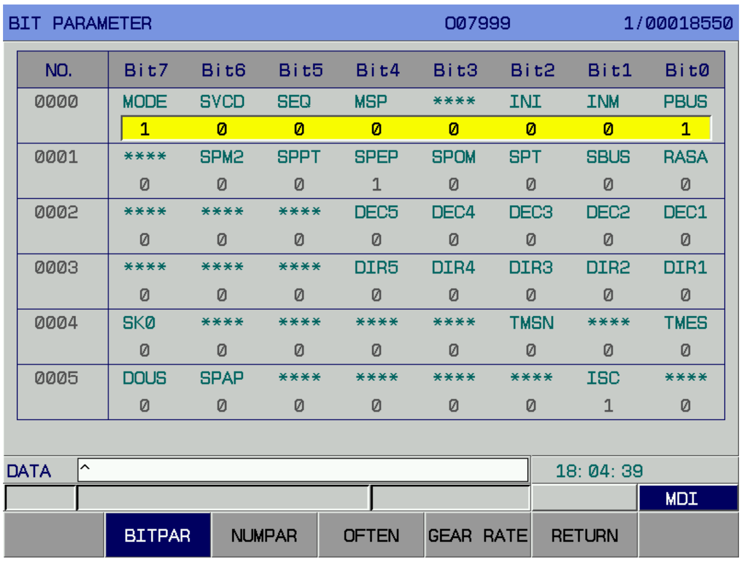


Fig. 3-3-2-1-1

For the specific definition of each parameter, please refer to “Appendix I Parameter description”.

2) Press the [Number parameter] soft key to enter the number parameter interface (see Fig. 3-3-2-1-2).

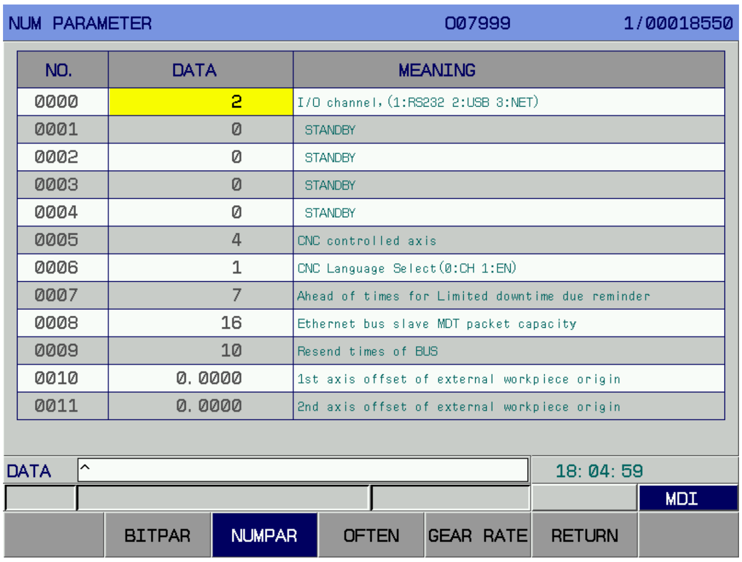


Fig. 3-3-2-1-2

For the specific definition of each parameter, please refer to “Appendix I Parameter description”.

3.3.2.2 Alteration and setting of parameter value

1) Select <Entry> operation mode.

2) Press the SYSTEM **key**, and then press the [⊞ parameter] soft key to enter the parameter display page.

3) Move the cursor to the position of the parameter number to be altered:

Method 1: press the Page-Up and Page-Down keys to display the page where the parameter needs to be set; press the direction key to move the cursor to locate the parameter to be altered.

Method 2: after inputting the parameter number, press SEARCH key to locate.

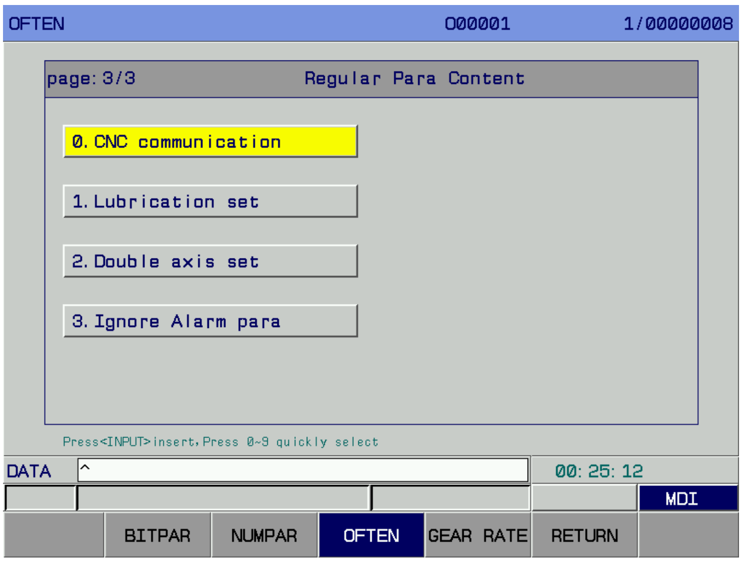
4) Enter the new parameter value with the number key and press the INPUT key to confirm.

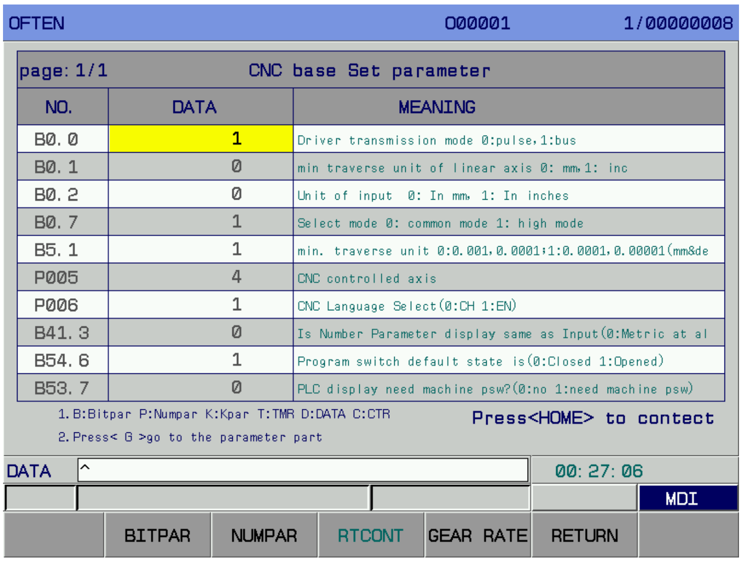
5) The system pops up a prompt box (depending on the different level parameters, it is necessary to open the parameter switch and enter the password permission of the corresponding level)

6) Depending on the different level parameters, it is necessary to open the parameter switch and enter the password permission of the corresponding level and

Press INPUT key to **confirm, to input and display parameter value.**

3.3.2.3 Common parameter directory

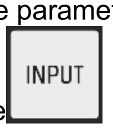




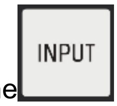
1) Select <Entry> operation mode.

2) Press the 

key, and then press the [⊞ parameter] soft key to enter the [Common parameter directory] page.

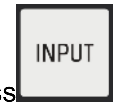
3) Press the Page-Up and Page-Down keys to find the page where the parameter needs to be set, press the 0-9 number keys to quickly select or press the 

key to select the block and enter, where B: bit parameter P: number parameter K: K parameter T: TMR D: DATA C: CTR then press letter [G] key to jump to the parameter interface where the cursor selects parameter.

4) Enter the new parameter value with the number key and press the 

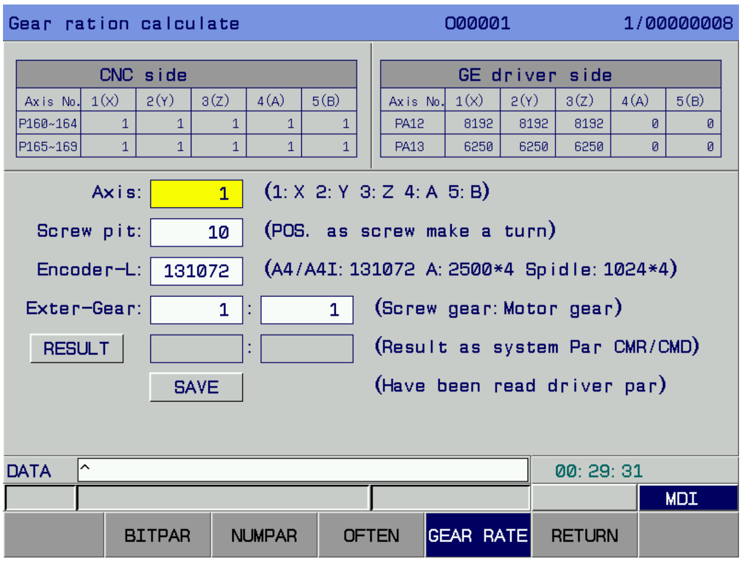
key to confirm.

5) The system pops up a prompt box (depending on the different level parameters, it is necessary to open the parameter switch and enter the password permission of the corresponding level)

6) Depending on the different level parameters, it is necessary to open the parameter switch and enter the password permission of the corresponding level and Press 

key to **confirm, to input and display parameter value.**

3.3.2.4 Electronic gear ratio calculation of system



Note: This interface is convenient for users to calculate the electronic gear ratio of the system. By default, the electronic gear ratio at the drive unit side should be 1:1 (note: the bit parameter **N0: 5#1 is the minimum moving unit**), and the calculation result can be written to the system side.

3.3.3 Macro Variable Display, Alteration and Setting

3.3.3.1 Macro variable display

Press the [⊞ macro variable] soft key to enter the macro variable page display including two interfaces such as [User variable] and [System variable], which can be viewed or altered by the corresponding soft keys, as follows:

1) for **User variables page**, press the [User variable] soft key to enter the user variables interface (see Fig. 3-3-3-1-1).

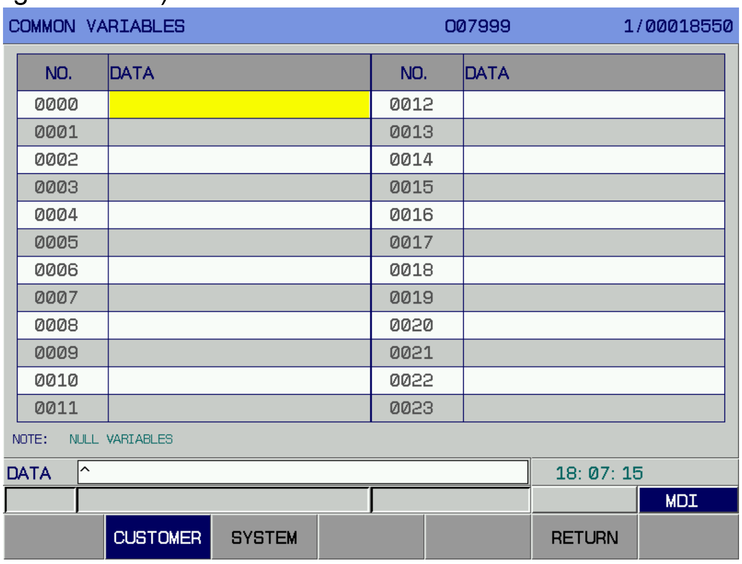


Fig. 3-3-3-1-1

2) for **System variables page**, press the [System variable] soft key to enter the system variables interface (see Fig. **3-3-3-1-2**).

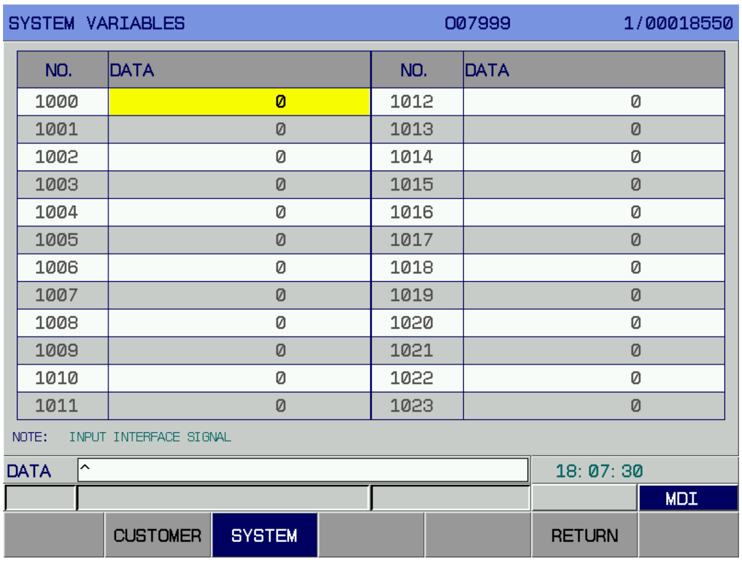


Fig. 3-3-3-1-2

For details about description on and use method for macro variables, see “4.7.2 Macro variables” in the Programming Part.

3.3.3.2 Macro variable alteration and setting

1) Select <Entry> operation mode.

2) Press the SYSTEM **key**, and then press the [⊞ macro variable] soft key to enter the macro variable display page.

3) Move the cursor to the position of the variable number to be altered:

Method 1: press the Page-Up and Page-Down keys to display the page where the variable needs to be set; press the direction key to move the cursor to locate the variable to be altered.

Method 2: after inputting the variable number, press SEARCH key to locate.

4) Enter the new value with the number key.

5) Press INPUT key to confirm, to input and display the value.

3.3.4 Pitch Error Compensation Display, Alteration and Setting

3.3.4.1 Pitch error compensation display

Press [Pitch error compensation] soft key to enter the pitch error compensation display page to display as follows (see Fig. 3-3-4-1-1).

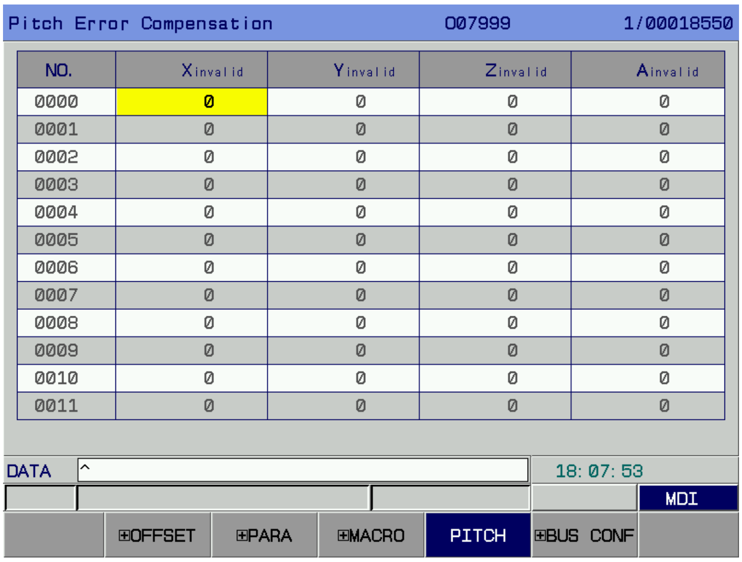


Fig. 3-3-4-1-1

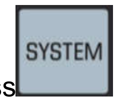
3.3.4.2 Pitch error compensation alteration and setting

1) Set the pitch error compensation number of each axis by the data parameters **P216-P220**, and the pitch error compensation spacing by the data parameters **P226-P230**.

2) Under the <Entry mode>, enter the compensation amount for each point in turn.

Note: For the setting of the pitch error compensation, please refer to "Part IV Installation connection" in *PLC and Installation & Connection Manual of GSK218MC CNC System*.

3.3.5 Bus Servo Parameter Display, Alteration and Setting

Press 

key to enter the system page, and switch and display the [⊞ bus configuration] interface through corresponding soft key. See Fig. 3-3-5-1 for details.

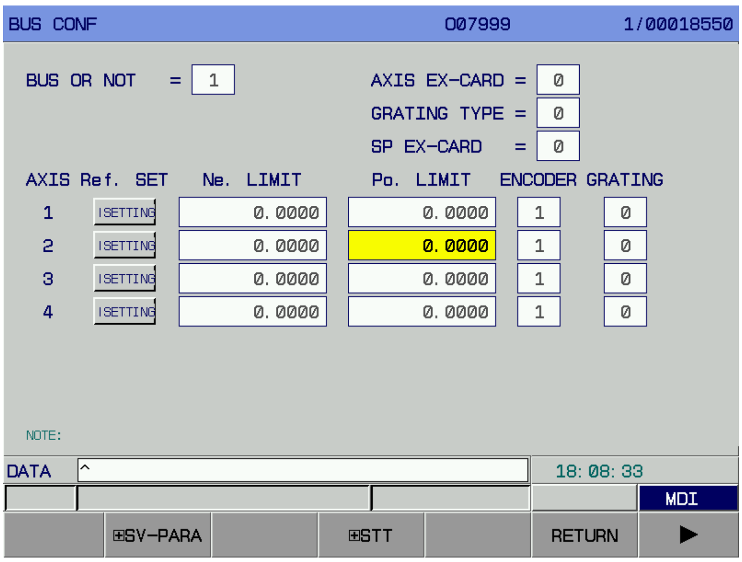


Fig. 3-3-5-1

[⊞ bus configuration] interface operating instructions

Press the [⊞ bus configuration] soft key to enter the bus configuration interface as shown in Figure 3-3-5-1 to view and alter related parameters. Specific operation methods and steps are as follows:

1. Enter the <**Input**> operation mode;
2. Press the up, down, left and right keys to move the cursor to the item to be altered;
3. Make alteration as described below;:

1) Bus or not
0: the transmission mode of the drive unit is pulse 1: the transmission mode of driver unit is bus

2) Encoder type
0: incremental 1: absolute type

3) Axis expansion card
0: no expansion card 1: with expansion card

4) Grating type
0: incremental 1: absolute type

5) Spindle expansion card
0: no expansion card 1: with expansion card

6) Multi-turn absolute zero setting

a) First, set the gear ratio, feed axis direction and the zeroing direction the system end, and power on again after power-off.

b) In MDI mode, set "Bus or not" to 1 and "Encoder type" to 1 in the bus configuration interface, and manually move each axis to set the machine zero position.

c) Move the cursor to Setting, press <Input> button twice according to the prompt, the zeroing indicator lights up; record the current position of the absolute encoder of each axis motor as the machine zero point, power on the system when it is powered off and the zeroing indicator still lights up. The negative boundary and the positive boundary can be manually set according to the maximum travel of the machine tool, so that the current machine tool absolute coordinate is offset forward or backward by one value, and finally the position parameter **No.61#6** is set to 1 to validate the positive and negative limit.

d) Grating configuration or not. Decide whether the grating is configured or not for each axis. 0: without grating, 1: with grating.

4. Press INPUT key to confirm.

Note 1: After the machine zero is set, modification to the zeroing direction of each axis of the system, the moving direction of the feed axis, the servo and the system gear ratio will lead to the lost zero and resetting of the machine zero.

Note 2: After the machine zero is reset, other reference points will be influenced, such as the second and third reference points which must be reset.

3.3.5.1 Servo parameter display

Press [⊞ bus configuration] to enter the servo debugging interface, and then press [⊞ servo parameter] soft key to enter the servo parameter interface. The contents displayed on this interface are shown in Fig. 3-3-5-1-1.


SETTING (SERVO): O07999 1/00018550

No.	X	Y	Z	A
0000	****	****	****	****
0001	65	65	65	0
0002	3. 10	3. 87	3. 87	0. 00
0003	0	0	0	0
0004	0	0	0	0
0005	150	200	200	0
0006	6	6	15	0
0007	100	100	100	0
0008	200	200	200	0
0009	190	150	102	0
0010	0	0	0	0
0011	350	350	350	0

Password(0 - 9999)
DATA ^ 18: 08: 58
MDI
GRADE CLR BACKUP COMEBACK RETURN

Fig. 3-3-5-1-1

3.3.5.1.1 Servo parameter alteration and setting

1) Select <Entry> operation mode.

2) Press the SETTING key to enter the <Set> interface and set the parameter switch to “1”.

3) Press SYSTEM key and the [⊞ bus configuration] to enter the servo debugging interface, and then press [⊞ servo parameter] soft key to enter the parameter setting and display interface.

4) Move the cursor to the current selection axis parameter #0, enter the password 315 (**0-42** parameters are visible and modifiable), press the input key to download the drive unit parameters to the system, and alter the servo parameters on the [Servo parameter] interface.

5) Move the cursor to the position of the parameter number to be altered:

Method 1: press the Page-Up and Page-Down keys to display the page where the parameter needs to be set; or press the direction key to move the cursor to locate the parameter to be altered.

Method 2: after inputting the parameter number, press SEARCH key to locate.

6) Press INPUT key to confirm, send the parameter value to the drive unit, and the status bar shows “Successful drive unit parameter downloading! ”.

7) Press the SAVE key to let the servo save the updated parameters. The status bar shows “Successful drive unit parameter storage! ”.

8) After setting all parameters, turn off the parameter switch.

3.3.5.1.2 Setting of parameters matching with servo and motor type

1) Select <Entry> operation mode.

Volume II Operation Instructions

2) Press the SETTING key to enter the <Set> interface and set the parameter switch to “1”.

3) Press SETTING key and the [⊞ bus configuration] to enter the servo debugging interface, and then press [⊞ servo parameter] soft key to enter the parameter display interface.

4) Move the cursor to the current selection axis parameter **#0**, enter the password **385**, press the input key to download the drive unit parameters to the system, and alter the servo parameters on the [Servo parameter] interface.

5) Move the cursor to parameter #1 and enter the value that matches the motor type:

6) Press INPUT key to confirm, send the parameter value to the drive unit, and the status bar shows “Successful drive unit parameter downloading! ”.

7) Press the SAVE key to let the servo save the updated parameters. The status bar shows “Successful drive unit parameter storage! ”.

8) After setting all parameters, turn off the parameter switch.

3.3.5.1.3 Servo parameter backup

1) Select <Entry> operation mode.

2) Press the SETTING key to enter the <Set> interface and set the parameter switch to “1”.

3) Press the SETTING key to enter the <Set> interface and enter **terminal user password and the password above level.**

4) Press SETTING key and the [⊞ bus configuration] to enter the servo debugging interface, and then press [⊞ servo parameter] soft key to enter the parameter display interface.

5) Select the [Backup] key to back up the parameters of the axis currently selected to the file DrvParXX.txt (XX axis number. For example: Backup X axis is documented as: DrvPar01.txt).

6) After setting all parameters, turn off the parameter switch.

3.3.5.1.4 Servo parameter recovery

1) Select <Entry> operation mode.

2) Press the SETTING key to enter the <Set> interface and set the parameter switch to “1”.

3) Press the SETTING key to enter the <Set> interface and enter **terminal user password and the password above level.**

4) Press SYSTEM key and the [⊞ bus configuration] to enter the servo debugging interface, and then press [⊞ servo parameter] soft key to enter the parameter display interface.

5) Select the [Recovery] key to recover the backup parameter file DrvParXX.txt of the axis currently selected to the servo drive (XX axis number/ For example: Backup X axis is documented as: DrvPar01.txt).

6) Press the SAVE key to let the servo save the updated parameters. The status bar shows "Successful drive unit parameter storage! ".

7) After setting all parameters, turn off the parameter switch.

3.3.5.1.5 Servo level zeroing

When the parameters are debugged, the servo parameters are too rigid to shake the machine. To avoid danger, the servo level zeroing function can quickly restore the servo parameters to the 0-level initial state parameters.

1) Select <Entry> operation mode.

2) Press the SETTING key to enter the <Set> interface and set the parameter switch to "1".

3) Press the SETTING key to enter the <Set> interface and enter **terminal user password and the password above level.**

4) Press SYSTEM key and the [⊞ bus configuration] to enter the servo debugging interface, and then press [⊞ servo parameter] soft key to enter the parameter display interface.

5) Select the [Level zeroing] key to restore all servo axis parameters to 0 level parameters.

6) Press the SAVE key to let the servo save the updated parameters. The status bar shows "Successful drive unit parameter storage! ".

7) After setting all parameters, turn off the parameter switch.

3.3.5.2 Servo debugging (only suitable for GE drive unit)

In order to ensure that the servo debugging function truly reflects the servo performance, please cancel the gear ratio at the drive side and cancel the compensation at the system side (including pitch error compensation and backlash compensation).

3.3.5.2.1 Interface composition

Press the [⊞ STT] soft key to enter the servo debugging tool interface. The contents displayed on this interface are shown in Fig. 3-3-5-2-1 - Fig. 3-3-5-2-2.

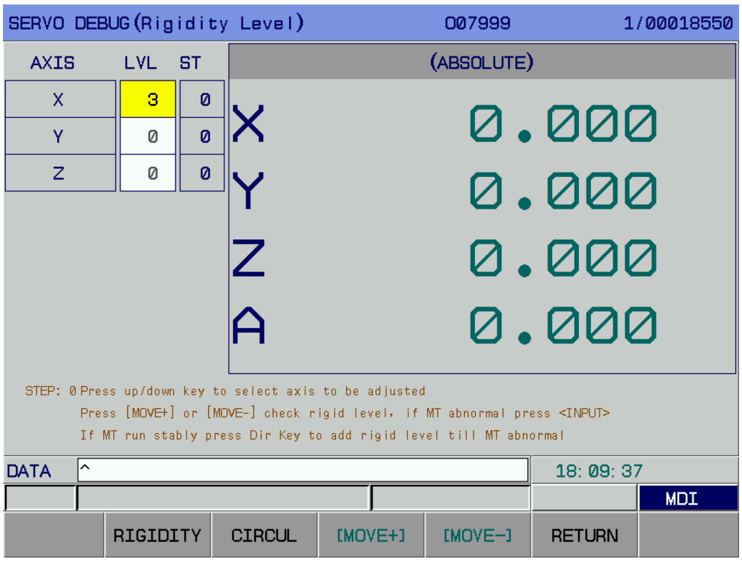


Fig. 3-3-5-2-1 Rigidity level interface

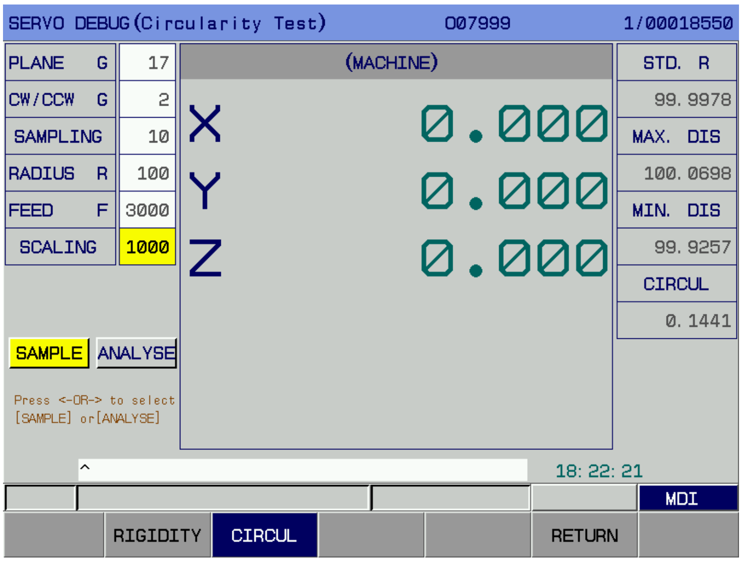


Fig. 3-3-5-2-2 Roundness test interface

Note: The coordinate axis display on the servo debug interface depends on the number of system control axes and the number of bus servo slaves, whichever is smaller.

3.3.5.2.2 Function Introduction

1.Rigidity level and parameter optimization operation function

This function is to set the servo parameters to the optimal servo performance state.

2. Roundness test

The roundness test simulates the circular cutting motion circle and collects the position information about the motor code disc to determine the synchronization of the servo axis response of the machine.

3.3.5.2.3 Operation instructions

1. Rigidity level debugging operation

Description: The debugging and setting of the rigidity level can only be performed on one axis at a time.

Operation keys:

A. [↑] **and** [↓] **key:** Select the axis.

Note: Once entering the optimization process, not to use the up and down arrow keys to change the axis in the current operation.

B. [←] **and** [→] **key: Decrease** or increase the rigidity level of the current axis, and decrease or increase the rigidity level by pressing once.

C. [Axis shift +] and [Axis shift -] soft keys: Move the current axis a certain distance in the positive or negative direction according to the speed set by the data parameter **P393**. This distance is set by the data parameter **P392**. Press the **[Axis shift +]** and **[Axis shift -]** soft keys to move the axis repeatedly to check whether the motor vibrates or abnormally sounds before entering the optimization process. However, once entering the optimization process, do not press **[Axis shift +]** and **[Axis shift]** continuously. -] soft keys to moves the axis to get the motor characteristic data.

Note 1: After entering the optimization process, press the [Axis Shift +] and [Axis Shift -] soft keys to move the axis and collect data.

Note 2: Non-professionals should not arbitrarily change the number parameters P392 and P393, or otherwise the optimization may fail.

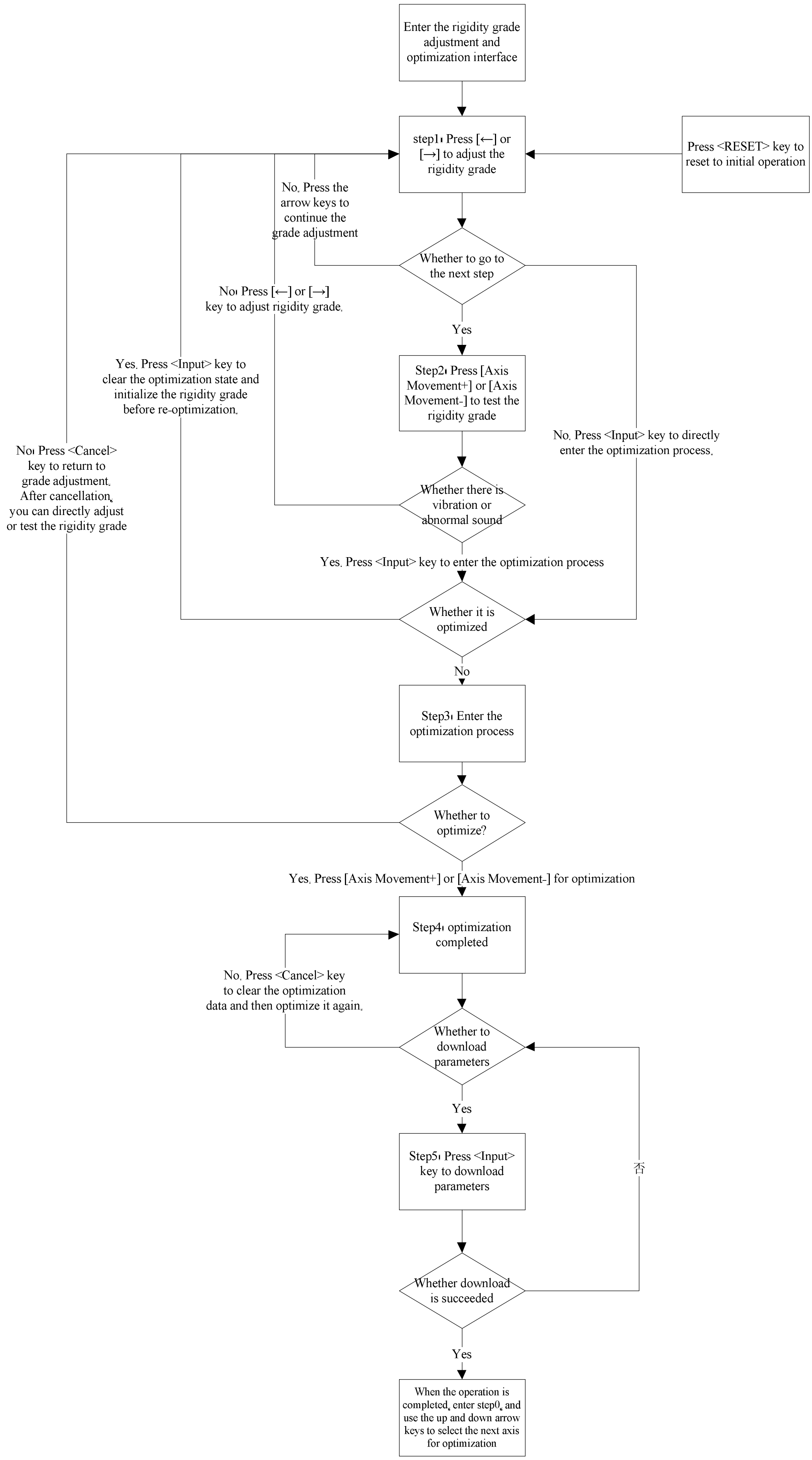
D. [INPUT] key: Confirm the operation or go to the next step.

E. [CANCEL] **Key:** Cancel certain operation or return to the previous step.

F. [RESET] **Key:** Reset operation, return to the initial operation step.

G. [SAVE] **Key:** Storage operation, store the optimized parameters.

Operating procedures: As shown below:

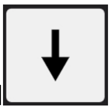


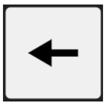
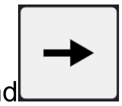
Volume II Operation Instructions

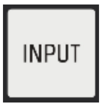
2. Roundness test

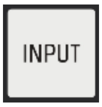
Operation keys:

A. Number key: Enter each parameter value;

B.  and  key: Select the parameter item;

C.  and  key: Select function (collection and analysis):

D.  key: Enter the parameter value or confirm and perform the operation;

E.  **key:** Clear the data and reset to the initial state.

Parameter item:

A. Plane: Select test planes G17, G18, G19;

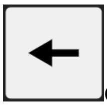
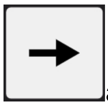
B. Clockwise and counterclockwise circle: Select the circle direction G02, G03;

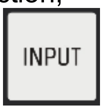
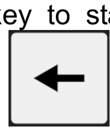
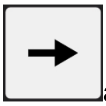
C Sampling period: Set the sampling period according to the radius of the circle and the feed speed. The larger the radius, the longer the sampling period should be; the slower the feed speed, the longer the sampling period is;

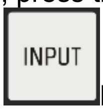
D. feed speed: The movement speed when testing.

E. Magnification times: Roundness analysis is for the error magnification.

Operation steps:

Step 1: After setting various parameters, press  or  arrow key to select the collection function;

Step 2: Press the  key to start the arc movement and start collecting data. After collection, press the  or  arrow key to select the analysis function.

Step 3: Press the  key to start the analysis function and output the roundness data and plot the circle error distribution as shown in the figure below.

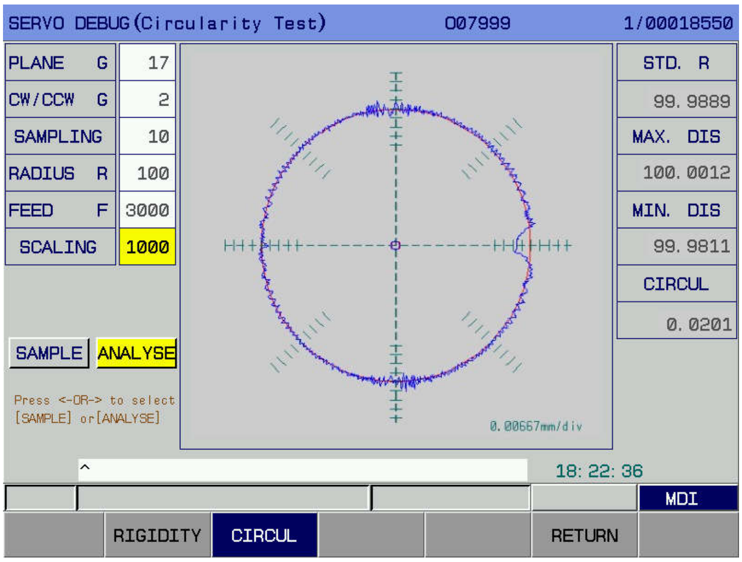


Fig. 3-3-5-2-3-1

Note: After rigidity level and parameter optimization function debugging, the roundness test tool is needed to test the synchronization of the current feed axes. The roundness of each plane can be considered as of good synchronization of the current servo axes within 6u via testing, and the parameter debugging is basically successful.

3.4 Setting Display

3.4.1 Setting Page

1. Page entrance

Press the SETTING key to enter the setting information display interface including five interfaces: [Set], [⊞ Workpiece Coordinate], [⊞ Centering and tool setting] [⊞ Data] and [Password], which can be viewed or altered through the corresponding soft keys. See Fig. 3-4-1-1 for details.

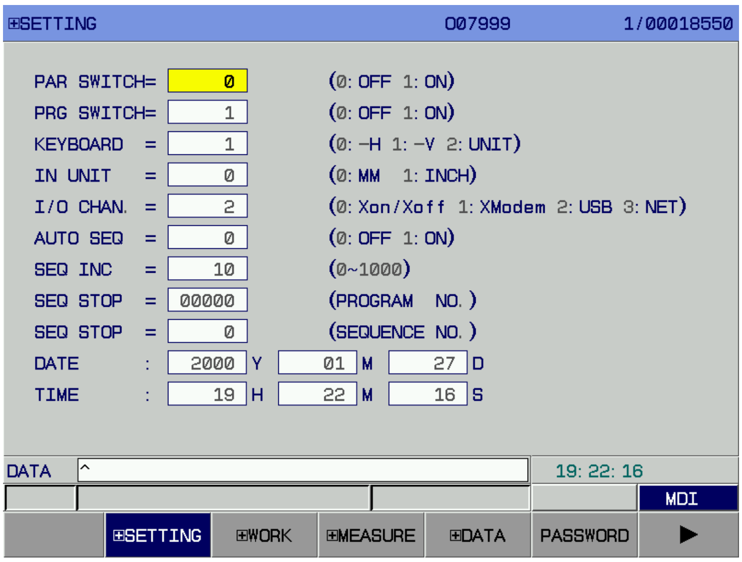


Fig. 3-4-1-1

2. [Set] interface operating instructions

Press the [Set] soft key to enter the setting interface as shown in Figure 3-4-1-1 to view and alter related parameters. Specific operation methods and steps are as follows:

(a) Enter <**Entry**> operation mode;

(b). Press the Page-up and Page-down keys to move the cursor to the item to be altered;

(c) Enter 0 or 1 as described below or use left or right key to make alteration;

1) Parameter switch

0: Turn off the parameter switch　　　1: Turn on the parameter switch

When setting the parameter switch to “0”, prohibit from altering or setting the system parameters, and the system alarm (0100: parameter written is valid) will cancel. When setting the parameter switch to “1”, the system will alarm (0100: parameter written is valid). Then, press the 

+ 

key to cancel the alarm (this operation is only valid on the setting interface).

2) Program switch

0: Turn off the program switch　　　1: Turn on the program switch

Prohibit from editing program when setting the program switch to “0”

3) Keyboard selection

0: 218MC-H　　　1: 218MC-V　　　2: 218MC

Note: In any mode, if you press the emergency stop key, you can also alter the keyboard selection.

4) Input unit

The input unit of the setting program is metric or imperial

0: Metric.　　　1: Imperial.

5) I/O channel

The user can set it according to the actual needs. If the user needs to use U disk DNC processing, set the channel to 2, and if using the Network port DNC for processing, set the channel to 3.

0: NET

1: RS232

2: USB

6) Automatic sequence number (SN)

0: When entering a program with the keyboard in edit mode, the system will not automatically insert SN.

1: When entering a program with the keyboard in edit mode, the system will automatically insert SN. The increment value of SN between each program segment is set by the data parameter **P210**.

7) SN increment

Set the increment value when SN is automatically inserted, ranging from 0 to 1000.

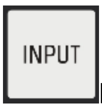
8) Stop SN

This function can be used to set a program to execute a single program segment pause during running to the specified segment. It is valid only when both the program number and the program segment SN are specified. For example: 00060 (program number) is the program number O00060; 00100 (SN) is the segment number N00100.

Note: When the program runs to the target segment and the stop SN is automatically set to -1, the single segment pause will not be executed.

9) Date and time

The user can set the system date and time here.

(d) Press 

key to input.

3.4.2 Workpiece Coordinate Setting Page

1. Press the [⊞ Workpiece coordinate] soft key to enter the coordinate system setting interface to display the contents as shown in Fig. 3-4-2-1.

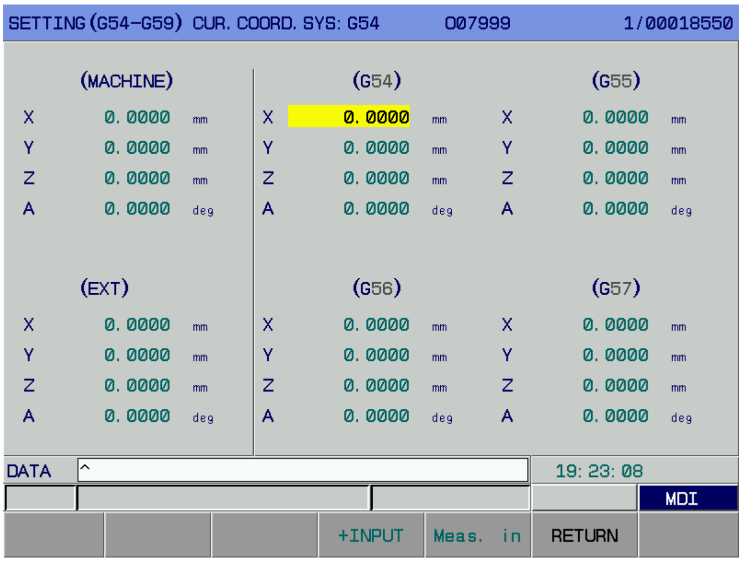


Fig. 3-4-2-1

In addition to the six standard workpiece coordinate systems (G54 to G59 coordinate systems), 50 additional workpiece coordinate systems can be used. As shown in the figure 3-4-2-2. Each coordinate system is viewed or altered by the Page-Up and Page-Down keys. For the operation of the additional coordinate system, see “4.2.9 Additional workpiece coordinate system” in the Programming Part.

Fig. 3-4-2-2

2. There are two ways to input coordinates:

1) After entering the interface in any mode, move the cursor to the coordinate system to be altered, and set the axis name of the value as required, press INPUT to confirm, and then set the value of the current machine coordinate system to the origin of G coordinate system, such as: Press “X” and then INPUT key, or press “X0” and then INPUT key. The system will automatically enter the X-axis machine coordinates of the point; in addition, such as: Enter X10(X-10) and press INPUT to represent X machine coordinate value +10(-10).

2) After entering the interface in any mode, move the cursor to the coordinate system axis to be altered, directly input the machine coordinate value of the workpiece coordinate system origin, and press INPUT to confirm.

3) After entering the interface in any mode, move the cursor to the coordinate system axis to be altered, directly input the machine coordinate value of the workpiece coordinate system origin, and press the soft key <Input> to confirm. In addition, press the <+Input> soft key after entering the coordinate value, and the system will automatically calculate and display the new coordinate value.

3. Coordinate system search method:

1) In any mode, enter the coordinate system and press SEARCH key to locate. For example: Enter “G56”.

2) In any mode, enter the following: “P6” or “P06”, press SEARCH key and locate the cursor to the additional workpiece coordinate system “G54 P06”.

3.4.3 Centering and Tool Setting Functions

Press the [⊞Centering and tool setting] soft key to enter the centering and tool setting function to display the contents on this interface as shown in Fig. 3-4-3-1.

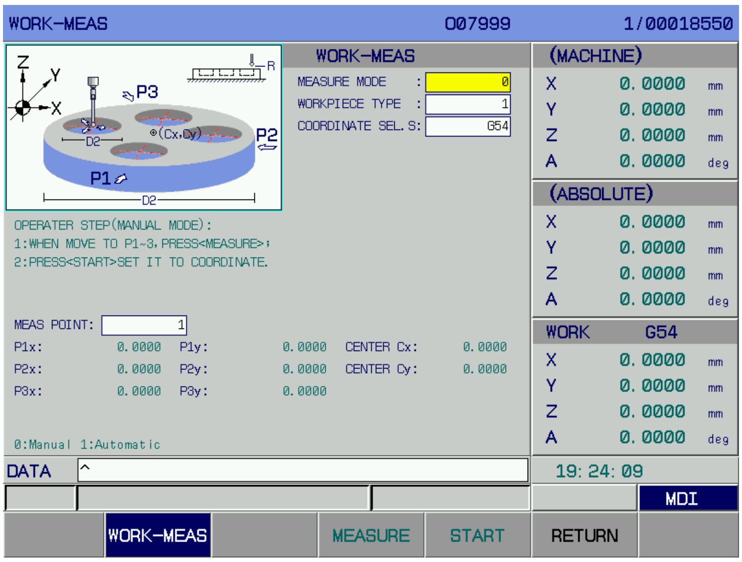


Fig. 3-4-3-1

3.4.3.1 Centering function introduction and operation instructions

Centering measurement: It mainly includes two types: manual centering and automatic centering.

Among them, the manual centering can only measure the hole or outer circle, the boss or groove; the automatic centering can measure the hole or outer circle, the boss or groove, the vector hole or outer circle, the vector boss or groove.

I. Manual centering

◆ **Interface display**

A. Hole or outer circle:

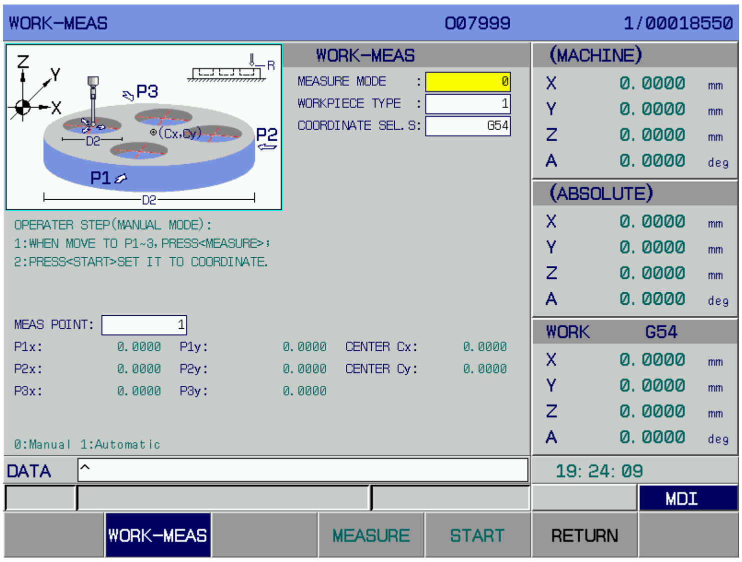


Fig. 3-4-3-1-1

B. Boss or groove:

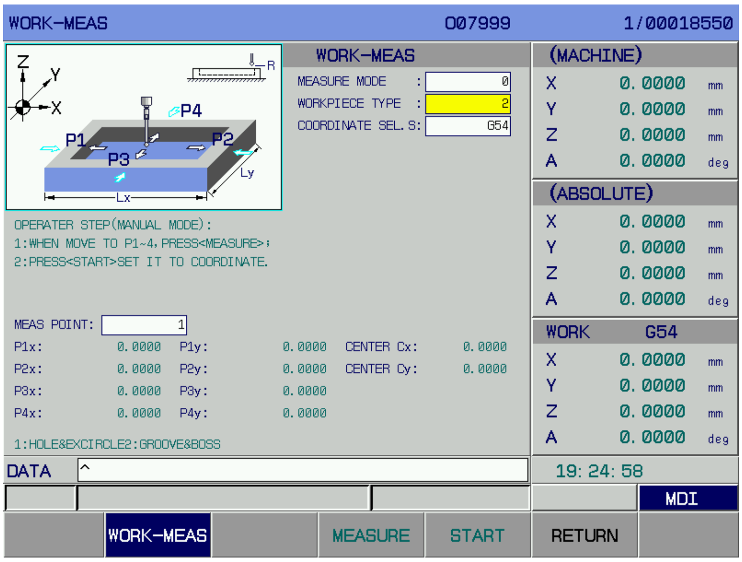


Fig. 3-4-3-1-2

◆ **Manual centering operation:**

A. Option description:

1. Measurement method:

0: Manual 1: Automatic

2. Workpiece type:

1: Hole or outer circle 2: Boss or groove

3. Coordinate system selection S:

G54-G59 G54 P1-P50 Set the midpoint to the coordinate system to be set after measurement.

4. Measurement points:

a) When the workpiece type is a hole or an outer circle: The number of measurement points is 3 (P1-P3), and the measurement is in no particular order: if three points coincide, then any one of them is the coordinate of circle center; if three points are on a straight line, the coordinate of circle center cannot be calculated, and one or all of the points need to be re-measured;

b) When the workpiece type is a boss or groove: The number of measurement points is 4 (P1-P4), and the measurement is in no particular order, where P1 and P2 are respectively two points in the X-axis; P3 and P4 are two points in the Y-axis, respectively. The X-axis center point coordinates are calculated by taking X coordinates of P1 and P2, and Y-axis center point coordinates are calculated by taking Y coordinates of P3 and P4.

B Operating procedures:

Step 1: Manually move the tool or center the optical edge finder to the 1st measurement point and press the <Measure> soft key.

Step 2: Repeat the action of step 1 until all measurement points have been measured (3 points in a circle, 4 points in a rectangle).

Step 3: Press the <Start> soft key to set the coordinates of the center point to the selected coordinate system.

II. Automatic centering

◆ **Interface display and parameter option description**

A. Common parameter options

1. Measurement method:

0: Manual 1: Automatic

2. Workpiece type:

±1: Hole and outer circle ±2: Boss and groove ±3: Vector hole and outer circle ±4: Vector boss and groove

[Note]-1: Hole +1: Outer circle-2: Groove +2: Boss-3: Vector hole +3: Vector outer 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. This tool offset number stores the radius compensation value of the tool during interpolation machining.

5. Tool offset E with empirical value:

The tool offset number with the empirical value stored. Do not assign the same value to E and T during programming.

6. Rough center coordinates Cx:

The X-axis absolute coordinate value of the rough center of the workpiece. If requiring to set the current point to a rough center, directly press <Input> key to enter a null value.

7. Rough center coordinates Cy:

The absolute value of the Y axis of the rough center of the workpiece. If requiring to set the current point to a rough center, directly press <Input> key to enter a null value.

8. Measuring point coordinates Z:

The absolute position of the Z axis in measurement. If the current point is a measurement point, directly press the <Input> key to enter a null value.

9. Profile size tolerance H:

The size tolerance value of the tested profile.

10. Radial clearance R:

When measuring the external profile, the distance of the probe from the target surface before Z axis moves. The power-on clearance is 8mm by default (0.3149inch).

11. Probe overstroke distance Q:

The overstroke of the probe. By programming, enter a value which is used by the probe as the distance beyond the target size, so as to find the surface. If not programmed, the default value is 10.0 mm (0.394 inch).

B. Hole and outer circle parameters

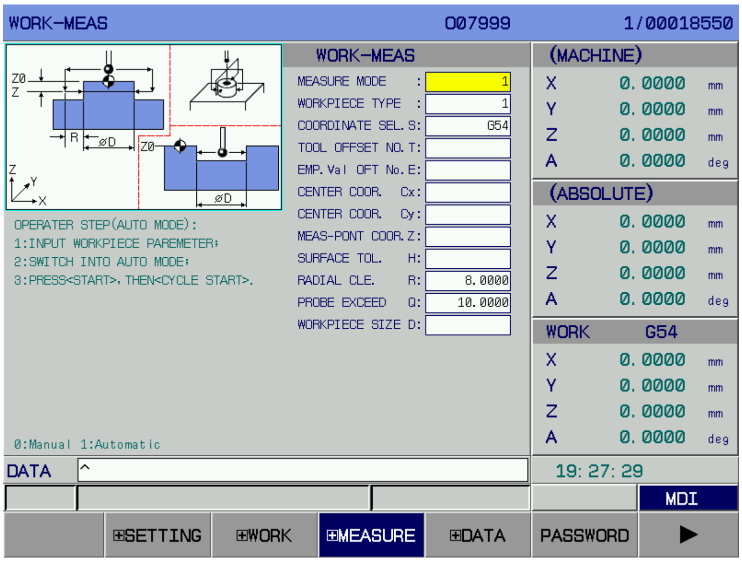


Fig. 3-4-3-1-3

1. Target size D:

The diameter of the hole or outer circle to be measured. This value cannot be null or 0.

C. Boss and groove parameters

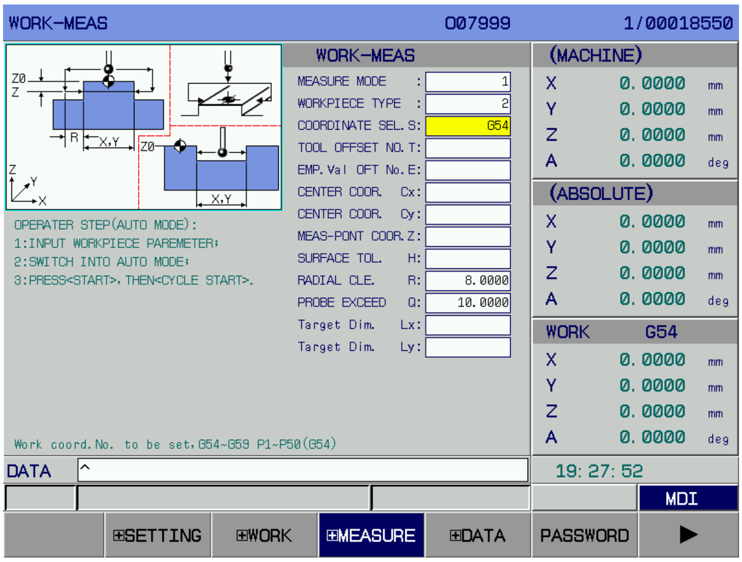


Fig. 3-4-3-1-4

1. Target size Lx:

The size of the X-axis profile to be measured. This axial measurement is not conducted when this parameter option is null or 0.

2. Target size Ly:

The size of the profile of the Y-axis to be measured. This axial measurement is not conducted when this parameter option is null or 0.

[Note]: Lx, Ly cannot be null or 0 at the same time.

D. Vector hole and outer circle

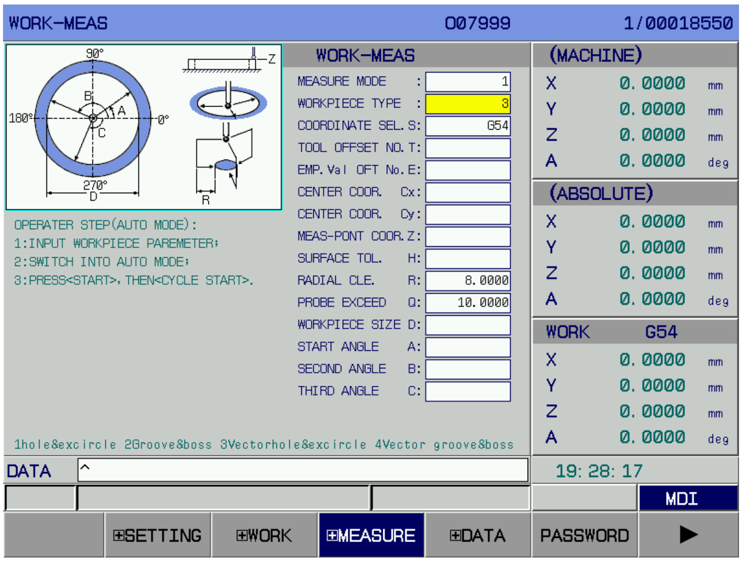


Fig. 3-4-3-1-5

1. Target size D:

The diameter of the hole or outer circle to be measured. This value cannot be null or 0.

2. Starting angle A:

The angle of the first vector measurement, starting from the X+ direction. If omitted, an alarm will appear.

3. The second point angle B:

The angle of the second vector measurement, starting from the X+ direction. If omitted, an alarm will appear.

4. The third point angle C:

The angle of the third vector measurement, starting from the X+ direction. If omitted, an alarm will appear.

[Note] The minimum difference between any two angles is determined by “#5=” in the program O09729. The default value is 5. If you want to change the minimum difference, just alter “#5=”.

E. Vector boss and groove

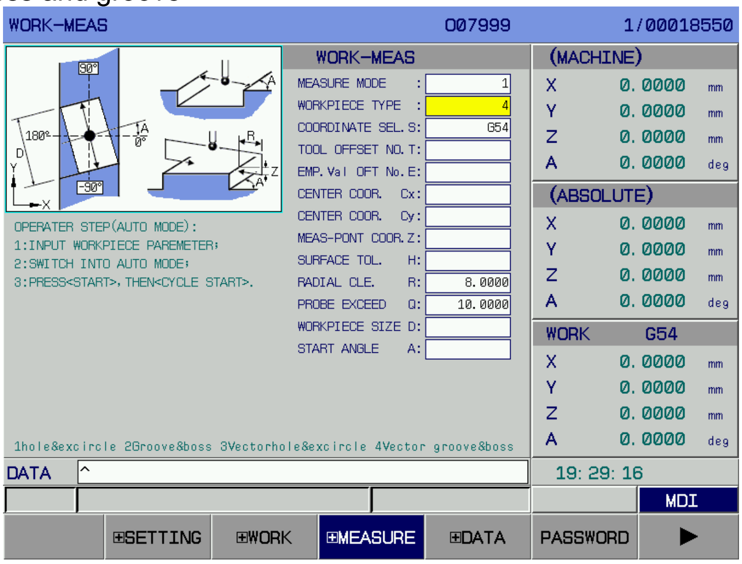


Fig. 3-4-3-1-6

1. Target size D:

The size of the profile to be measured. This axial measurement is not conducted when this parameter option is null or 0.

2. Starting angle A:

The angle at which the plane being measured starts from the X+ direction.

◆ **Data input**

A. Data input conditions

When automatic centering measurement is not started, data can be entered in any operation mode.

B. Input format

1. Data + <Input> enter the data to be entered.
2. Press <Input> to enter a null value directly.
3. When the current actions include the rough center coordinate X, the rough center coordinate Y, and the measurement point coordinate Z, input as follows:

① Directly press <Input> to enter a null value.

② X/Y/Z + <Input> enter the current absolute coordinate value of the selected axis;

③ X/Y/Z + data + <Input> enter the current absolute coordinate value + data of the selected axis;

④ Directly press [Measure] software to input the absolute value of the current axis;

⑤ X/Y/Z + [Measure] enter the absolute coordinate value of the current axis;

⑥ X/Y/Z + data + [Measure] enter the current absolute coordinate value + data of the selected axis.

◆ **Operation steps:**

Step 1: Set each centering parameter in turn.

Step 2: Switch to automatic mode.

Step 3: Press the <Start> soft key to start the automatic centering program, and then press the <Cycle start> key to run the measurement macro-program. After successful measurement, the system automatically sets the center point coordinate to the selected workpiece coordinate system.

3.4.3.2 Tool setting function and operation instructions

◆ **Interface display and function introduction**

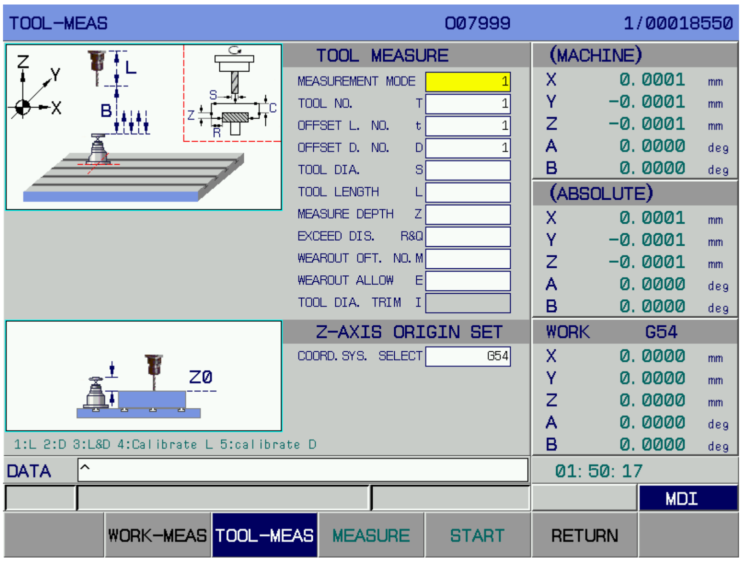


Fig. 3-4-3-2-1

The tool setting function consists of two parts: Automatic tool length measurement and Z-axis workpiece origin setting.

A. Tool measurement:

The automatic tool length measurement is to measure the length and diameter of different tools by the tool setting instrument installed on the workbench, and automatically set the length and diameter of each tool to the specified tool offset register, so as to ensure the different lengths and diameters of the tool can be used for processing correctly when running the same program.

B. Z-axis workpiece origin setting:

After tool length measurement, move the tool to the workpiece surface. Then, press <Measure> to set the current machine coordinate value as the origin to the selected workpiece coordinate system (G54-G59 G54 P1-P50).

◆ **Tool measurement**

A. Parameter option description

1. Measurement mode selection:

1: Length 2: Diameter 3: Length & Diameter 4: Length calibration 5: Diameter calibration.

2. Tool number T:

The tool number to be measured currently.

3. Tool length offset number H:

Store the offset number of the current tool length (the same as T by default).

4. Tool radius offset number D:

Store the offset number of the current tool diameter (the same as T by default).

5. Tool diameter S:

The diameter of the tool to be tested, when S is a “+” value, the tool is a right-handed cutting tool; when S is a “-” value, the tool is a left-handed cutting tool. When the tool radius offset number D is already a nominal tool diameter in the register, it is unnecessary to enter any value. After the tool number T is altered, the parameter value is cleared.

6. Tool length estimate L:

The length of the tool to be tested. When the tool length offset number H is already a nominal tool length in the register, it is unnecessary to enter any value. After the tool number T is altered, the parameter value is cleared.

[Note 1]: When the measurement mode is selected as the length calibration, such length must be entered and is the exact length of the standard tool (reference mandrel).

7. Measuring depth Z:

Depth from probe surface to diameter measurement position (default value -5.0mm [-0.20 inch]), and negative value indicates downward.

8, Overstroke amount R&Q:

The overstroke amount and the radial clearance when moving down to the probe side (default value 4.0 mm [0.16 inch]).

[Note 2] When measuring the length, it is the length in the length direction; when measuring the diameter, it is the radial overstroke amount; when measuring the length & diameter, the overstroke amount in length direction and the radial overstroke amount are the same.

9. Tool offset with damage identification:

A spare tool offset is used as the location for the tool damage identification.

10. Damage tolerance I:

Tool size adjustment to compensate for the cutting state of the tool. A positive value makes the actual radius smaller than the specified value; for example I=.01 makes the tool radius smaller by 0.01. It is also allowed to set the nominal tool radius value to zero by entering the nominal tool radius value.

[Note 3]: Used to set the diameter of the tool probe during diameter calibration.

B. Operation steps of measurement parameter input:

1. Item selection: Move the up and down cursor keys to select.
2. Data input: When automatic tool measurement is not started, the data can be altered by entering the data in any operation mode and then pressing the “Enter” key.

C. Operation steps

Step 1: Set each tool measurement parameter in turn

Step 2: Switch to automatic mode.

Step 3: Press the <Start> soft key to start the main program of automatic tool setting, and then press the <Cycle start> key to run the measurement macro-program. After successful measurement, automatically write the tool length and radius into the specified offset register.

◆ **Z-axis workpiece origin setting**

Note: Before Z-axis workpiece origin setting, please ensure that the current tool has been automatically measured, or otherwise it may cause machining errors, tool and equipment damage or even personnel safety accidents.

A. Coordinate system selection:

1. Setting range: G54-G59 G54 P1-P50
2. Data input: When automatic tool measurement is not started, move the cursor to the coordinate system selection item to enter the data in the following format under any operation mode:

a) Integer of 54-59;

Volume II Operation Instructions

b) G54-G59;

c) P1-P50. Then press <**Input**> key.

B. Workpiece origin setting:

1. Setting range: -9999.999-9999.999
2. Data input: When automatic tool measurement is not started, move the cursor to the coordinate system selection item under any operation mode, directly press [Measure] soft key to set current Z-axis machine coordinate value to the Z-axis of currently selected workpiece coordinate system, or enter the data in the following format:

a). Input format: Z;

b) Z + data; then press [Measure] soft key to set current Z-axis machine coordinate value + entered data to the Z-axis of currently selected workpiece coordinate system:

3.4.4 Data Backup, Recovery and Transmission:

Press [⊞ data] soft key to enter the setting (data processing) interface. User data (ladder diagram, ladder diagram parameters, system parameter value, tool compensation value, pitch compensation value, system macro variable, user macro-program, CNC part program) can be backup (saved) and restored (read); it can also be output and input through U disk or PC. The backup and recovery of data will not affect the part program stored in CNC (see Fig. 3-4-4-1).

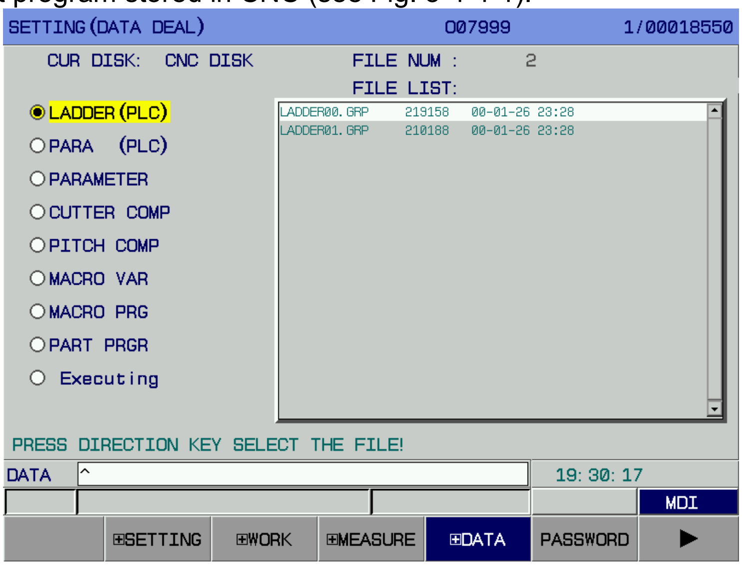


Fig. 3-4-4-1

Operation method:

1. Press [Password] soft key to set the corresponding level password in the password interface. Please refer to “3.4.5 Password permission setting and modification” for the password level corresponding to each data operation.
2. Press [⊞ data] soft key twice to enter the data processing operation interface, as shown in Fig. 3-4-4-2.

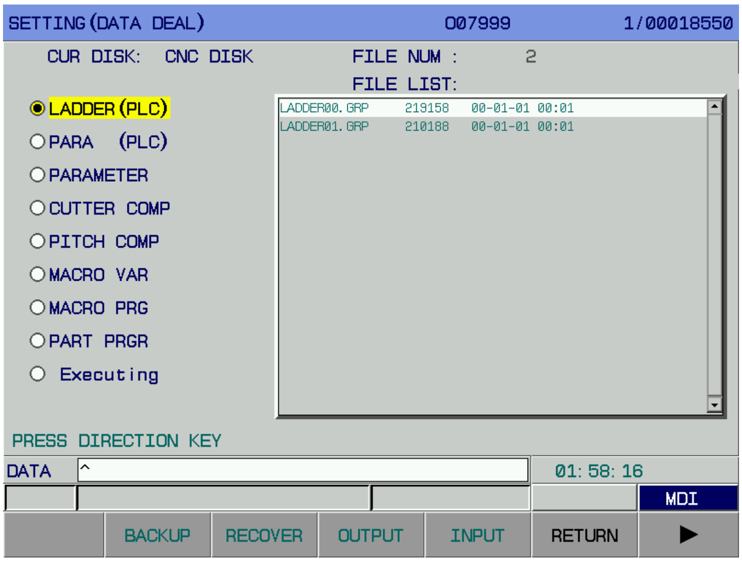
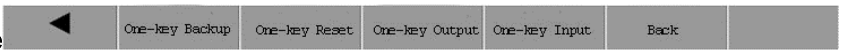


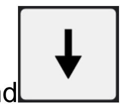
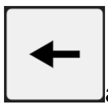
Fig. 3-4-4-2

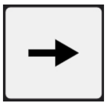
Press [▶] to go to the next page 

The functions of each operation item are shown in Table 3-4-4-1.

Table 3-4-4-1

Operation items:	Function description
Data backup	Separate data backup is applicable for the ladder diagram (PLC), parameters (PLC), system parameter value, **tool compensation value, pitch compensation value, system macro variables and other files.** After the data backup operation, the system will generate a backup file with the suffix.bak
Data recovery	The ladder diagram (PLC), parameters (PLC), system parameters value, tool compensation value, pitch compensation value, system macro variables and other files can be recovered separately. The data recovery operation is to read and recover the backup files in the system
Data output	The data output operation is capable of outputting data from the system disk to an external storage device.
Data input	Data input operation is capable of inputting data from external storage devices into the system disk.
One-click backup	Multiple data items can be backed up to the system disk simultaneously
One-click recovery	The backup files of multiple data items can be recovered simultaneously
One-click output	Multiple data items files in the system disk can be copied to the U disk
One-click input	Multiple files in the U disk can be copied to the corresponding data items of the system disk

3. Press  and  direction key to select the target file, press  and

 to switch the data item directory and file directory table.

4. Press the corresponding soft key to perform data backup, data recovery, data output, data input, one-click backup, one-click recovery, one-click output, and one-click input operations.

Note:

1) When I/O channel is set to a USB flash disk, the data output and data input soft key functions are the same.
2) For the data output / input operation, please ensure that the I/O channel settings are correct. When using a USB flash disk, set I/O channel to 2; when using the transfer software on a PC, set I/O channel set 0 or 1.
3) The content of one-click operation is determined by the password authority. For the correspondence between each data item and password authority, please refer to 3.4.5 of "Part II Operation instructions".
4) Relevant parameters:
   ① Set by position parameter N0:54#7: Whether one-click input/output is valid for the part program when debugging and above.
   ② Set by position parameter N0:27#0: Whether to prohibit the editing of the program number 80000-89999 subprogram.
   ③ Set by position parameter N0:27#4: Whether to prohibit the editing of the program number 90000-99999 subprogram.
5) In data processing, the system sets related operation prompts which are shown in Table 3-4-4-2.

Table 3-4-4-2

S/N	**Tooltip information**	**Cause**	**Handling**
1	One-click operation completed	Done	Transfer completed
2	When one-click operation is completed, the system prompts: please alter the parameters before copying.	The input / output operation of the macro program is executed, but the system related parameters is not set.	Skip the input/output operations of this file.
3	System alarms when one-click operation is completed: parameters that primary power must be cut off have been altered.	The ladder and ladder diagram parameters are updated and power-on is required again.	Transfer completed, please power on again
4	File reading failed	File error	Input and output operation interruption
5	File writing failed	File error	Input and output operation interruption
6	file copying failed	File error	Input and output operation interruption
7	The file is too large, please use DNC	Part program greater than 4M	Input and output operation interruption
8	Insufficient remaining space	Insufficient space	Input and output operation interruption

6) The LADCHI**.TXT file is invalid after being transferred into the system and requires power-off.

3.4.5 Password Permission Setting and Modification

In order to prevent the processing program and **CNC** parameters from being maliciously modified, the **GSK218MC** system provides the permission setting function. The password level is divided into 5 levels from high to low, including level 1 (system manufacturer level), level 2 (machine manufacturer level), level 3 (system debugging level), level 4 (end user level) and level 5 (processing level). The default minimum level of power-on system (see Fig. 3-4-5-1).

Level 1, Level 2: It is allowed to modify the **CNC** status parameters, data parameters, tool offset data, and transfer PLC ladder diagrams.

Level 3: It is allowed to modify the **CNC** status parameters, data parameters, tool offset data, and pitch compensation.

Level 4: The status parameters and data parameters of some **CNC**s are modifiable.

Level 5: Without password level, the tool compensation data and macro variables are modifiable, the operation of the machine operation panel can be performed, but the **CNC** status parameters, data parameters and pitch compensation data are not modifiable.

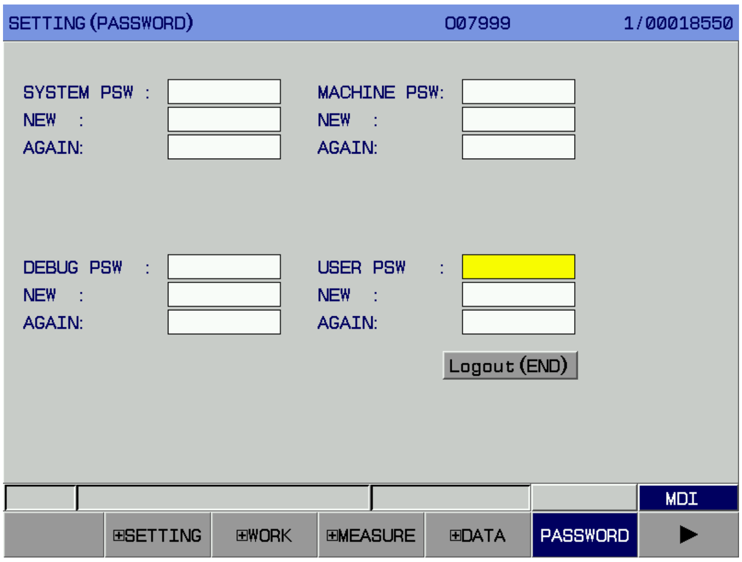


Fig. 3-4-5-1

1) After entering the interface under the <Input mode>, move the cursor to the target position.

2) Enter the password of the corresponding level and press the INPUT key. If it is correct, the system will prompt "Password is correct".

3) Enter 0-6 digits or letters when modifying the system password, and press the INPUT key to confirm.

4) After completing modification, press the END key and move the cursor to the "Logout (END)" button; the interface prompts: "Press <Input> key to confirm logout!" After pressing INPUT key to confirm, the interface prompts: "Logout completed!" At the same time, return the cursor back to the password setting field. The password will also be automatically logged out after power-off and reboot.

3.5 Graphic Display

Press the GRAPH key to enter the graphic page including two display interfaces: **[Graphic parameter]** and **[⊞ Graphic]** which are switched and displayed by the corresponding soft keys: Details are shown in Fig. **3-5-1**.

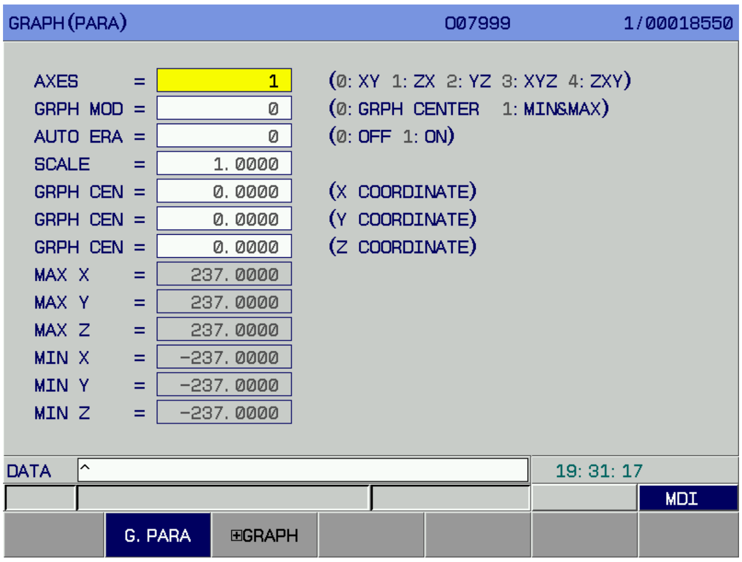


Fig. 3-5-1

1) For the graphic parameter interface, press the [Graphic parameter] soft key to enter the graphic parameter interface, as shown in Fig. 3-5-1.

A. Meaning of the graphic parameters

Coordinate selection: Set the drawing plane including 6 options (0-5), as shown in the second line.

Graphic mode: Set the graphic display mode.

Auto erase: When set to 1, the program graphics are automatically erased when the next cycle is started after the program ends.

Scaling: Set the drawing scale.

Graphic center: Set the workpiece coordinate value corresponding to LCD center in the workpiece coordinate system.

Maximum and minimum: When the maximum and minimum values of the display axis are set, the **CNC** system automatically sets the scaling and the graphic center value automatically.

X maximum: The maximum value of the X direction in the graphic display (unit: 0.0001mm/0.0001inch)

X minimum: The minimum value of X direction in the graphic display (unit: 0.0001mm/0.0001inch)

Y maximum: The maximum value of Y direction in the graphic display (unit: 0.0001mm/0.0001inch)

Y minimum: The minimum value of Y direction in the graphic display (unit: 0.0001mm/0.0001inch)

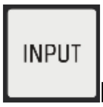
Z maximum: The maximum value of Z direction in the graphic display (unit: 0.0001mm/0.0001inch)

Z minimum: The minimum value of Z direction in the graphic display (unit: 0.0001mm/0.0001inch)

B. Setting method for graphic parameters

a) Move the cursor to the parameter to be set;

b) Type the corresponding value according to the actual requirements;

c) Press the 

key to confirm.

2) For the graphic interface, press **[⊞ Graphics]** to enter the graphical interface (see Fig. 3-5-2).

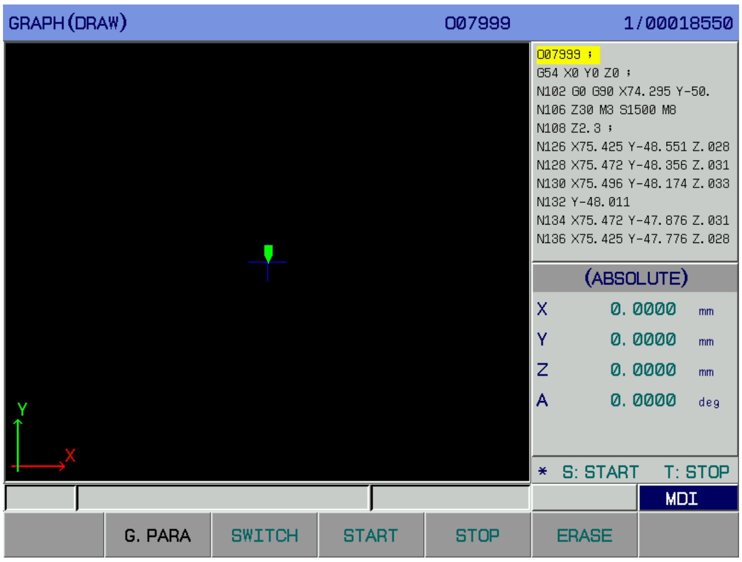
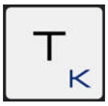


Fig. 3-5-2

In the graphics page, you can monitor the machining path of the running program.

A. Press the [Start] soft key or the 

key to enter the drawing starting status. Then, move '*' number to S: **Before** the drawing starts;

B. Press the [Stop] soft key or the 

key to enter the drawing stopping status. Then, move the '*' number to T: **Before** the drawing stops;

C. Pressing the [Switch] soft key each time, the graph switches between the coordinate displays corresponding to 0 to 5.

D. Press the [Clear] soft key or the 

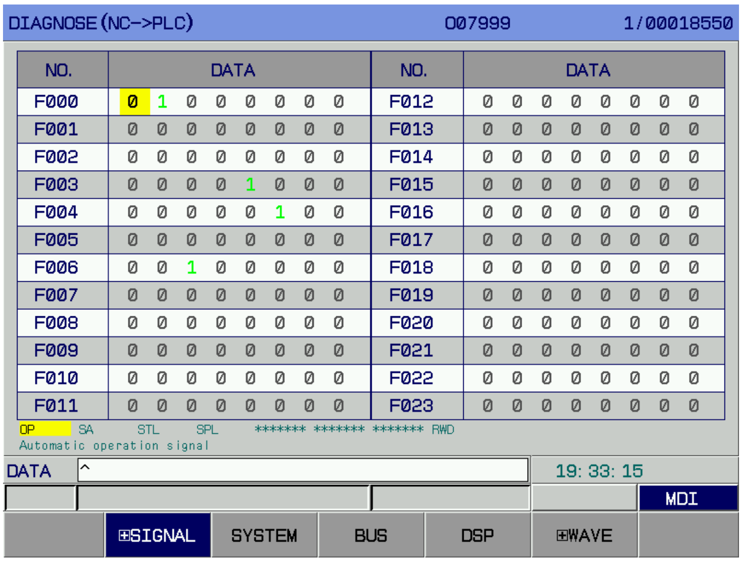
key to clear the drawn graph.

3.6 Diagnosis Display

The status of the DI/DO signal between **CNC** and the machine, the signal status transmitted between **CNC** and the **PLC**, **PLC** internal data, and **CNC** internal status are all displayed on the diagnosis page. For the meaning and setting method of each diagnostic number, please refer to the ***GSK218MC Machining Center CNC System PLC and Installation & Connection Manual***, a supporting manual of this book.

This part of the diagnosis is used to detect the running status of CNC interface signal and the internal signal and is unmodifiable.

Press the  key to enter the diagnosis display page including five interfaces: [⊞ Signal], [System], [Bus], [DSP] and [⊞ Waveform]. Switch and view via corresponding soft keys (see Fig. 3-6-1).



NO.	DATA	NO.	DATA
F000	0 1 0 0 0 0 0 0	F012	0 0 0 0 0 0 0 0
F001	0 0 0 0 0 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 1 0 0	F016	0 0 0 0 0 0 0 0
F005	0 0 0 0 0 0 0 0	F017	0 0 0 0 0 0 0 0
F006	0 0 1 0 0 0 0 0	F018	0 0 0 0 0 0 0 0
F007	0 0 0 0 0 0 0 0	F019	0 0 0 0 0 0 0 0
F008	0 0 0 0 0 0 0 0	F020	0 0 0 0 0 0 0 0
F009	0 0 0 0 0 0 0 0	F021	0 0 0 0 0 0 0 0
F010	0 0 0 0 0 0 0 0	F022	0 0 0 0 0 0 0 0
F011	0 0 0 0 0 0 0 0	F023	0 0 0 0 0 0 0 0

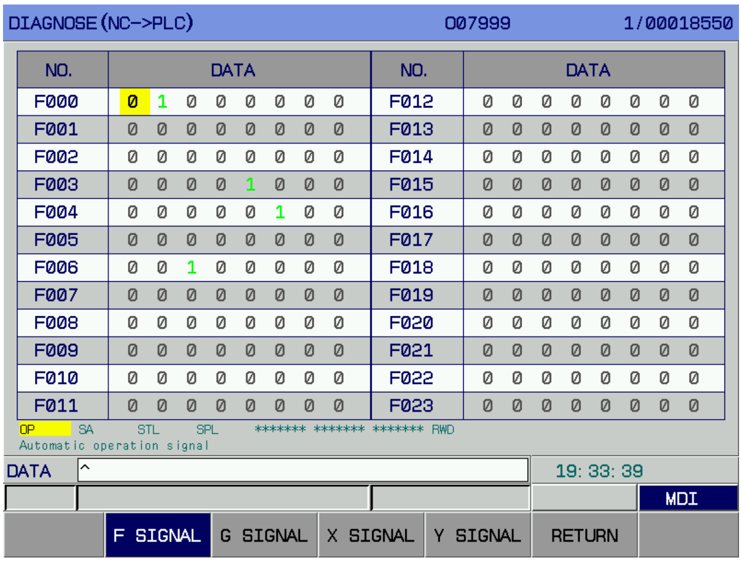
Fig. 3-6-1

3.6.1 Diagnostic Data Display

3.6.1.1 Signal parameter display

Press the [Signal] soft key to enter the signal diagnosis interface. The content displayed on this interface is shown in Fig. 3-6-1-1-1 to Fig. 3-6-1-1-).

1. For F signal interface, press [F Signal] in the <Diagnosis> interface to enter the diagnosis (NC→PLC) interface (as shown in Fig. 3-6-1-1-1).

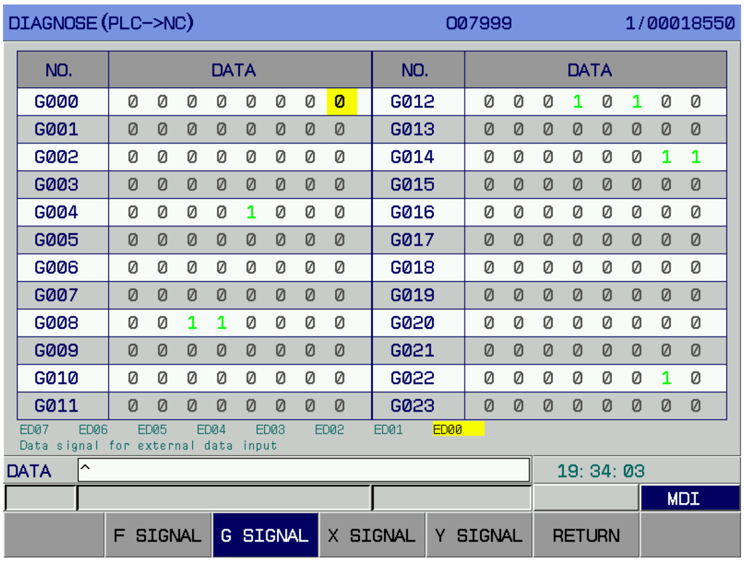


NO.	DATA	NO.	DATA
F000	0 1 0 0 0 0 0 0	F012	0 0 0 0 0 0 0 0
F001	0 0 0 0 0 0 0 0	F013	0 0 0 0 0 0 0 0
F002	0 0 0 0 0 0 0 0	F014	0 0 0 0 0 0 0 0
F003	0 0 0 0 1 0 0 0	F015	0 0 0 0 0 0 0 0
F004	0 0 0 0 0 1 0 0	F016	0 0 0 0 0 0 0 0
F005	0 0 0 0 0 0 0 0	F017	0 0 0 0 0 0 0 0
F006	0 0 1 0 0 0 0 0	F018	0 0 0 0 0 0 0 0
F007	0 0 0 0 0 0 0 0	F019	0 0 0 0 0 0 0 0
F008	0 0 0 0 0 0 0 0	F020	0 0 0 0 0 0 0 0
F009	0 0 0 0 0 0 0 0	F021	0 0 0 0 0 0 0 0
F010	0 0 0 0 0 0 0 0	F022	0 0 0 0 0 0 0 0
F011	0 0 0 0 0 0 0 0	F023	0 0 0 0 0 0 0 0

Fig. 3-6-1-1-1

This is a PLC signal from the system. For the meaning and setting method of each diagnostic number, please refer to the ***GSK218MC*** *Machining Center CNC System* ***PLC*** *and Installation & Connection Manual*, a supporting manual of this book.

2. For G signal interface, press [G Signal] in the <Diagnosis> interface to enter the diagnosis (PLC→NC) interface (as shown in Fig. 3-6-1-1-2).

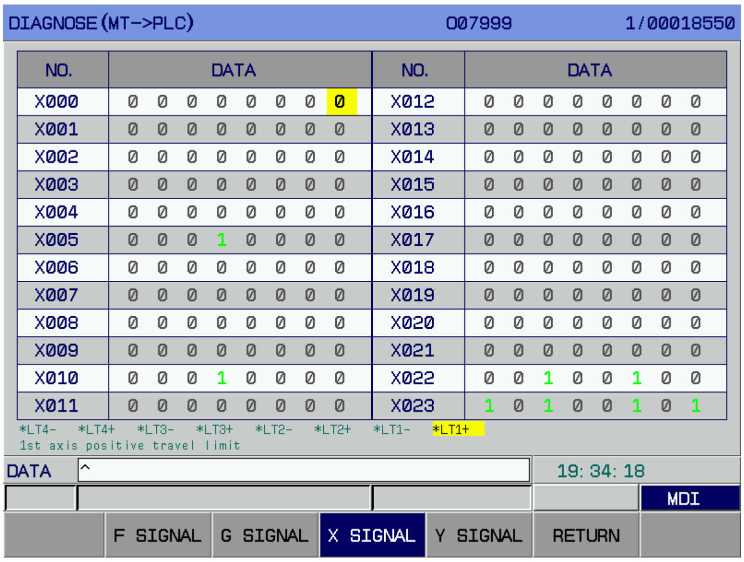


NO.	DATA	NO.	DATA
G000	0 0 0 0 0 0 0 0	G012	0 0 0 1 0 1 0 0
G001	0 0 0 0 0 0 0 0	G013	0 0 0 0 0 0 0 0
G002	0 0 0 0 0 0 0 0	G014	0 0 0 0 0 0 1 1
G003	0 0 0 0 0 0 0 0	G015	0 0 0 0 0 0 0 0
G004	0 0 0 0 1 0 0 0	G016	0 0 0 0 0 0 0 0
G005	0 0 0 0 0 0 0 0	G017	0 0 0 0 0 0 0 0
G006	0 0 0 0 0 0 0 0	G018	0 0 0 0 0 0 0 0
G007	0 0 0 0 0 0 0 0	G019	0 0 0 0 0 0 0 0
G008	0 0 1 1 0 0 0 0	G020	0 0 0 0 0 0 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 0 0 0 0 0 0 0	G023	0 0 0 0 0 0 0 0

Fig. 3-6-1-1-2

This is a system signal from PLC. For the meaning and setting method of each diagnostic number, please refer to the ***GSK218MC*** *Machining Center CNC System* ***PLC*** *and Installation & Connection Manual*, a supporting manual of this book.

3. For X signal interface, press [X Signal] in the <Diagnosis> interface to enter the diagnosis (MT→PLC) interface (as shown in Fig. 3-6-1-1-3).



NO.	DATA	NO.	DATA
X000	0 0 0 0 0 0 0 0	X012	0 0 0 0 0 0 0 0
X001	0 0 0 0 0 0 0 0	X013	0 0 0 0 0 0 0 0
X002	0 0 0 0 0 0 0 0	X014	0 0 0 0 0 0 0 0
X003	0 0 0 0 0 0 0 0	X015	0 0 0 0 0 0 0 0
X004	0 0 0 0 0 0 0 0	X016	0 0 0 0 0 0 0 0
X005	0 0 0 1 0 0 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 0 0 0	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 1 0 0 0 0	X022	0 0 1 0 0 1 0 0
X011	0 0 0 0 0 0 0 0	X023	1 0 1 0 0 1 0 1

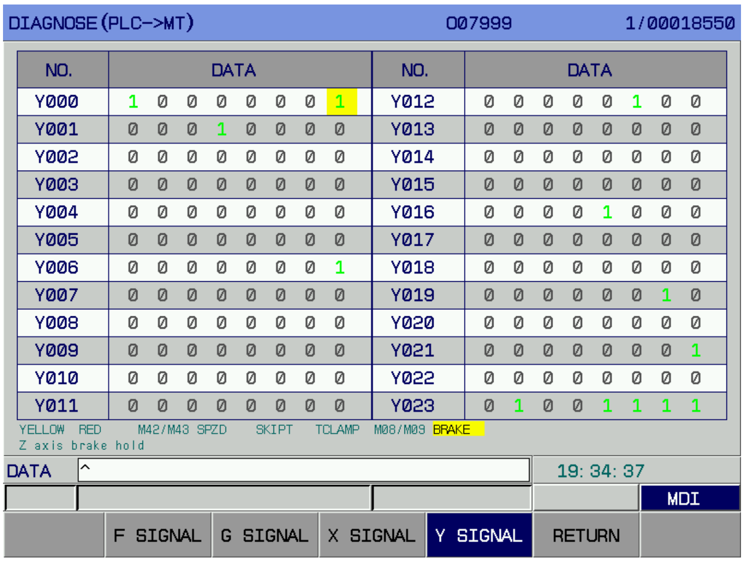
Fig. 3-6-1-1-3

This is a PLC signal from the machine. For the meaning and setting method of each diagnostic number, please refer to the ***GSK218MC*** *Machining Center CNC System* ***PLC*** *and Installation & Connection Manual*, a supporting manual of this book.

4. For Y signal interface, press [Y Signal] in the <Diagnosis> interface to enter (PLC→MT)

Volume II Operation Instructions

interface (as shown in Fig. 3-6-1-1-4).



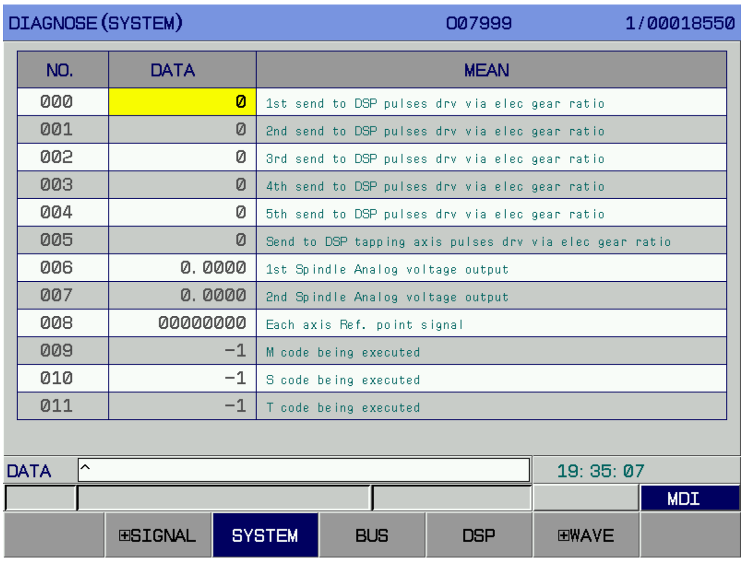
NO.	DATA	NO.	DATA
Y000	1 0 0 0 0 0 0 1	Y012	0 0 0 0 0 1 0 0
Y001	0 0 0 1 0 0 0 0	Y013	0 0 0 0 0 0 0 0
Y002	0 0 0 0 0 0 0 0	Y014	0 0 0 0 0 0 0 0
Y003	0 0 0 0 0 0 0 0	Y015	0 0 0 0 0 0 0 0
Y004	0 0 0 0 0 0 0 0	Y016	0 0 0 0 1 0 0 0
Y005	0 0 0 0 0 0 0 0	Y017	0 0 0 0 0 0 0 0
Y006	0 0 0 0 0 0 0 1	Y018	0 0 0 0 0 0 0 0
Y007	0 0 0 0 0 0 0 0	Y019	0 0 0 0 0 0 1 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 0 0
Y011	0 0 0 0 0 0 0 0	Y023	0 1 0 0 1 1 1 1

Fig. 3-6-1-1-4

This is a machine signal from PLC. For the meaning and setting method of each diagnostic number, please refer to the ***GSK218MC** Machining Center CNC System **PLC** and Installation & Connection Manual*, a supporting manual of this book.

3.6.1.2 System parameter display

Press the [System] soft key to enter the system signal diagnosis interface. The content displayed on this interface is shown below (see Fig. 3-6-1-2-1).

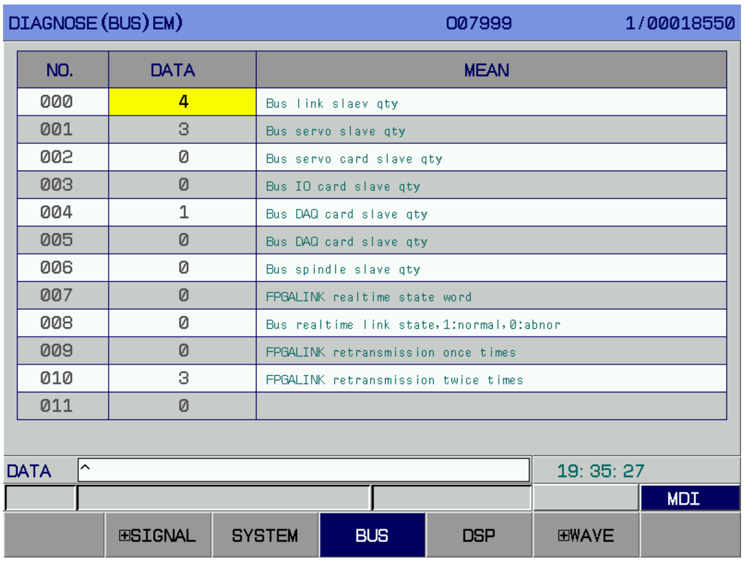


NO.	DATA	MEAN
000	0	1st send to DSP pulses drv via elec gear ratio
001	0	2nd send to DSP pulses drv via elec gear ratio
002	0	3rd send to DSP pulses drv via elec gear ratio
003	0	4th send to DSP pulses drv via elec gear ratio
004	0	5th send to DSP pulses drv via elec gear ratio
005	0	Send to DSP tapping axis pulses drv via elec gear ratio
006	0.0000	1st Spindle Analog voltage output
007	0.0000	2nd Spindle Analog voltage output
008	00000000	Each axis Ref. point signal
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

Press the [Bus] soft key to enter the bus signal diagnosis interface. The content displayed on this interface is shown below (see Fig. 3-6-1-3-1).

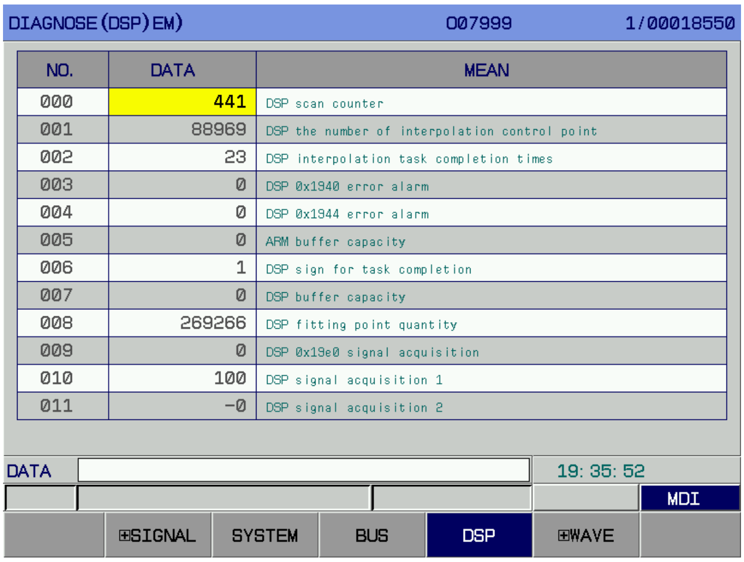


NO.	DATA	MEAN
000	4	Bus link slaev qty
001	3	Bus servo slave qty
002	0	Bus servo card slave qty
003	0	Bus IO card slave qty
004	1	Bus DAQ card slave qty
005	0	Bus DAQ card slave qty
006	0	Bus spindle slave qty
007	0	FPGALINK realtime state word
008	0	Bus realtime link state,1:normal,0:abnor
009	0	FPGALINK retransmission once times
010	3	FPGALINK retransmission twice times
011	0	

Fig. 3-6-1-3-1

3.6.1.4 DSP parameter display

Press the [DSP] soft key to enter the system signal diagnosis interface. The content displayed on this interface is shown below (see Fig. 3-6-1-4-1).



NO.	DATA	MEAN
000	441	DSP scan counter
001	88969	DSP the number of interpolation control point
002	23	DSP interpolation task completion times
003	0	DSP 0x1940 error alarm
004	0	DSP 0x1944 error alarm
005	0	ARM buffer capacity
006	1	DSP sign for task completion
007	0	DSP buffer capacity
008	269266	DSP fitting point quantity
009	0	DSP 0x19e0 signal acquisition
010	100	DSP signal acquisition 1
011	-0	DSP signal acquisition 2

Fig. 3-6-1-4-1

3.6.1.5 Waveform parameter display

Press the [Waveform] soft key to enter the waveform interface, as shown in Fig. 3-6-1-5-1.

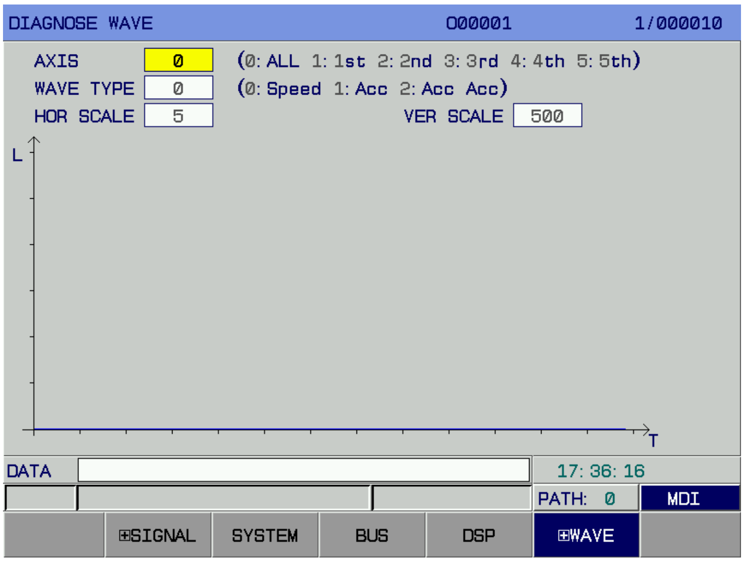
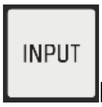


Fig. 3-6-1-5-1

Axis selection: Select the axis for waveform diagnosis.
Waveform selection: Select the contents of waveform diagnosis.
Horizontal axis and vertical axis ratio: Select the proportion of the drawn graph

Data: In any mode, enter the corresponding data and press the 

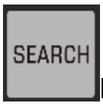
key to confirm.
Use the <Start> key to monitor the signal and the <Stop> key to stop the signal monitoring.

3.6.2 Viewing Signal Status

1) Press the 

key to select the corresponding display page.

2) When moving the cursor left or right, there is a corresponding address interpretation and meaning at the bottom left of the screen.

3) Press 

key to find the target address by moving the cursor or entering the parameter address.

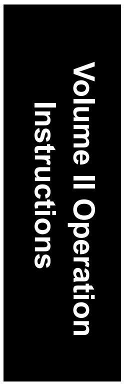
4) [Waveform] interface can display the speed, acceleration and jerk of each feed axis for debugging and find the best adaptation parameters of drive and motor.

3.7 Alarm Display

When the system error alarm occurs, the “alarm” message will be displayed in the lower left corner of the screen. Then, press the 

key to display the alarm page including four display interfaces of [Alarm], [User], [History], and [Record], and to switch and view through the corresponding soft keys (see Fig. 3-7-1 to Fig. 3-7-4). It is also possible to switch to the alarm interface when an alarm occurs via the position parameter **N0:24#6**.

1. For the alarm interface, press the [Alarm] soft key in the <Alarm> interface to enter the alarm



interface, as shown in Fig. 3-7-1.

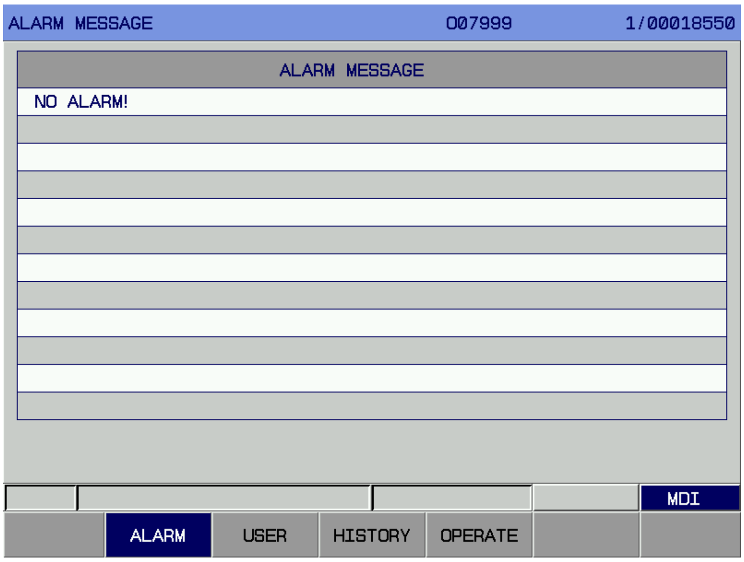


Fig. 3-7-1

The details of the current P/S alarm number are displayed on the alarm display screen. See Appendix II for specific alarm content.

2. For user interface, press the [User] soft key in the <Alarm> interface to enter the external alarm interface, as shown in Fig. 3-7-2.

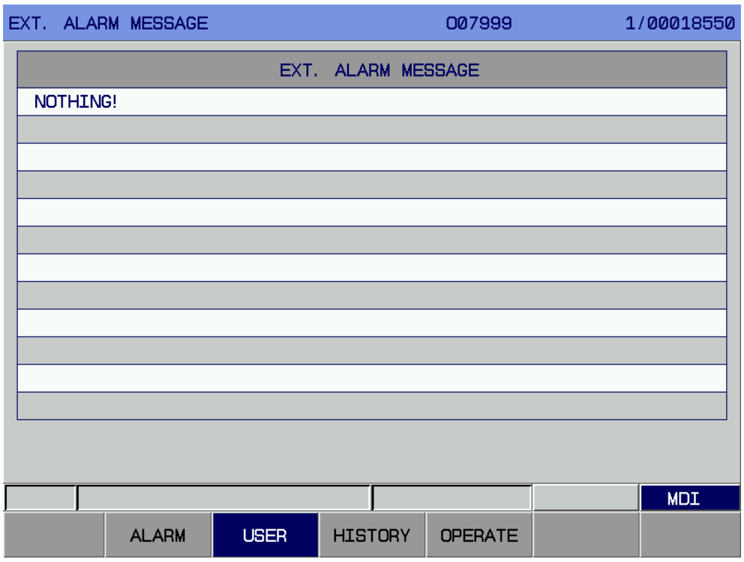


Fig. 3-7-2

For details of each user's alarm information, please refer to the ***GSK218MC** Machining Center CNC System **PLC** and Installation & Connection Manual,* a supporting manual of this book.

Note: Users can set and edit the alarm number for the external alarm according to the actual situation of the site. The edited alarm content is input into the system via the system transmission software. The external alarm is the A of the edited file LadChi**.txt, and the latter two digits are set according to the values of the position parameters 53.0-53.3. (The default value is 01; namely, the file name is LadChi01.txt)

3. For the history interface, press [History] in the <Alarm> interface to enter the historical alarm information interface, as shown in **Fig. 3-7-3**.

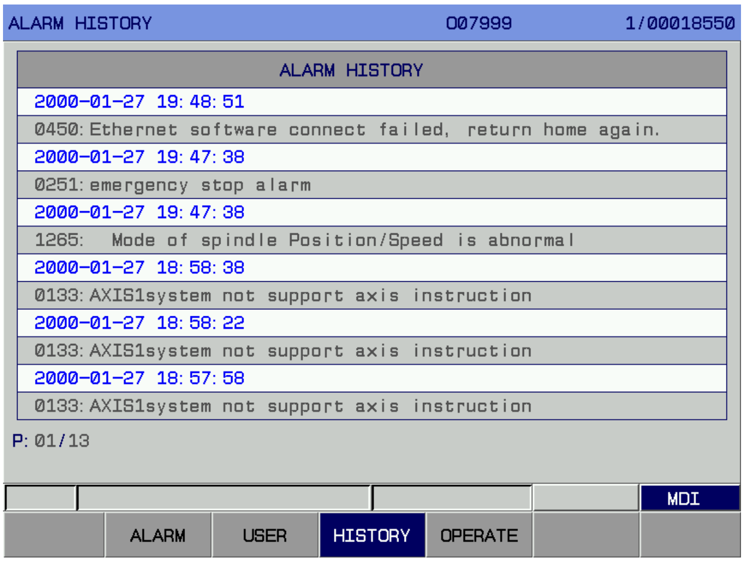


Fig. 3-7-3

In this interface, it is arranged in chronological order from near to far to facilitate the users to view.

4. For the record, interface press [Record] soft key in the <Alarm> interface, as shown in Fig. 3-7-4.

The contents displayed on the operation record interface are the specific modification information of the system parameters and the ladder diagram, such as the modification content and time.

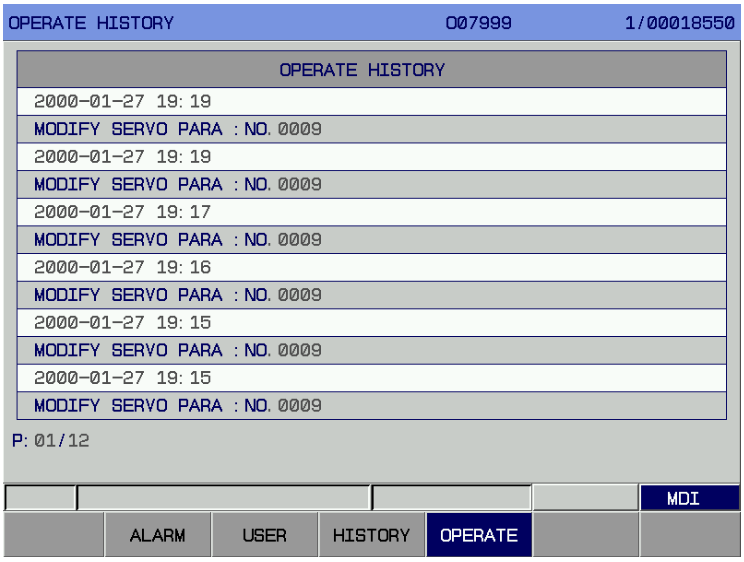
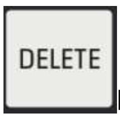


Fig. 3-7-4

The operation record can display 34 pages, and the history alarm information can display 9 pages, such as: View the alarm time, alarm number, alarm information, page number, etc. that appear with the Page-Up and Page-Down keys.

Delete the history and records by pressing the 

key (the password level is at the debugging level or above).

3.8 Program Control Display

Press the  key to enter the program control display page including five display interfaces: [Ladder diagram information], [⊞Ladder diagram], [⊞Ladder diagram parameter], [Signal judgment], and [⊞Signal tracking], which can be switched by corresponding soft keys. The specific display is shown in Fig. 3-8-1 to Fig. 3-8-5.

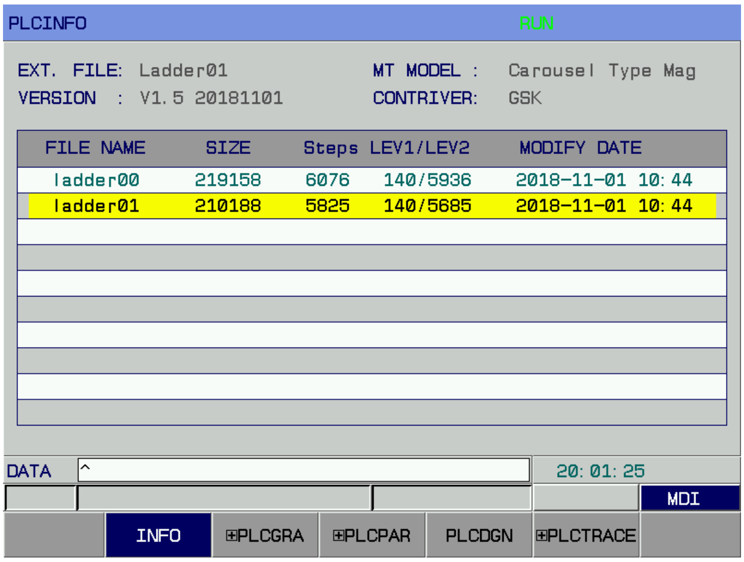


Fig. 3-8-1

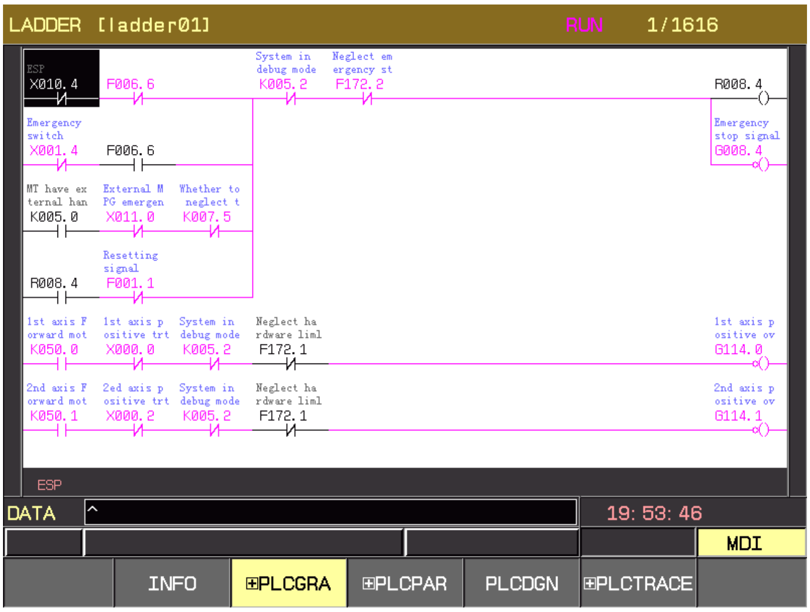
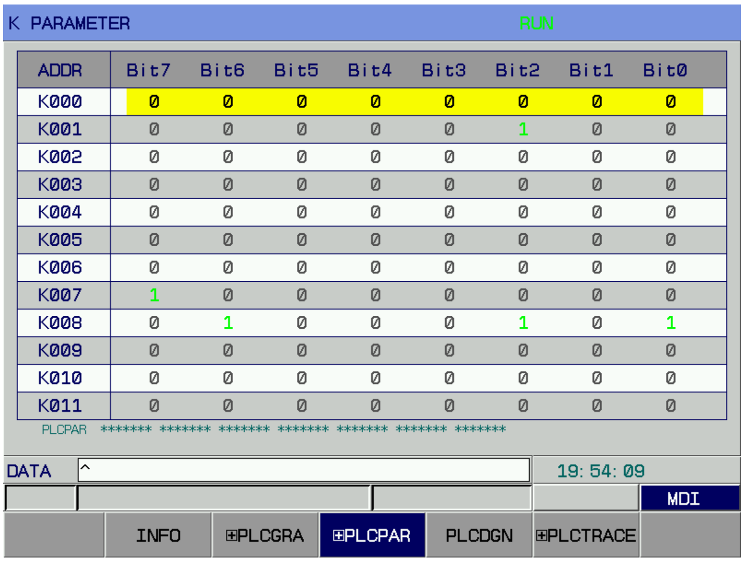
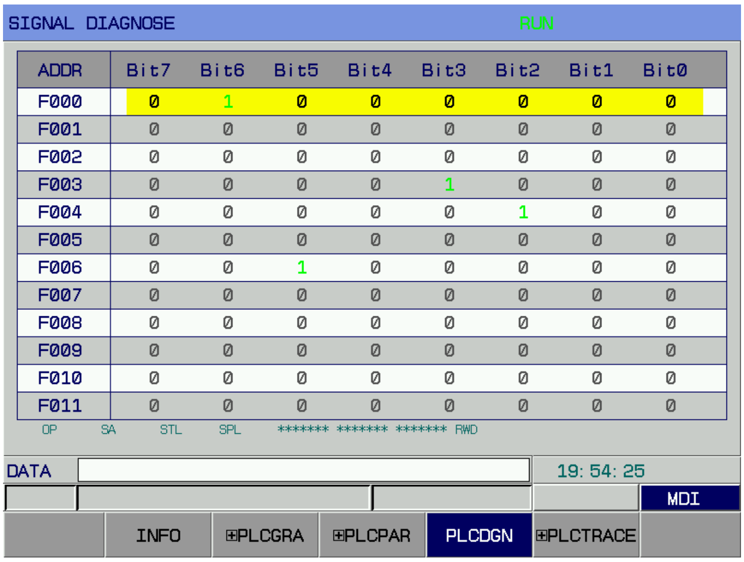


Fig. 3-8-2



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	0
K003	0	0	0	0	0	0	0	0
K004	0	0	0	0	0	0	0	0
K005	0	0	0	0	0	0	0	0
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	0	0	0	0
F002	0	0	0	0	0	0	0	0
F003	0	0	0	0	1	0	0	0
F004	0	0	0	0	0	1	0	0
F005	0	0	0	0	0	0	0	0
F006	0	0	1	0	0	0	0	0
F007	0	0	0	0	0	0	0	0
F008	0	0	0	0	0	0	0	0
F009	0	0	0	0	0	0	0	0
F010	0	0	0	0	0	0	0	0
F011	0	0	0	0	0	0	0	0

Fig. 3-8-4

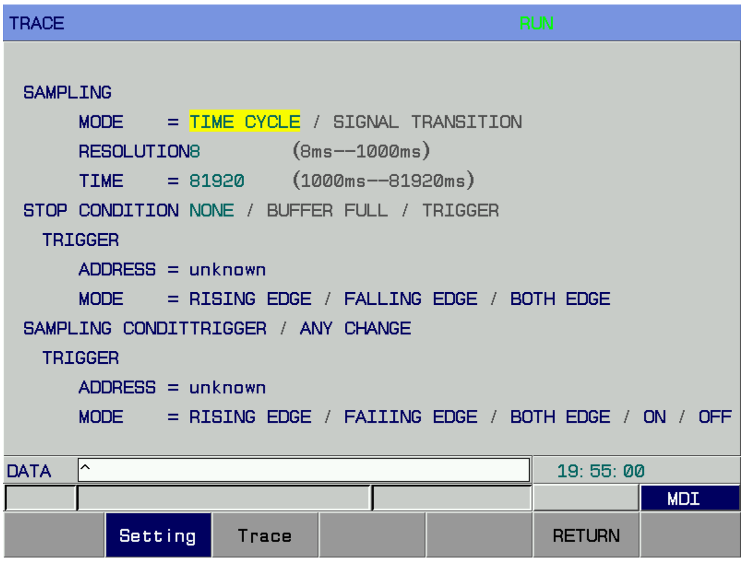


Fig. 3-8-5

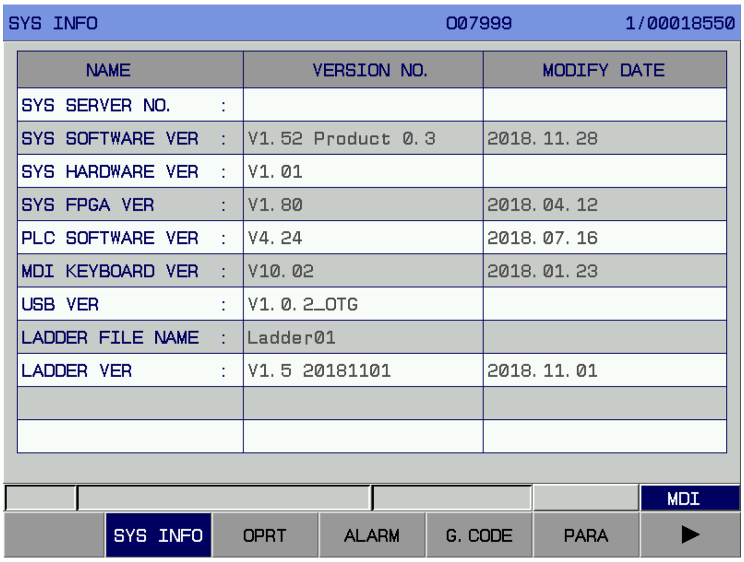
Volume II Operation Instructions

Note: For the modification method and related information about PLC ladder diagram, please refer to *GSK218MC Machining Center CNC System PLC and Installation & Connection Manual.*

3.9 Help Display

Press the  key to enter the help display page including eight display interfaces: [System information] [Operation table] [Alarm table] [G Code table] [Parameter table] [Macro command] [⊞PLC.AD] and [Calculator], which can be viewed through correspondingly soft keys. The specific display is shown in Fig. 3-9-1 to Fig. 3-9-12.

1. For the system information interface, press the [System information] soft key on the <Help> interface to enter the system information interface, as shown in Fig. 3-9-1.



NAME	VERSION NO.	MODIFY DATE
SYS SERVER NO. :		
SYS SOFTWARE VER :	V1.52 Product 0.3	2018.11.28
SYS HARDWARE VER :	V1.01	
SYS FPGA VER :	V1.80	2018.04.12
PLC SOFTWARE VER :	V4.24	2018.07.16
MDI KEYBOARD VER :	V10.02	2018.01.23
USB VER :	V1.0.2_OTG	
LADDER FILE NAME :	Ladder01	
LADDER VER :	V1.5 20181101	2018.11.01

Fig. 3-9-1

2. For the operation table interface, press the [Operation table] soft key on the <Help> interface to enter the help information (operation list) interface, as shown in Fig. 3-9-2.

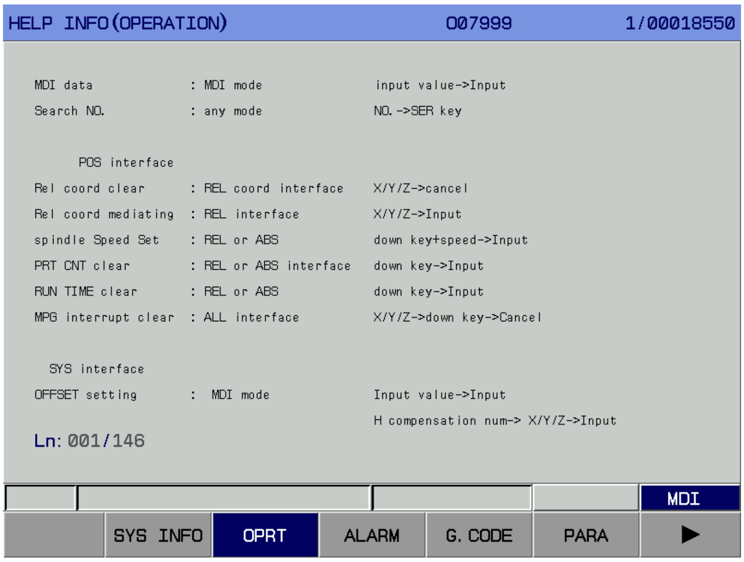


Fig. 3-9-2

In the help information (operation list) interface, the various operation steps and methods under each interface are described in detail. If the operation is unfamiliar or unclear, search and compare in the help interface.

3. For the alarm interface, press the [Alarm table] soft key on the <Help> interface to enter the help information (operation list) interface, as shown in Fig. 3-9-3.

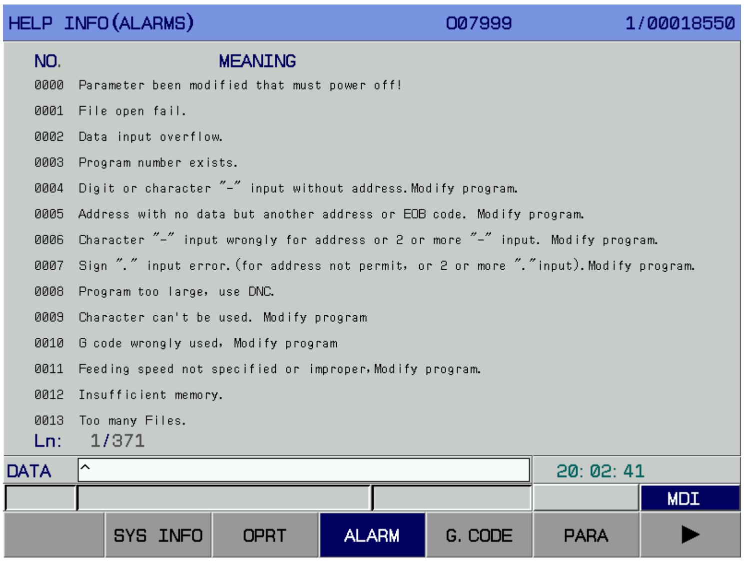


Fig. 3-9-3

The meaning and processing method of each alarm number is described in detail in this interface.

4. For G code table interface, press [G code table] soft key on the <Help> interface to enter the help information (G code list) interface, as shown in Fig. 3-9-4.

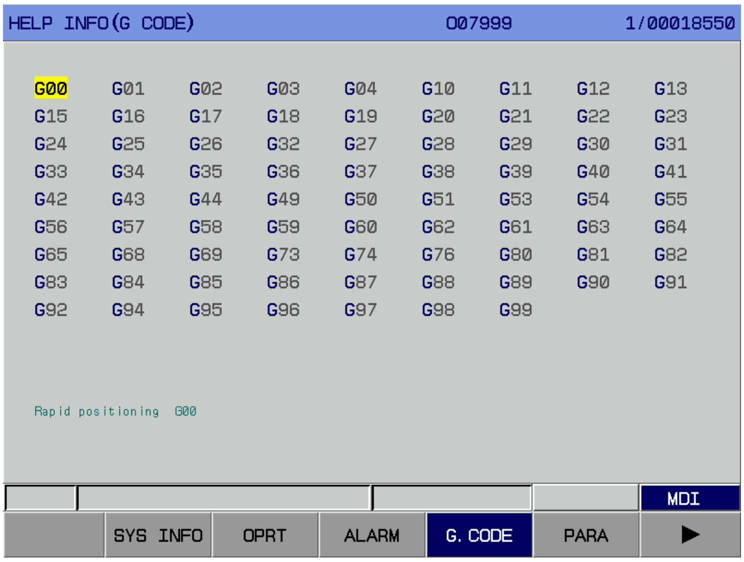
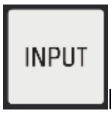


Fig. 3-9-4

In the G code interface, the definition of each G code used by the system is introduced. Use cursor to select and view G code which is defined in the lower left corner of the interface. As shown in the figure 3-9-4. If you want to know the specific format and usage of the G code, select the G code and press the 

key on the panel. Press 

key to return, as shown in Fig. 3-9-5.

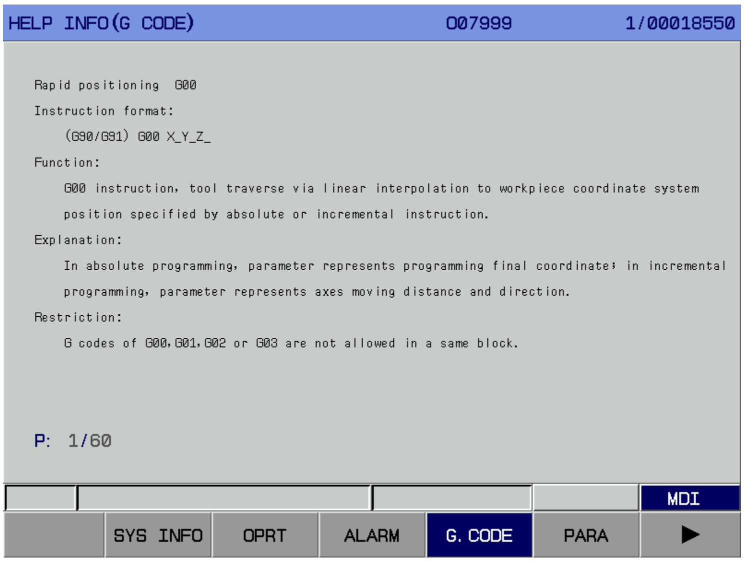


Fig. 3-9-5

In this interface, the format, function, description and restrictions of the code are introduced in detail. If the code is unfamiliar or unclear, search and compare it in this interface.

5. For the parameter table interface, press the [Parameter table] soft key on the <Help> interface to enter the help information (parameter/diagnosis list) interface, as shown in Fig. 3-9-6.

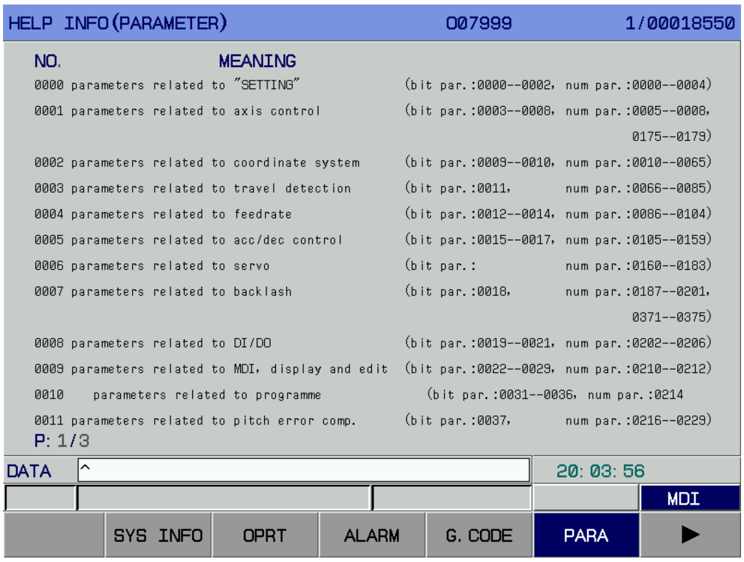


Fig. 3-9-6

In this interface, the parameter settings of each function are introduced in detail. If the parameter settings are unfamiliar or unclear, search and compare it in this interface.

6. For the macro command interface, press the [Macro command] soft key on the <Help> interface to enter the help information (macro list) interface, as shown in Fig. 3-9-7.

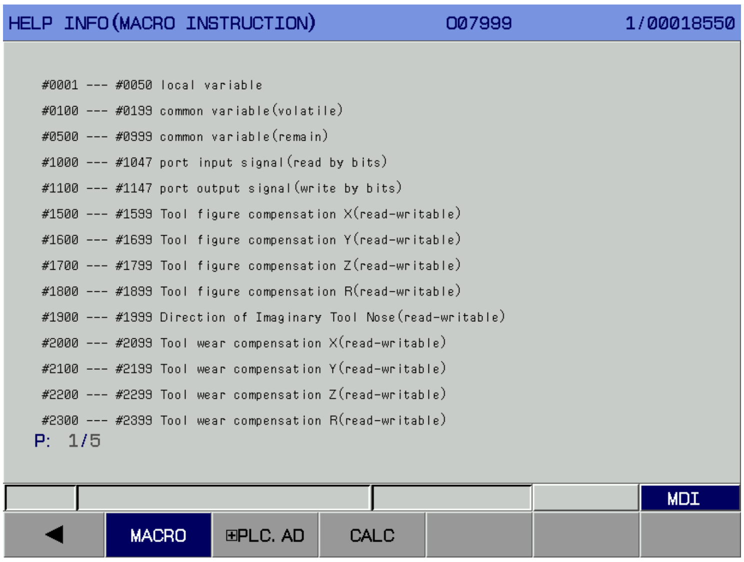


Fig. 3-9-7

In this interface, the format of the macro command and various operation codes are introduced, and the setting range of local variables, general variables and system variables is given. Unfamiliar or unclear macro command operations can be found in this interface.

7. For PLC.AD interface, press the [PLC.AD] soft key on the <Help> interface to enter the help information (PLC address list) interface. The PLC address page has four interfaces: [F address], [G address], [X address], and [Y address]. The specific contents are shown in Fig. 3-9-8 to Fig. 3-9-11.

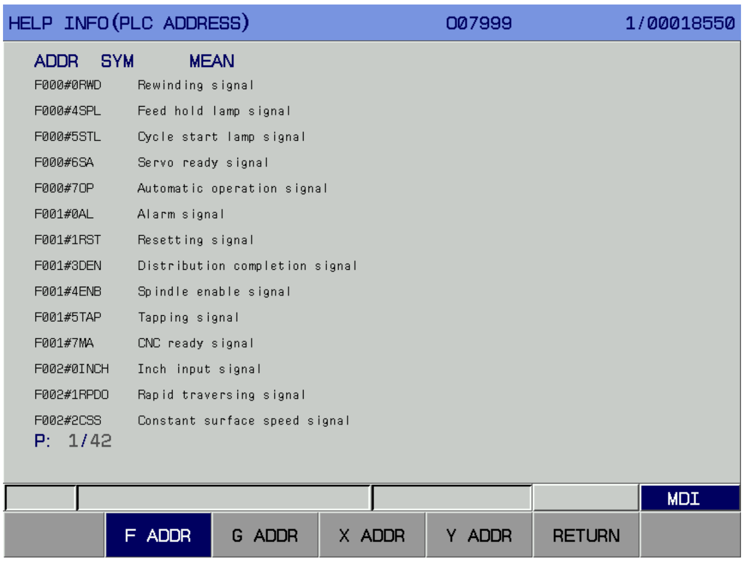


Fig. 3-9-8

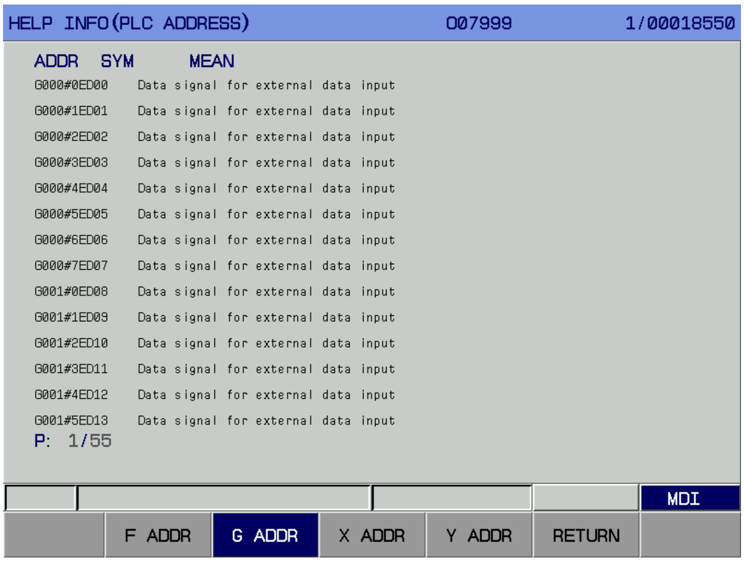


Fig. 3-9-9

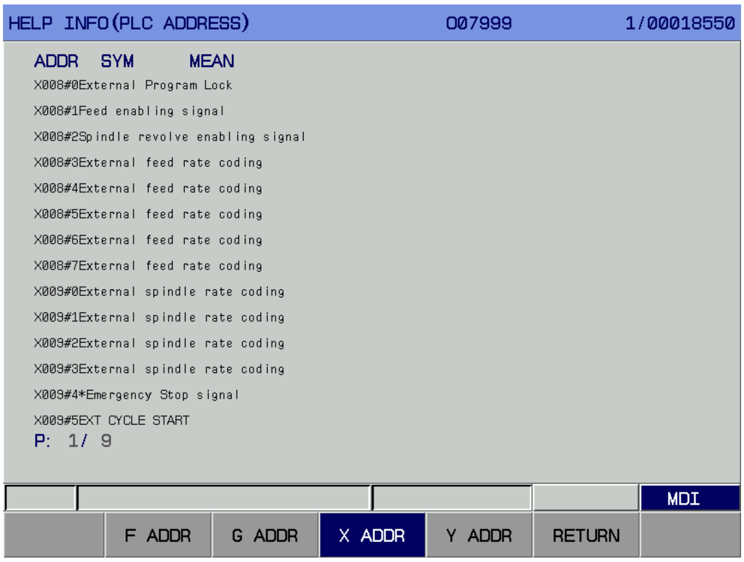


Fig. 3-9-10

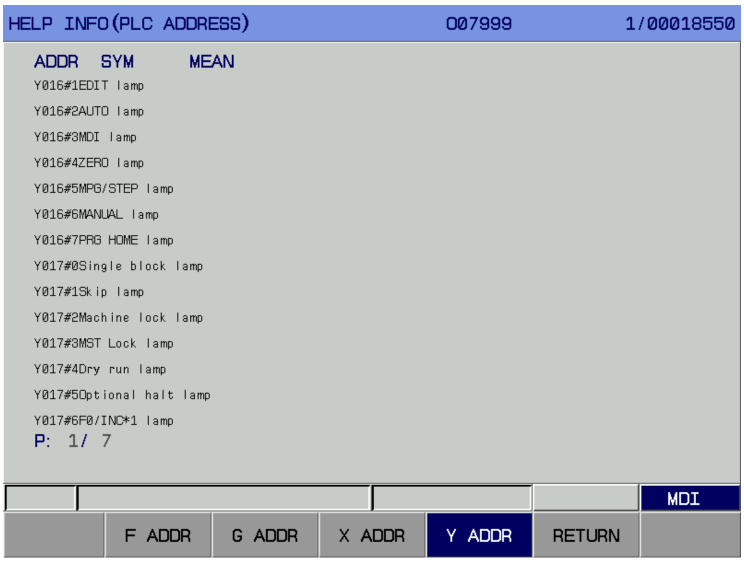


Fig. 3-9-11

The PLC address, symbol and meaning are introduced in detail in the PLC address interface. If the PLC address is unfamiliar or unclear, search and compare it in this interface.

8. For the calculator interface, press the [Calculator] soft key on the <Help> interface to enter the calculator interface, as shown in Fig. 3-9-12.

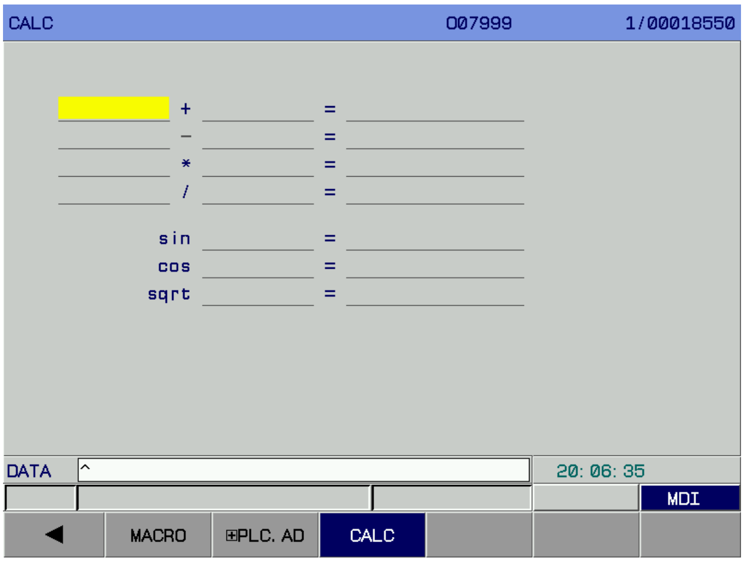
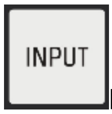


Fig. 3-9-12

In this interface, the system provides the operation modes of addition, subtraction, multiplication, division, sine, cosine and square root. Move the cursor to the space where the data needs to be input, input the data, press the 

key to confirm; after inputting the required data, the system will automatically calculate the result and output it in the space after the =. For the requirement of re-entering the data for calculation, press the 

key, and clear all interface data.

Chapter IV Manual Operation

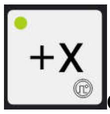
Press 

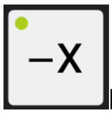
key to enter the manual operation mode, mainly including manual feed, spindle control and machine panel control.

4.1 Coordinate Axis Movement

Under the manual mode, operate each axis at manual feed speed or manual fast moving speed.

4.1.1 Manual Feed

Under the <Manual mode>, press and hold the feed axis 

or 

key to move the corresponding axis at the speed subject change by the adjustment of feed override, and release the key to stop its movement; so does Y axis and Z axis. This system does not support manual multi-axis simultaneous movement, but supports simultaneous zeroing of each axis.

Note: The manual feed speed of each axis is set by P98 parameter.

4.1.2 Manual Fast Movement

Press the 

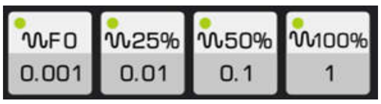
key to light up the indicator light, and then enter the manual fast movement status, then press the feed direction axis key to make each axis run at the fast running speed.

Note 1: The manual fast movement speed is set by P170 to P173.

Note 2: The manual fast movement is set by the position parameter N0:12#0 to be valid before returning to the reference point.

4.1.3 Speed Selection For Manual Feed and Manual Fast Movement

In manual feed, the manual feed override can be selected by the band switch for a total of **21** levels (0%--200%).

In the manual fast movement, 218MC, 218MC-H, 218MC-V can select the manual rapid movement speed override according to 

, of which the rapid override has four levels, including Fo, 25%, 50%, 100%. The Fo speed is set by the data parameter **P93**).

Note: The fast override selection is effective for the following movement speeds.

(1) G00 fast feed (2) Fast feed in the fixed cycle

(3) Fast feed at G28 (4) Manual fast feed

For example: When the fast feed speed is 6 m/min, if the override is 50%, the speed is 3 m/min.

4.1.4 Manual Intervention

Manual intervention can be performed when the program is switched to the manual mode through feed hold when running under the automatic, entry, and DNC modes. Move each axis, and then switch to the mode in which the previous program is running after the action is completed. When pressing

key to run this program, the axes will quickly return to the original manual intervention point in G00 mode and then continue to run the program.

Detailed description:

1. If the single-segment running is carried out during the return running, the tool will perform a single-segment pause at the manual intervention point.
2. If an alarm or reset occurs during manual intervention or return, this function will be canceled.
3. When using manual intervention, use the machine locking, mirror, and zoom functions carefully.
4. When performing manual intervention, pay attention to the machining process and the workpiece shape to avoid damage to the tool or machine.

The manual intervention action is as shown below:

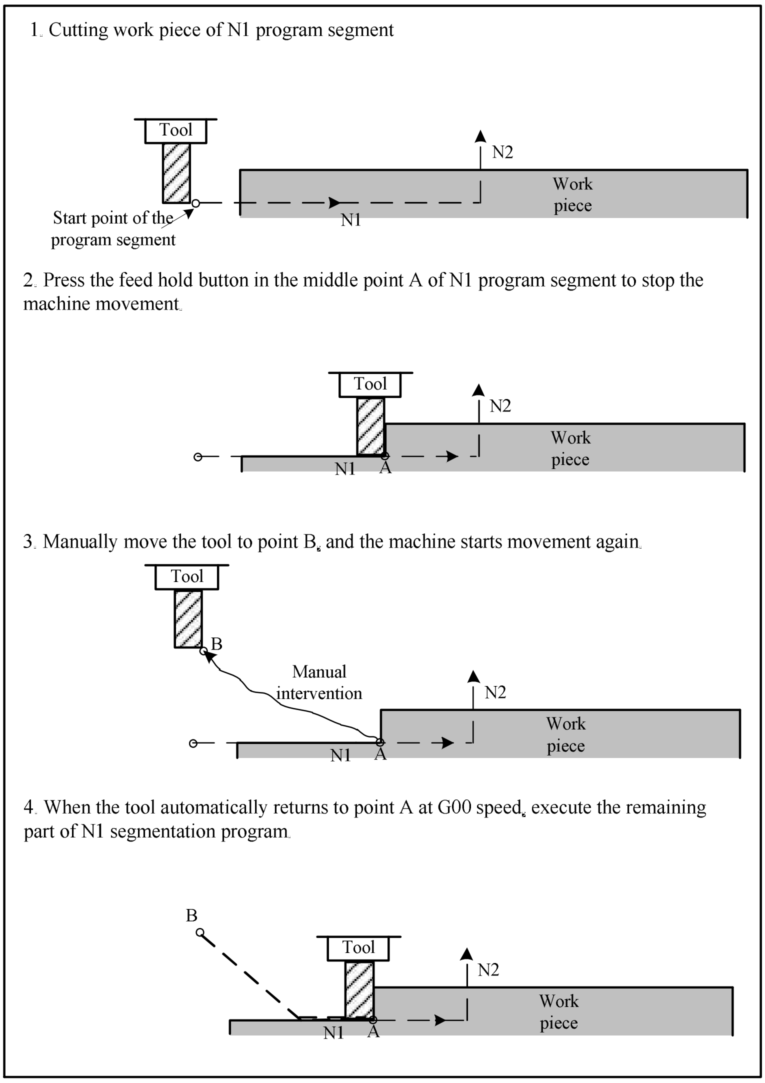


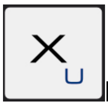
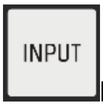
Fig. 4-1-4-1

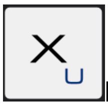
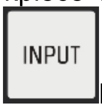
4.1.5 Workpiece Alignment

In order to ensure the machining accuracy (size, shape and positional accuracy) and surface quality of the part, it is necessary to align the workpiece or the fixture of the clamped workpiece.

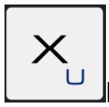
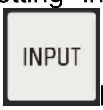
Commonly used alignment methods include: Marking-off alignment method and trial cut alignment method; for the characteristics of the alignment, the **GSK 218MC** system has designed a special operation method for alignment with tool. For example, trial cut and bipartition alignment method (also known as centering alignment method) is used to locate the center of X-Y Plane of a rectangular workpiece by the operation steps as follows:

1） Start the spindle at a certain speed.

2） Switch the system to the display interface of the relative coordinates. Firstly align X direction: Manually operate each motion axis to the positive side of the workpiece X, move the Z axis down to make the tool tip lower than the workpiece surface, and then move to the negative direction of the workpiece at a lower speed (usually using the MPG feed) and stop movement until the tool just cuts into the workpiece. Then, press the  key in the edit panel area, and then press the  key to set the X coordinate to zero. (If the method for setting to other values is similar, such as entering “X20”, press  key to confirm.)

3） Similarly, move the tool cutting workpiece to one side of the workpiece in the negative direction, press the  key after positioning, and then press the  key to complete the centering operation. Note that the centering position will not change the absolute coordinate values and machine coordinate values.

4） Move the tool to the relative coordinates displayed as 0, namely the center of the X direction.

5） Under the “Setting” interface, select the “Workpiece coordinate” page, press the  key and then the  key to complete the setting of the X-axis zero point.

6） At the center of XY (that is, the XY value of the relative coordinates is 0, the position for machine positioning), it is allowed to use G92 to establish a floating coordinate system, or record the XY machine coordinates of this point into the workpiece coordinate system parameters of G54-G59 for system calls.

7） Then, the operation of aligning the center of the rectangular workpiece with the trial cut centering method is completed.

The flexible grasp of the relative coordinate assignment method and the use of the centering function setting will improve the alignment speed and facilitate operation.

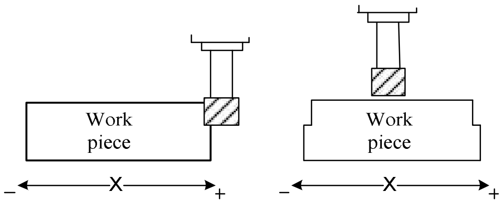


Fig. 4-1-5-1

Note 1: This system can only input the coordinates displayed in relative position. (The position of the relative coordinates can be set where the offset can be altered.)

Note 2: Calculation function can be used to add and subtract the displayed coordinate values and then set to the displayed coordinates.

Note 3: After the coordinate system is set, if the coordinate system set for G92 will be lost due to mechanical zeroing or the workpiece coordinate system G54-G59, and if it will not be lost when it is recorded to the workpiece coordinate system of G54-G59 via the parameter recording machine, the operator should flexibly set it as required, but usually the latter method is recommended.

4.2 Spindle Control

4.2.1 Clockwise Rotation of Spindle

: Under the entry mode, give the S speed, and under the manual/MPG/single-step modes, press this key to make the spindle rotate counterclockwise.

4.2.2 Counterclockwise Rotation of Spindle

: Under the entry mode, give the S speed, and under the manual/MPG/single-step modes, press this key to make the spindle rotate clockwise.

4.2.3 Spindle Stop

: Under the manual/MPG/single-step mode, press this key to stop the spindle.

4.2.4 Automatic Gear Shift of Spindle

Select the spindle selected as the inverter control or I/O point control by the position parameter NO:1#2. In case of NO:1#2=0, the spindle speed is controlled by the S speed command to realize automatic gear shift. At present, the system can perform three-gear control, and the corresponding maximum speed is set by parameters (P246, P247, P248). In case of NO:1#2=1, the spindle speed is automatically changed by I/O point control. At present, the system can perform three-gear control (S1, S2, S3), and the ladder output can be modified to increase gear output. After executing the S speed command for rotation, the system will automatically select the corresponding gear.

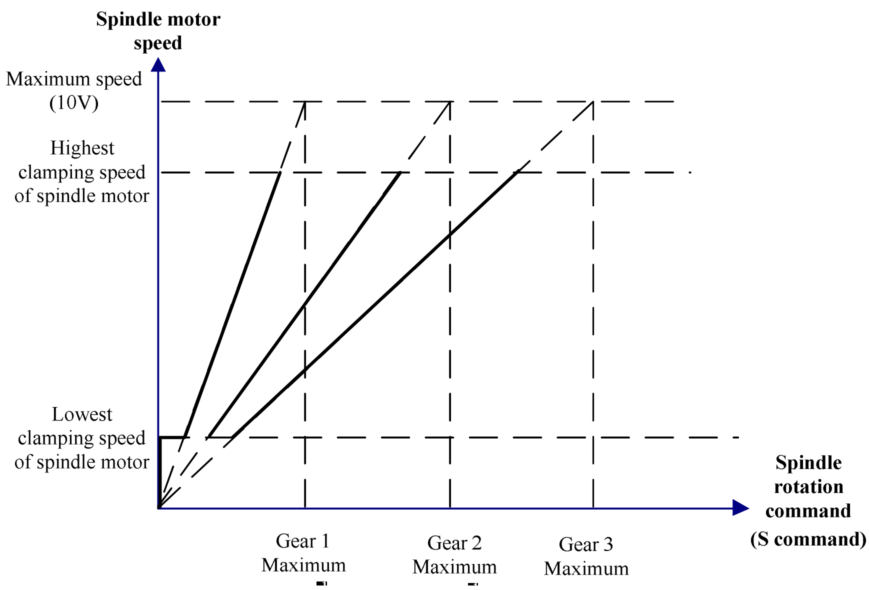


Fig. 4-2-4-1

Note: When the automatic shift control is enabled, detect the spindle gear by the shift in-position signal and execute S code.

4.3 Other Manual Operations

4.3.1 Coolant Control

: The coolant is switched between on and off. The indicator light is on, showing the coolant is on, vice versa.

4.3.2 Lubrication Control

: Press and hold the lubrication key to turn it on, and release the key to turn it off. The indicator light is on, showing the coolant is on, vice versa.

4.3.3 Chip Removal Control

: The chip removal is switched between on and off. The indicator light is on, showing the chip removal is on, vice versa.

4.3.4 Work Light Control

: The work light is switched between connection and disconnection. The indicator light is on, showing the coolant is connected, vice versa.

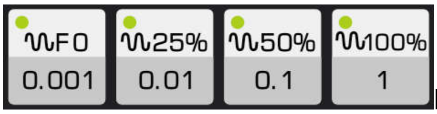
Chapter V Single-step Operation

5.1 Single-Step Feed

Press 

key to enter the single-step mode. In the single-step feed mode, the machine moves each time according to the step defined by the system.

5.1.1 Selection of Movement Amount

Press any one of the 

key to select the movement increment which will be displayed on the page. Press 

key to display the single-step length on the <**Position**> interface: 0.100 (see Fig.5-1-1-1).

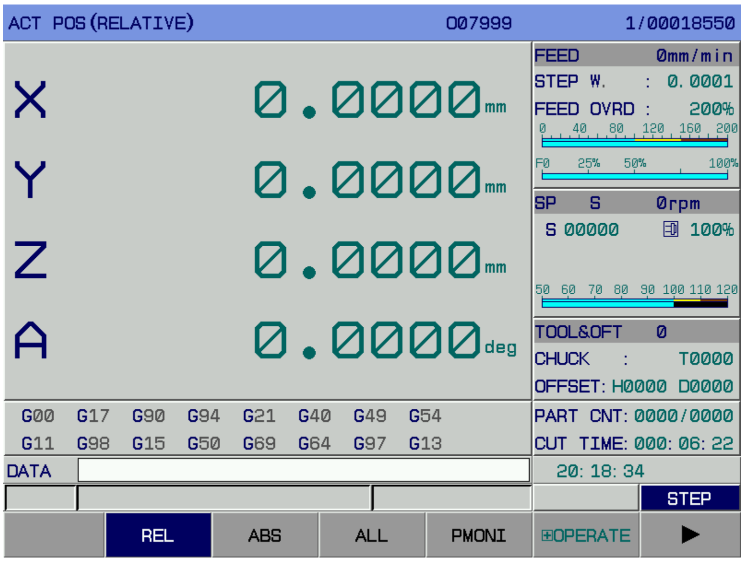
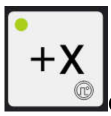
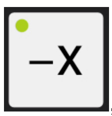


Fig. 5-1-1-1

Each time the movement key is pressed, the corresponding axis of the machine moves by 0.1 mm.

5.1.2 Selection of Movement Axis and Movement Direction

Press the feed axis and direction selection key 

or the 

, press X axis direction key to move the X axis in the positive or negative direction. Each time the key is pressed, the corresponding axis moves the distance defined by the system in a single step; so does the Y axis and the Z axis. This system does not support manual simultaneous three-axis movement, and support simultaneous three-axis zeroing.

5.1.3 Notes On Single-Step Feed

The highest clamp speed for single-step feed is set by the data parameter **P155**.
The single-step feed speed is not controlled by feed speed override and rapid override.

5.2 Single Step Interruption

When there is a program running in the automatic, entry, and DNC mode switched to the single-step mode via pause, the single-step interruption function is executed. The single-step interruption coordinate system is consistent with MPG interruption coordinate system, and the operation function is also in line with the MPG (electronic MPG, refers to the manual pulse generator in the standard
- the MPG, the same below). Please refer to "6.2 Control during MPG interruption operation" in "Part II Operation Instructions".

5.3 Auxiliary Control During Single-Step Operation

As with the manual operation, please refer to 4.2 and 4.3 of the "Part II Operation instructions" for details.

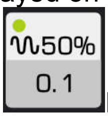
Chapter VI MPG Operation

6.1 MPG Feed

Press the key to enter the MPG mode. In the MPG feed mode, the machine tool is controlled by MPG.

6.1.1 Selection of Movement Amount

Press any one of the key to select the movement increment which will be displayed on the position page.

Press key to display the MPG increment on the <**Position**> interface: 0.100 (see Fig. 6-1-1-1).

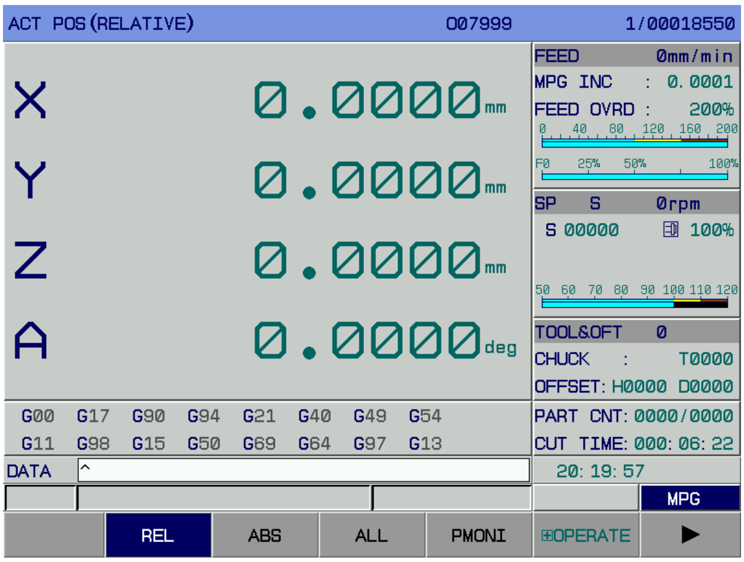
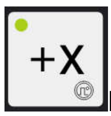


Fig. 6-1-1-1

6.1.2 Selection of Movement Axis and Direction

Under the MPG mode, select the movement axis to be controlled by the MPG, and press the corresponding key to move the axis through the MPG.

Under the MPG mode, if you want to move the X axis with MPG, press the key, then shake MPG to move the X axis.

Rotate MPG to the control the feed direction; see the machine manufacturer's instruction manual for details. In general, the clockwise MPG feed is in the positive direction and the counterclockwise MPG feed is in the negative direction.

6.1.3 Notes On MPG Feed

1. The relationship between the MPG scale and the machine tool movement amount is shown in the following table:

Table 6-1-3-1

	The movement amount on each scale of MPG		
MPG increment (mm)	0.001	0.01	0.1
Machine movement amount (mm)	0.001	0.01	0.1
MPG increment (inch)	0.001	0.01	0.1
Machine movement amount (inch)	0.0001	0.001	0.01

2. The values in the above table vary according to the mechanical transmission. For details, see the machine manufacturer's instruction manual;
3. When the position parameter No.56#0 is set to 1, the MPG movement amount is selected to be fully running. The rotation speed of MPG should not exceed 5r/s. If it exceeds 5r/s, the scale and movement amount may not match with each other.

6.2 Control During MPG Interruption Operation

6.2.1 MPG Interruption Operation

The MPG interruption operation can be superimposed with the automatic movement in the automatic mode.

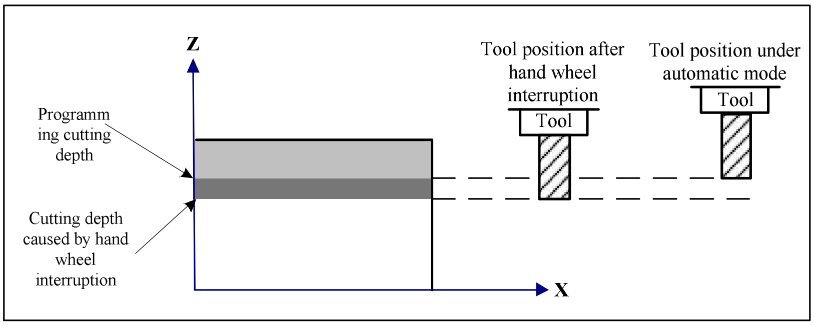


Fig. 6-2-1-1

Operating procedures are shown as follows:

1) Under the automatic mode, after the running program is paused and then switched to the MPG mode, and switch the sub-page <**Comprehensive**> of the <**Position**> interface. Type the "Operation" extension key to select the interruption switch.

2) Use MPG to move the tool position, such as the up and down movement of the Z axis or the translation of the X and Y axes, and the rotation of the A axis, so as to achieve the purpose of modifying the coordinate system.

3) Start after switching to the automatic mode, and keep the workpiece coordinate unchanged until the coordinate recovers its actual value after the mechanical zeroing again.

MPG interruption function can be performed when the program is switched to the MPG mode through feed hold when running under the automatic, entry, and DNC modes. See Fig. 6-2-1-2 for MPG interruption coordinate system.

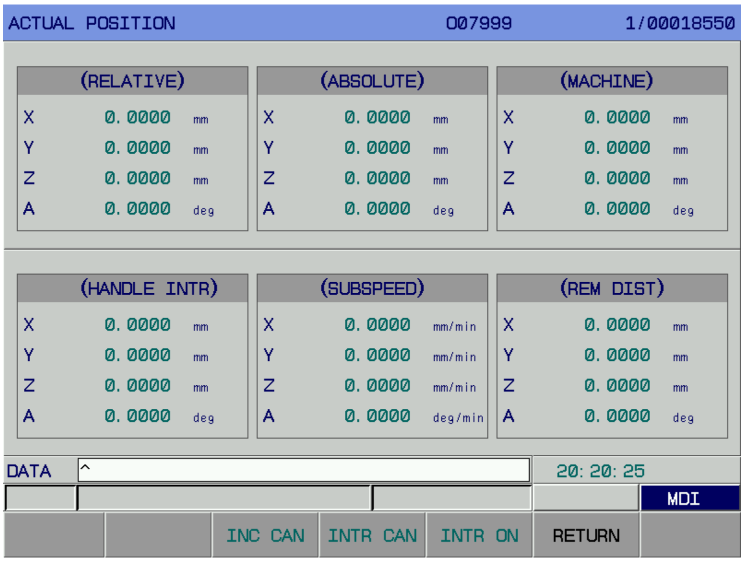


Fig. 6-2-1-2

The zeroing operation of the MPG interruption coordinate system is as follows: Press the X key, move the cursor downward to the MPG interruption coordinate system X to flash, press the 

key to zero the coordinate system, so does the Y and Z axes; when the zeroing operation is performed, the coordinate system will also be automatically zeroed.

Note: The MPG interruption function is canceled when an alarm or reset occurs during the adjustment of the coordinate system by using the MPG interruption function.

6.2.2 Relationship Between MPG Interruption and Other Functions

Table 6-2-2-1

Display	Relationship
Machine locking	After the machine locking is in effect, MPG interruption to move the machine becomes invalid
Absolute coordinate value	MPG interruption does not change the absolute coordinate value
Relative coordinate value	MPG interruption does not change the relative coordinate value
Machine coordinate value	Change amount of the machine coordinate value is the amount of displacement caused by the rotation of MPG

Note: When each axis manually returns to the reference point, the movement amount of the MPG interruption is cleared.

6.3 Auxiliary Control During MPG Operation

As with the manual operation, please refer to **4.2** and **4.3** of the “Part II Operation instructions” of this Manual for details.

6.4 Electronic MPG Drive (Demonstration) Function

The manual rotation of MPG is to control the part program running, and the machine is operated along the tool path commanded by the machining program. This function is mostly used for workpiece rial cut and inspection on machining programs.

Operation method:

In the automatic mode, switch the sub-page <**Program monitoring**> of the <**Position**> interface, type the teaching switch of the “Operation” extension key, press the <Cycle start> key, and then the system axes will not move ; then rotate the MPG to control the part program running; the faster the movement of the MPG, the faster the speed of program execution is, vice versa, as shown in Fig. 6-2-1-2.

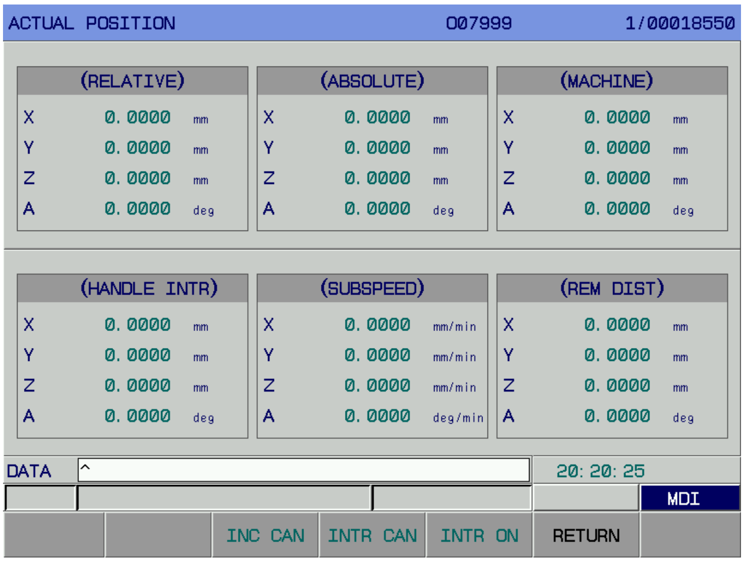


Fig. 6-2-1-2

Note: In single-segment mode, the execution of a single-segment pause is valid.

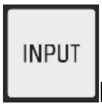
Chapter VII Automatic Operation

7.1 Selection of Automatic Running Program

1. Program loading in automatic mode

(a) Press  **key** to enter the automatic operation mode;

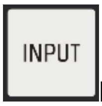
(b) Press  **key** to enter the [Directory] page display and move the cursor to find the target program;

(c) Press  key to confirm.

2. Program loading in edit mode

(a) Press  **key** to enter the edit operation mode;

(b) Press  **key** to enter the [Directory] page display and move the cursor to find the target program;

(c) Press  key to confirm.

(d) Press  **key** to enter the automatic operation mode.

7.2 Automatic Running Start-Up

After selecting the program to start by two methods in 7.1, press  key to start the autorun program, which can be switched to <Position>, <Program monitoring>, <Graphic> and other interfaces to observe the running situation of the program.

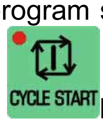
The program starts running at the line where the cursor is located; so it is better to check whether the cursor is on the program line for running and whether modal values are correct before pressing  key. To start from the starting line when the cursor is not in the row, press the reset  key, and then press the  key to run the program automatically from the starting line.

Note: Do not alter the workpiece coordinate system and base offsets in the process of running the program in automatic mode.

7.3 Automatic Running Stop

In the automatic running of the program, the system provides five methods to stop the automatic operation of the program:

1.Program stop (M00)

After the execution of the program segment with M00, the program is paused and the modal information is saved. Press 

key to continue running the program.

2. Program selection-stop (M01)

Before the program running, if pressing 

key, the program will pause after the execution of the program segment with M01, and the modal information is saved. Press 

key to continue running the program.

3. Press 

key

Press 

key during automatic running to make the machine in the following status:

1) Machine feed decelerates and stops;
2) When executing pause (G04 code), stop timing and enter the feed hold status;
3) The remaining modal information is saved;
4) Press 

key to continue running the program.

4. Press 

key

Please refer to section 2.3.1 of “Part II Operation instructions”.

5. Press the emergency stop button.

Please refer to section 2.3.2 of “Part II Operation instructions”.

In addition, when running the program on the MDI interface in the automatic mode, DNC mode and entry mode, switch to other modes to stop the machine. Details:

1) Switch to edit, entry and DNC interfaces to stop the machine after running the current program segment.
2) switch to manual, MPG and single-step interface to immediately interrupt and stop the machine.
3) Switch to the mechanical zeroing interface to decelerate and stop the machine.

7.4 Automatic Running From Any Segment

The system supports automatic running from any segment of the current program. Detailed operation steps are as below:

1. Press the 

key to enter manual mode to start the spindle and other auxiliary functions;
2. When running the program in MDI mode, ensure the modal value is correct;
3. Press 

key to enter the edit operation mode, and press 

key to enter the program page display to find the program to be processed in the [Directory].
4. Open the program, and move the cursor to the program segment to be run;
5. Press 

key to enter automatic operation mode;
6. Press 

key to run the program automatically.

Note 1: before a program runs, confirm that the current coordinate point is the end position of the previous program segment of the running program segment (if the running program segment is absolute programming and is in G00/G01 movement, it is unnecessary to confirm the current coordinate point).

Note 2: if the running program segment executes the tool change or other actions, firstly confirm that the current position will not interfere with the workpiece to avoid machine tool damage and personal accident.

7.5 Dry Running

Before the program processing, use “dry running” to check the program, generally in conjunction with the “auxiliary lock”, “machine lock”.

Press 

key to enter the automatic operation mode and press 

key (the indicator light on the key is on, indicating that it has entered the dry running status).

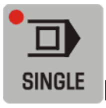
In fast feed, the program speed is the dry running speed × fast feed override.

In cutting feed, the program speed is the dry running speed × the cutting feed override.

Note 1: dry running speed is set by data parameter P86;
Note 2: during tapping, the validity of dry running is set by the position parameter NO:12#5;
Note 3: the validity of dry running in cutting feed is set by the position parameter NO:12#6;
Note 4: the validity of dry running in fast positioning is set by the position parameter NO:12#7;

7.6 Single-Segment Running

If it is required to test the single-segment running of the program, select “single-segment program” running.

Under the automatic, DNC or MDI mode, press 

key (the indicator light on the key is on, indicating that it has entered the single-segment running status).

During single-segment running, the system will stop after executing a program segment; press 

key to continue running the next segment and repeat it until the program running is completed.

Note: In G28, at the intermediate point, a single-segment stop is also performed.

7.7 Machine Locking Running

Under the <**Automatic**> mode, press 

key (the indicator light on the key is on, indicating that it has entered the machine lock running status). Then, the axes of the machine do not move, but the display of the position coordinates is the same as that of the machine movement, and M, S, and T can be executed. This function is used for program verification.

7.8 Auxiliary Function Locking Running

Under the <**Automatic**> operating mode, press 

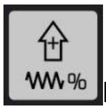
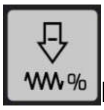
key (the indicator on the button lights up, indicating that the auxiliary function has entered the auxiliary function locking running status). Then, the M, S, and T codes are not executed, and are used together with the machine locking function for program verification.

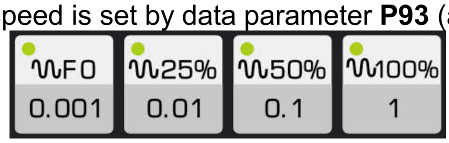
Note: M00, M01, M02, M30, M98, M99 are performed as usual.

7.9 Feed and Fast Speed Adjustment Under Automatic Running

During <**Automatic**> running, the system can change the movement speed during running by adjusting the feed and fast movement override.

In automatic running, press 

key to select the feed speed and to reach 21-level real-time adjustment of the feed override. Press the  key once to increase the feed override by one level; each level is 10%, and it will not increase when it reaches 200%; press the  key once to decrease the feed override by one level; each level is 10%, if the override is set to FO, position parameter NO: 12#4 is used to determine whether the axis stops. If it is set to 0, it will not stop; the actual fast movement speed is set by data parameter **P93** (applicable for all axes).

In automatic running, press 

key to select the fast movement speed, and to realize four-gear adjustment of the fast override, namely Fo, 25%, 50% and 100%.

GSK218MC-H and GSK218MC-V CNC systems select the feed speed through the feed override band switch 

, and reach 21-level real-time adjustment of the feed override.

Note 1: the value set by F in the feed override adjustment program

Actual feed speed = Value by F × Feed override

Note 2: the fast movement speed values obtained from the data parameters P88, P89, P90 and the final adjustment of the fast override are calculated as follows:

X axis actual fast movement speed = Value set by P88 X Fast override

The calculation method for actual fast movement speed calculation method of the Y axis and the Z axis is the same as above.

7.10 Spindle Speed Adjustment In Automatic Running

In automatic running, when the analog is selected to control the spindle speed, the spindle speed is adjustable.

During automatic running, press 

key to adjust the spindle override so as to change the spindle speed, and reach **8**-level real-time adjustment of the spindle override from **50% to 120%.**

Press the 

key once to increase the speed override by one level; each level is 10%, and it will not increase when it reaches 120%;

Press the 

key once to decrease the speed override by one level; each level is 10%, and the spindle speed will stop when it reaches 50%.

GSK218MC-H and GSK218MC-V CNC systems change the spindle speed by adjusting the spindle override through the spindle override band switch 

. The spindle override can reach **8**-level real-time adjustment from **50% to 120%.**

Actual spindle speed = Program instruction speed × Spindle override. The highest spindle speed is set by data parameter **P258**. In case of exceeding this value, it will rotate at this speed.

7.11 Background Edit In Automatic Running

The system supports background edit function during processing.

In the automatic mode, when the program is running, press <Program> key to enter the program display interface, and then press the [Program] soft key to enter the background edit interface, as shown in Fig. 7-11-1.

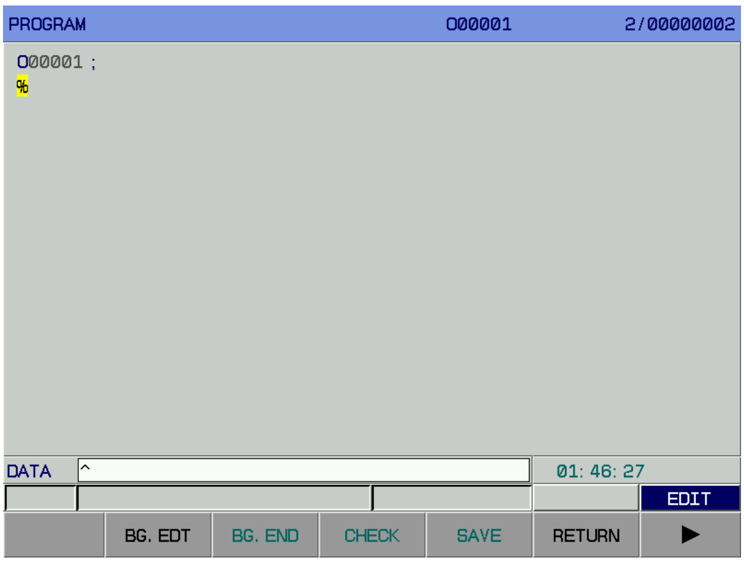


Fig. 7-11-1

Press [B. edit] soft key to enter the program background edit interface, and the program editing is the same as that in the edit mode. For details, please refer to “Chapter X Edit operation” in “Part II Operating instructions” of this Manual, and press [B. end] to save the edited program and exit the interface.

Note 1: the size of the background edit file is recommended not to exceed 3000 lines, or otherwise it will affect the processing effect.

Note 2: background edit can open the foreground program, but cannot edit or delete it.

Note 3: background edit cannot edit the running foreground program.

Chapter VIII MDI Entry Operation

In addition to entering and altering the parameters and offset under the entry mode, the system also provides **MDI** running function, which allows you to enter the code for running directly. Entering data and altering parameters and offsets are described in detail in Chapter 3 *Interface display and data alteration and setting*. This chapter introduces **MDI** running functions in the entry operation.

8.1 MDI Code Segment Input

There are two types of input under **MDI** mode:

1. [MDI] can input multiple sections of programs continuously;

2. [Present/module] can only enter a program.

The input under [MDI] mode is the same as the program input under the edit mode. For details, see Chapter 10, *Program edit operation*. The input under [Present/module] mode is introduced as follows.

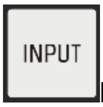
Example: Enter a program segment G00 X50 Y100 from the [Present/module] operation page, subjec to the following operation steps:

1) Press the 

key to enter the entry operation mode;

2) Press the 

key to enter the program interface, press [Present/module] soft key to enter the [Present/module] operation page (see Fig. **8-1-1**):

3) After entering the program segment G00X50Y100 on the keyboard, press the 

key to confirm, and then you can see that the program has been input into the input interface, as shown in Fig. 8-1-1.

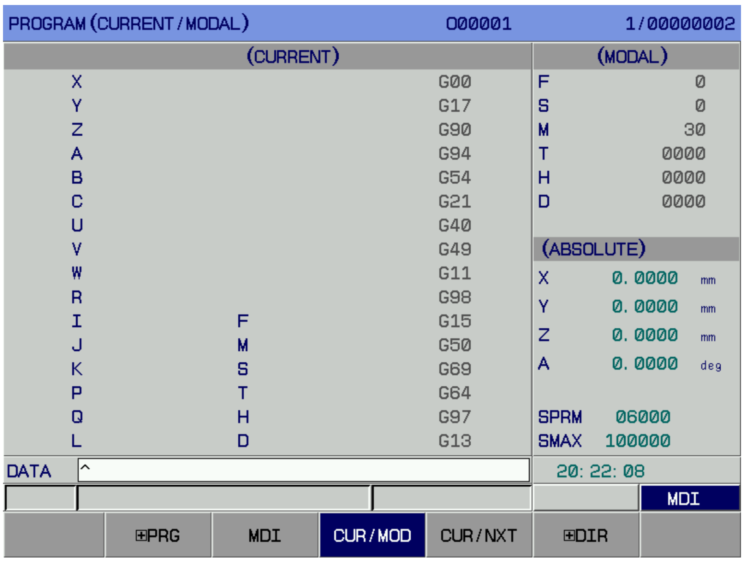


Fig. 8-1-1

8.2 MDI Code Segment Running and Stopping

After entering the code segment according to the steps in **Section 8.1**, press the CYCLE START key to perform **MDI** operation. During the running process, press the FEED HOLD key to stop the code segment running.

Note 1: MDI operation must be carried out under the entry operation mode!

Note 2: under the entry mode, the program input under the present/module interface is prioritized when program is running under the MDI and the present/module interface.

8.3 Alteration and Clear of MDI Code Segment Field Value

If there is an error in the field input, press CANCEL key to cancel the input; if finding the error after input, re-enter the correct content to replace the error one or press the RESET key to clear all input contents and re-enter.

8.4 Conversion of Various Operation Modes

When in the automatic, entry and DNC modes, there are running programs switched to the entry, DNC, automatic, edit modes, the system will stop running the program after current program segment running.

When there is a program running in the automatic, entry, and DNC mode switched to the single-step mode via pause, the single-step interruption function is executed. Refer to “5.2 Single-step interruption” in “Part II Operation instructions”. The MPG interruption function is executed by switching to the MPG mode after pause,; refer to “6.2 MPG Interruption” in Part II Operation instructions”. The manual intervention function is executed by switching to the manual mode after pause; refer to “4.1.4 Manual intervention” in “Part II Operation instructions”.

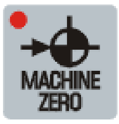
When in the automatic, entry and DNC modes, there are running programs directly switched to the single-step, MPG manual and zeroing modes, the program will decelerate and stop running.

Chapter IX Zeroing operation

9.1 Machine Zero (Mechanical Zero) Concept

The machine tool coordinate system is the inherent coordinate system of the machine tool. The origin of the machine tool coordinate system is called the mechanical zero (or machine tool zero). It is also called the reference point in this manual. It is the mechanical origin specified by the machine tool builder and is usually installed at the maximum travel in the positive direction of the X-axis, Y-axis, Z-axis, the **4th** axis, and the **5th** axis. When the numerical control device is powered on, the mechanical zero is not known, and automatic or manual zeroing is required generally.

9.2 Operation Steps of Pulse Type Servo Machine Zeroing

1. Press the 

key to enter the mechanical zeroing operation mode. At this time, the words "mechanical zeroing" are displayed in the lower right corner of the LCD screen.

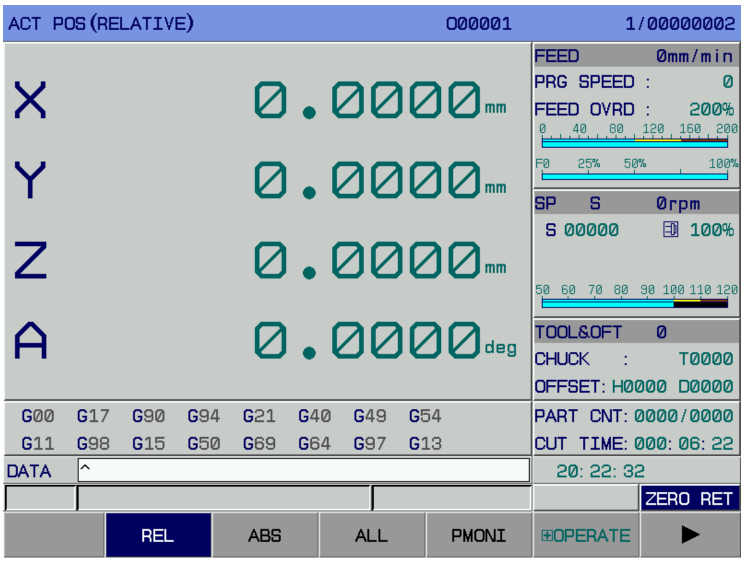


Fig. 9-2-1

2. Select the X-axis, Y-axis, Z-axis, 4TH-axis or 5TH-axis to return to the mechanical zero. The zeroing direction is determined by the position parameter **N0:7#0-N0:7#4**.

3. The machine tool moves along the mechanical zero. Before reaching the deceleration point, it moves quickly (the moving speed is set by the data parameters **P100-P104**). After it touches the deceleration switch, the zeroing speed of each axis is set by the data parameters **P342-P346**. After breaking away from the block, it moves to the mechanical zero (ie, the reference point) at the speed of FL (set by data parameter **P099**). When it returns to the mechanical zero, the coordinate axis stops moving and the zeroing indicator light is on.

Example:

Take the normal increment zeroing of the 1st axis as an example. The 1st axis starts to hit the block with a higher speed F4000 (data parameter **P100** is set to 4000), and passes through the block at F200 (data parameter **P342** is set to 200) after touching the deceleration switch. After it breaks away from the block, the one-turn Z pulse signal of the servo will be searched at a slow speed F40 (data parameter **P99** is set to 40). Such search will be stopped immediately once the signal is gained, as shown in Fig. 9-2-2.

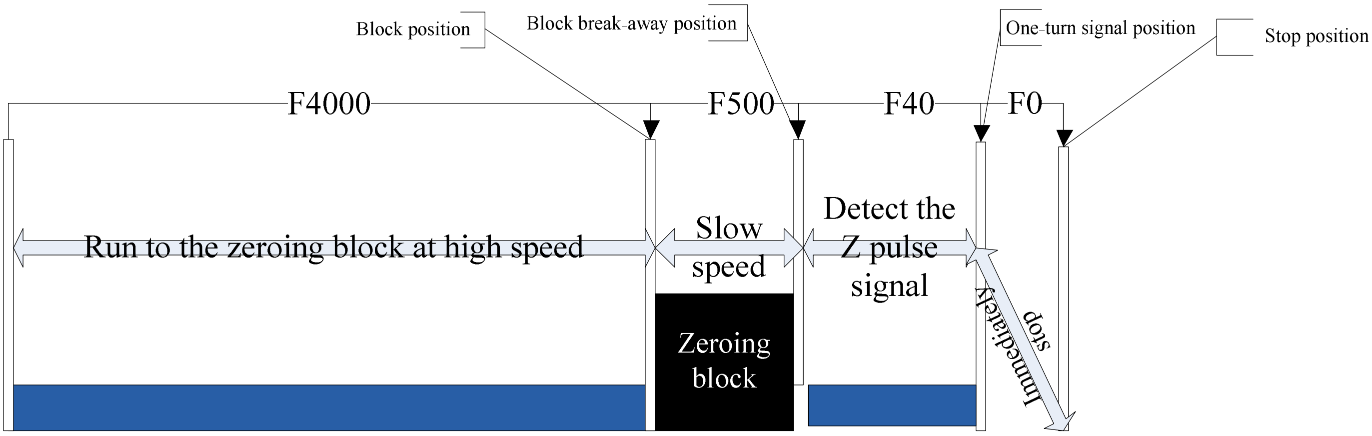


Fig. 9-2-2

9.3. Operation Steps For Mechanical Zeroing With Program Instruction

After the position parameter **NO**: **6#3** is set to **0**, the program instruction G28 can be used for zeroing, as the travel block is detected and is equivalent to the manual mechanical zeroing.

9.4 Bus Type Servo Zeroing Function Setting

There are three methods of zeroing for the system configured with bus servo, including normal zeroing, high-speed zeroing and multi-turn absolute zero setting. Such three methods are described below separately.

9.4.1 Normal Zeroing

Setting the position parameter No: **0#0=1,** position parameter **N.: 5#4=0**, the system will return to zero under the normal zeroing mode. The zeroing mode with or without one-turn signal may be selected. This zeroing mode can be used in system configured with DA98B, GE series incremental mode version. In the zeroing mode, the zeroing axis is valid.

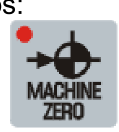
The specific operation steps are basically the same as the pulse type servo zeroing operation. Please refer to “9.2 Mechanical zeroing under pulse mode".

9.4.2 High-Speed Incremental Zeroing

Set the position parameter No: **0#0=1**, the system will return to zero under the high-speed zeroing mode. Only the zeroing mode with one-turn signal can be selected. This zeroing mode may be used in system configured with GE series incremental mode version.

In the zeroing mode, the zeroing axis is valid.

Zeroing steps:

1. Press the 

key to enter the mechanical zeroing operation mode. At this time, the words “mechanical zeroing” are displayed in the lower right corner of the LCD screen.

2. Select the X-axis, Y-axis, Z-axis, **4th**-axis or **5th-**axis to return to the mechanical zero. The zeroing direction is determined by the position parameter **N0:7#0-N0:7#4.**

3. The machine moves along the mechanical zero. Before reaching the deceleration point, it moves quickly (the moving speed is set by the data parameters **P100-P104**). After it touches the deceleration switch, the zeroing speed of each axis is set by the data parameters **P342-P346**. After breaking away from the block, it continues inquiring the one-turn signal position of Z pulse at the speed set by the data parameters **P342-P346** and decelerates and stops until it detects the above data; then, it returns to the mechanical zero (ie, the reference point) at the speed set by the data parameter **P354**. When it returns to the mechanical zero, the coordinate axis stops moving and the zeroing indicator light is on.

Example:

Take the high-speed increment zeroing of the 1st axis as an example. The 1st axis starts to hit the block with a higher speed F4000 (data parameter **P100** is set to 4000), and passes through the block at F500 (data parameter **P342** is set to 500) after touching the deceleration switch. After it breaks away from the block, it inquires the one-turn signal position of Z pulse at the speed F500 and decelerates and stops until it detects the above data; then, it returns to the mechanical zero (ie, the reference point) at the speed F200 (data parameter **P354** is set to 200), as shown in the Fig. 9-4-2-1.

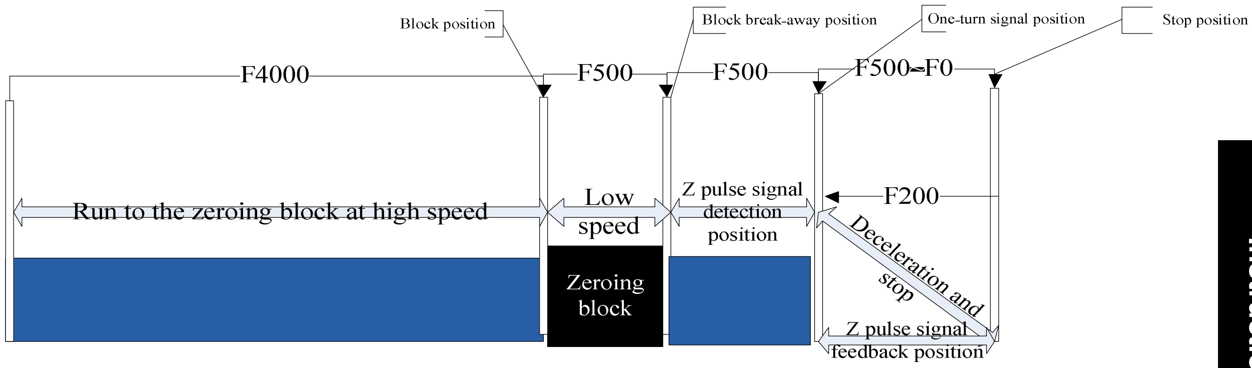


Fig. 9-4-2-1

9.4.3 Multi-Turn Absolute Zero Setting

Set the position parameter No: **0#0=1**

This zeroing mode can be set directly under the [⊞Bus configuration] interface. For details, please refer to 3.3.5 *Bus servo parameter display, alteration and setting.* In the zeroing mode, the zeroing indicator of each axis is on, indicating that the machine zero is successfully set.

Example:

For the zero setting of the absolute encoder, the zero position can be set according to the absolute position of the motor. As shown in the figure 9-4-3-1

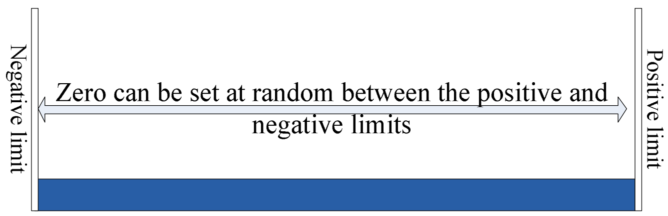


Fig. 9-4-3-1 Absolute encoder zero setting

Note 1: if your machine tool is not equipped with the zeroing deceleration switch or the mechanical zero is not set, please do not operate the mechanical zeroing.

Note 2: At the end of the returning to mechanical zero, the corresponding axis indicator light will be on.

Note 3: When the corresponding axis is not at the mechanical zero, the zeroing indicator light is off.

Note 4: for the direction of the mechanical zero (ie reference point), please refer to the machine tool manual provided by the machine tool manufacturer.

Note 5: the zeroing direction of each axis, the direction of the feed axis, and the gear ratio shall not be modified after the mechanical zero is set.

Note 6: the parameters related to mechanical zeroing and various mechanical zeroing methods are detailed in Section 4.8 of Part IV "Installation and connection" of the *PLC and Installation Connections Division of the Manual.*

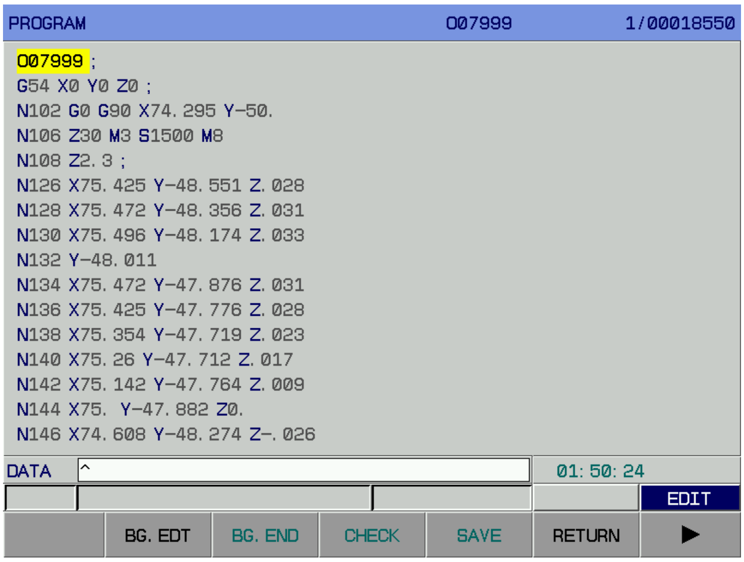
Chapter X Edit operation

10.1 Program Editing

The part program requires to be edited under the edit mode. Press the 

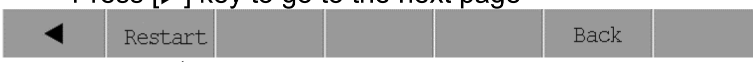
key to enter the edit mode; press the 

key on the panel to enter the program interface, press the [⊞Program] soft key to enter the edit and alteration interface of the program (see Fig. **10-1-1**).



Press [▶] key to go to the next page

Press [▶] key to go to the next page



Press [◀] key to return to the previous page

Fig. 10.1-1

Press the corresponding soft key to replace, cut, copy, paste, and restart the program.

Before the program is edited, the program switch must be opened for edit operation. Please refer to 3.4.1 of "Part II Operation instructions" of this Manual for operation details.

Note 1: as shown in Fig. 10-1-1, when the number of the first symbol "/" at the program segment is greater than 1, the system will skip the program segment even if the segment skipping function is not enabled.

Note 2: during debugging under the automatic mode, it is not allowed to switch to other modes, or otherwise it will cause accident.

Under the automatic mode, the debugging function is executed. When the program segment starts with a "/" symbol, program at the line after "/" will perform the debugging function regardless of whether the skip function is enabled.

10.1.1 Program Establishment

10.1.1.1 Automatic generation of sequence numbers (SN)

Set the “Auto SN” to 1 (see Fig. 10-1-1-1-1) as described in **3.4.1** of the “Part II Operation instructions” of this Manual.

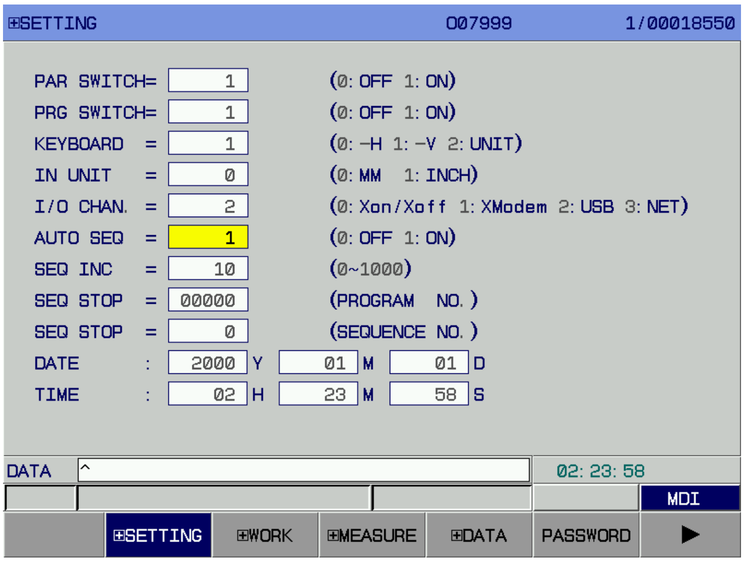


Fig. 10-1-1-1-1

Then, when the program is edited, the system will automatically insert SN between the segments, and the increment value of SN can be set in SN increment.

10.1.1.2 Input of program content

1. Press the 

key to enter the edit mode;

2. Press the 

key to enter the program page display (see Fig. 10-1-1-2-1).

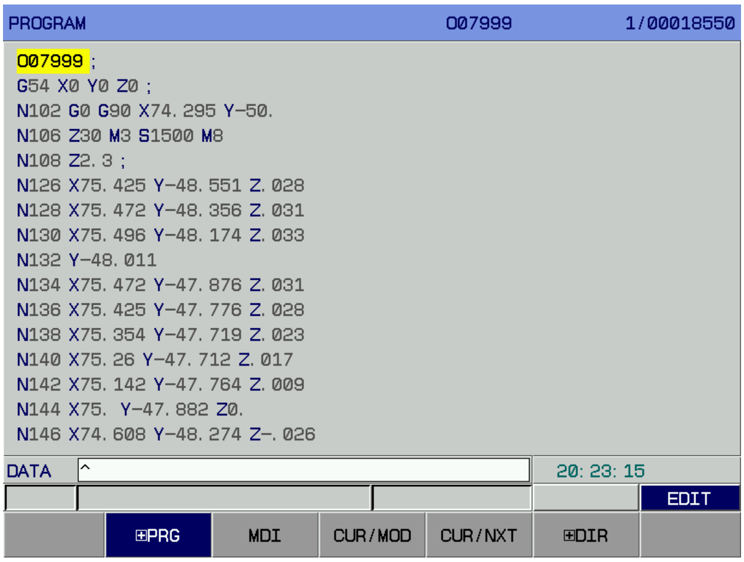
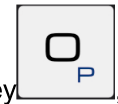
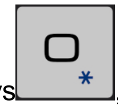


Fig. 10-1-1-2-1

3. Press the address key 

, and then type the number keys 

, 

, 

, 

and 

in order (taking established O00002 program name for an example) to display O00002 after the data column, as shown in Fig. **10-1-1-2-2**.

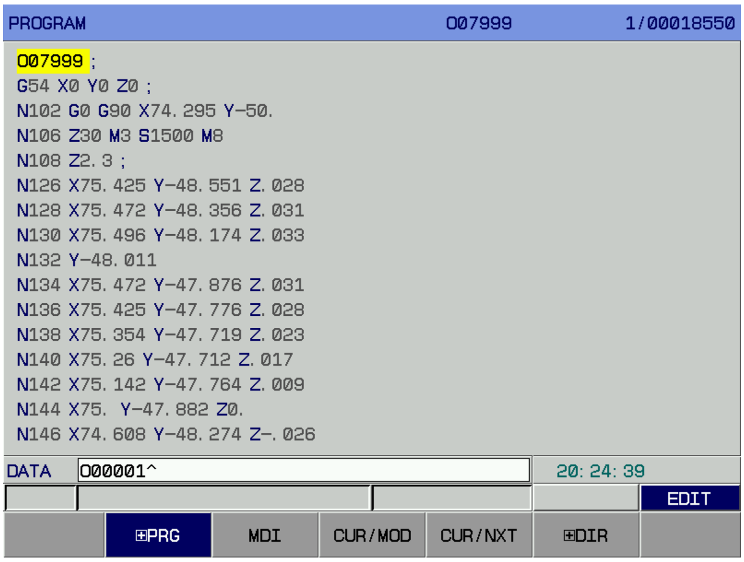
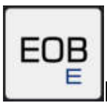


Fig. 10-1-1-2-2

4. Press the 

key to create a new program name, as shown in the figure below (see Fig. 10-1-1-2-3).

Volume II Operation Instructions

Fig. 10-1-1-2-3

5. Input the program to be written word by word. Switch to other working modes after the completion, and the program is automatically stored. For switching to other interfaces (such as the 

interface), firstly press 

key to save and complete the program input.

Note 1: in the edit mode, the system does not support the input of a separate number.

Note 2: When the program is input and the input code word is found to be in error, it is allowed to press 

key to cancel the input code.

Note 3: The maximum program segment to be input separately should not exceed 65 characters.

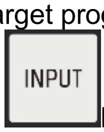
10.1.1.3 Search for SN, word and line numbers

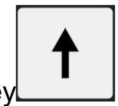
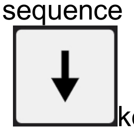
The SN search is to search certain sequence number within the search program. It is generally used to execute or edit a program from this sequence number. The program segment that is skipped due to search has no effect on CNC status. (Coordinate values, M, S, T codes and G codes in the skipped segment have no effect on the CNC coordinate values and modal values.)

If starting from a certain segment in the search program find out the machine status and CNC status at current time. For operation, it should be in line with the corresponding M, S, T codes and coordinate system settings (execution can be set by MDI mode).

The word search is used to search a specific address word or number in the program, and generally used to edit the program.

Steps to search the sequence number, word, and line number in the program:

1. Options: <**Edit**> or <**Automatic**> modes.
2. Find the target program in [Directory].
3. Press the 

key to enter the target program.
4. Enter the word or sequence number to be searched and press the direction key  or  key for searching.
5. To find the line number in the program, enter the line number to be searched and press the

SEARCH key to confirm.

Note 1: The search function will be automatically canceled when the sequence number and word are retrieved to the end of the program.

Note 2: The sequence number, word and line number can be retrieved under [Automatic] and [Edit] modes, but only can be performed in the background edit interface under the [Automatic] mode.

10.1.1.4 Cursor positioning method

Select the edit mode and press the PROGRAM key to display the program page.

a) Press the ↑ key to move the cursor up one line. If the column where the cursor is located is larger than the last column of the previous line, move the cursor to the end of the previous line.

b) Press the ↓ key to move the cursor down one line. If the column where the cursor is located is larger than the last column of the next line, move the cursor to the end of the next line.

c) Press the → key to move the cursor one column at the right. If the cursor is at the end of the line, move it to the beginning of the next line.

d) Press the ← key to move the cursor one column at the left. If the cursor is at the beginning of the line, move it to the end of the previous line.

e) Press the  key to scroll up and move the cursor to the previous screen.

f) Press the  key to scroll down and move the cursor to the next screen.

g) Press the HOME key and move the cursor to the beginning of the line where it is located.

h) Press the SHIFT + HOME keys and move the cursor back to the program beginning.

i) Press the END key and move the cursor to the end of the line where it is located.

j) Press the SHIFT + END keys and move the cursor to the program end.

10.1.1.5 Insertion, deletion, modification of words

Select <**Edit**> mode, press the PROGRAM key to display the program screen and locate the cursor at the position to be edited.

1. Word insertion

After entering the data, press the INSERT key, and the system will insert the input on the left side of the cursor;

2. Word deletion

Locate the cursor at the position for deletion, press the DELETE key, and the system will delete the content of the cursor.

3. Word modification

Move the cursor to the item to be modified, input the modified content, and then press the ALTER key, and the system will replace the content that the cursor positions with the content to be input.

10.1.1.6 Deletion of a single program segment

Select <**Edit**> mode, press the PROGRAM key to enter the program screen, and move the cursor to the beginning of the segment to be deleted, press N + DELETE keys to delete the segment where the cursor is located.

Note: Regardless of whether the segment has a sequence number, enter N and delete the program segment (the cursor must be at the beginning of the line).

10.1.1.7 Deletion of multiple program segments

From the currently displayed word, delete the segment to the specified sequence number.

N100 X100.0 M03 S2000; ..... N2233 S02 ; N 2300 M30 ;

Current cursor position

Delete the area

Fig. 10-1-1-7-1

Select <**Edit**> mode, press the PROGRAM key to enter the program display page, locate the cursor at the starting position of the target to be deleted (at the character N100 above), and then input the last complete character in the multiple segments to be deleted, such as S02 (see Fig. 10-1-1-7-1 above), and then press the DELETE key. Then delete the program between the cursor and the tag address.

Note 1: delete 100,000 lines of the program segment at most.

Note 2: if there are multiple identical characters to be deleted in the program, delete the program between the first complete character and the cursor character in the order of searching downwards.

Note 3: when deleting multiple program segments with N+ sequence number, the starting position of N+ sequence number of the target to be deleted must be at the beginning of the segment.

10.1.1.8 Deletion of multiple code words

From currently displayed codeword, delete to the specified code word.

N100 X100.0 M03 S2000; G01 X50.0 Y100.0 N2233 S02 ;

Current cursor position

Delete the area

Fig. 10-1-1-8-1

Select <**Edit**> mode, press the PROGRAM key to enter the program display page, locate the cursor at the starting position of the target to be deleted (at the character N100 above), and then input the last complete character in the multiple code words to be deleted, such as Y100.0 (see Fig. 10-1-1-8-1 above), and then press the DELETE key. Then delete the program between the cursor and the tag address.

Note: If the N+ sequence number is in the middle of the segment, the system will treat it as code word for processing.

10.1.2 Deletion of A Single Program

Deletion of a program in memory is subject to the steps as follows:

a) Select <Edit> operation mode;

b) Enter the program display page including two ways to delete the program:

1. Type the address O/P; enter the program name (type the number keys 0/*, 0/*, 0/* and 2/#, taking O0002 program as an example here), press the DELETE key to delete the program corresponding to the memory.
2. In the program interface, select the [Directory] interface, use the cursor to select the program name to be deleted, press the DELETE key, and the system status bar will prompt “Current file will be deleted, are you sure?” and then press the INPUT key; the status bar will prompt “delete successfully”, and then use the cursor to delete the program selected.

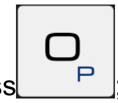
Note: If there is only one program file, regardless of the program name of O00001, in the edit mode program (directory) interface, program name will be changed to O00001 and the program content will be deleted after pressing the “Delete” key; if there are multiple program files, the program content and name of O00001 program are deleted together.

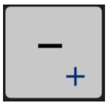
10.1.3 Deletion of All Programs

Deletion of all programs in memory is subject to the steps as follows:

a) Select <**Edit**> operation mode;

b) Enter the program display page;

c) Type the address ;

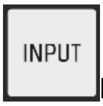
d) Type the address keys , , , ,  and  in order;

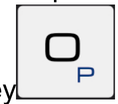
e) Press the  key to delete all programs in the memory.

10.1.4 Program Copying

Copy and save the current program as a new program name:

a) Select <**Edit**> mode

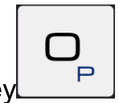
b) Enter the program display page; use the cursor to select the program to be copied in the [Directory] interface, and press the  key to enter the program display page;

c) Press the address key  to enter the new program number;

d) Press the [Copy] soft key to copy the file and enter a new program editing interface.

e) Go back to [Directory] to view the newly copied program name.

Copy the program on the program editing page (as shown in Fig. 10-1-1):

1. Press the address key  to enter a new program number;
2. Press the [Copy] soft key to copy the file and enter a new program editing interface.
3. Go back to [Directory] to view the newly copied program name.

10.1.5 Program Segment Copying and Pasting

Steps for copying and pasting program segments:

a) Move the cursor to the beginning of the segment to be copied.

b) Type the last character of the segment to be copied.

c) Press  +  key to complete copying the program between the cursor and the input character.

d) Move the cursor to the position to be pasted, press the  +  key, or press the [Paste] soft key to complete the pasting.

Copy and paste the program on the program editing page (as shown in Fig. 10-1-1):

1. Move the cursor to the beginning of the segment to be copied.

2. Type the last character of the segment to be copied.

3. Press the [Copy] key to complete copying the program between the cursor and the input character.

4. Move the cursor to the position to be pasted, press the [Paste] key to complete the pasting.

Note 1: if there are multiple identical characters to be copied in the program, copy the program between the first complete character and the cursor character in the order of searching downwards.

Note 2: if the program uses the N+ sequence number to copy, copy the program between the beginning of the cursor and the N+ sequence number. The N+ sequence number must be at the beginning of the block, and replication fails at other locations.

Note 3: copy 10,000 lines of the program segment at most.

10.1.6 Program Segment Cutting and Pasting

Steps for cutting program segments:
a) Enter the program editing page (as shown in Fig. 10-1-1).
b) Move the cursor to the beginning of the segment to be cut.
c) Type the last character of the segment to be cut.
d) Press the [Cut] soft key to cut the program onto the clipboard.
e) Move the cursor to the position to be pasted, press the [Paste] key to complete the pasting.

Note 1: if there are multiple identical characters to be cut in the program, cut the program between the first complete character and the cursor character in the order of searching downwards.

Note 2: if the program uses the N+ sequence number to cut, cut the program between the beginning of the cursor and the N+ sequence number.

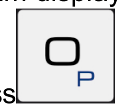
Note 3: in program interface under edit mode, when the program name is the same as the program content, the system can copy the characters following the program name, but not cut it.

10.1.7 Replacement of The Program Segment

Steps for program segment replacement:
a) Enter the program editing page (as shown in Fig. 10-1-1).
b) Move the cursor to the character to be replaced.
c) Type the replacement content.
d) Press the [Replace] soft key, the system will replace the content of the cursor positioning and all the same contents in the segment with the input content.

Note: This operation is only for the characters, but not for the entire program segment.

10.1.8 Program Renaming

Change the current program name to other name:
a) Select <**Edit**> operation mode;
b) Enter the program display page (use the cursor to specify the program name);
c) Type the address 

and enter a new program name;
d) Press the 

key to complete the file name change.

10.1.9 Program Restart

This function is used to return the program breakpoint in the dry running mode and continue executing the program after the system eliminates the accidents via the program restart function in case of any accident during the automatic running of the program, such as tool breakage, power failure, emergency stop, reset, etc. via the program restart function.

Operation steps for program restart:
1. Solve machine accidents. Such as changing tools, changing offsets, mechanical zeroing, etc.
2. In <**Automatic**> mode, press the 

key on the panel.
3. Press the 

key on the operation panel to enter the program interface, then press the [Program] soft key at the bottom of the LCD screen to enter the submenu. Press the [▶] key twice to scroll to the last page of the submenu and then press [Restart] soft key to enter the program restart interface. Record the code of the current modality different from that of the preloaded modality. (As shown in Fig. **10-1-9-1**).

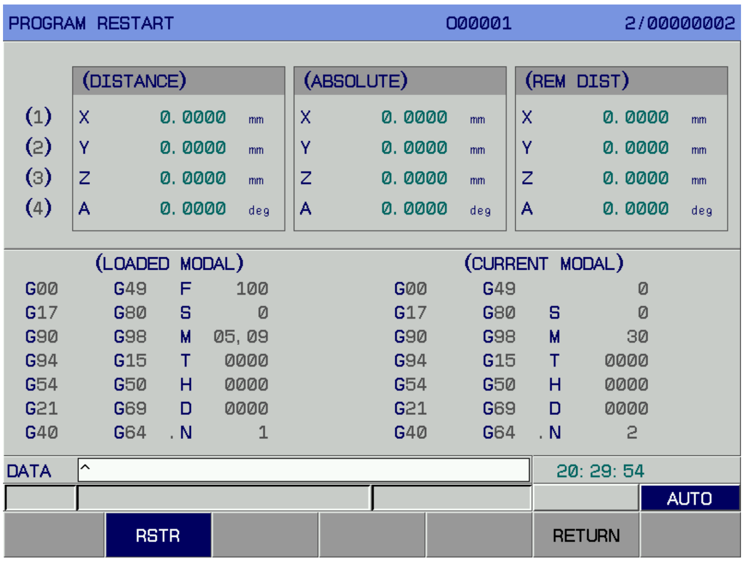


Fig. 10-1-9-1

4. Switch to <**MDI**> mode, press the [Present/module] soft key to enter the present module interface, input the corresponding modal code and M code according to the preloaded modal value in Fig. 10-1-9-1.

5. Return to <**Automatic**> mode, press the 

key on the panel, and then the 

key on the panel, the program will move to the starting point of the interrupted segment (ie the breakpoint of the previous segment) in the order of movement (1), (2) and (3) before the coordinates at the dry running speed, then restart the machining.

Description:

1. (1), (2), and (3) before the coordinate system are the order of movement of each axis to the restart position of the program, and their order is set by the data parameter **P376**.
2. When the restart position of the coordinate axis moves, the tool will stop each time when it finishes the movement in an axial direction. It is not possible to switch to MDI mode for intervention during execution.
3. The Z axis movement mode can be controlled by the position parameter **NO.49#0**. (0: G00, 1: G01), when the G00 mode is selected, the tool moves to the restart position of the program at the dry running speed in the order specified in the parameters, and then restarts machining.

Note 1: the operation can be performed anywhere; therefore, check whether the tool may collide with the workpiece or other objects when moving to the program restart position. If possible, move the tool to a place without any obstacle before executing the program and restart.

Note 2: the program segment restarted by the program is not necessarily the segment that has been interrupted midway, and it can restart from any segment for running. The method is the same as above, except that in the “MDI” mode of the step 4; firstly use the direction key "↓" to directly define the N line number of the preloaded modal value, and press the “Input” key to confirm. Then, enter the present module interface to input the corresponding modal code and M code.

Note 3: during the period from program segment retrieval in restart to the execution of restarting the program, do not execute reset. Otherwise, the program start must be re-executed from the first step.
Note 4: if the machine is not provided with the absolute position detector (absolute encoder), the reference point return must be performed before restart and after power-on.
Note 5: the program restart function does not support programs containing subprograms;
Note 6: the program restart function does not support programs with rotation, mirroring, scaling, and polar coordinate modes;
Note 7: the program restart function does not support fixed cycle program;
Note 8: the program restart function does not support DNC online processing program;
Note 9: the program restart function does not support macroprogram (including Class A, Class B).

10.2 Program Management

10.2.1 Retrieval of Program Catalog

Press , and press [Catalog] in the program interface to enter the program catalog display page (see Figure 10-2-1-1).

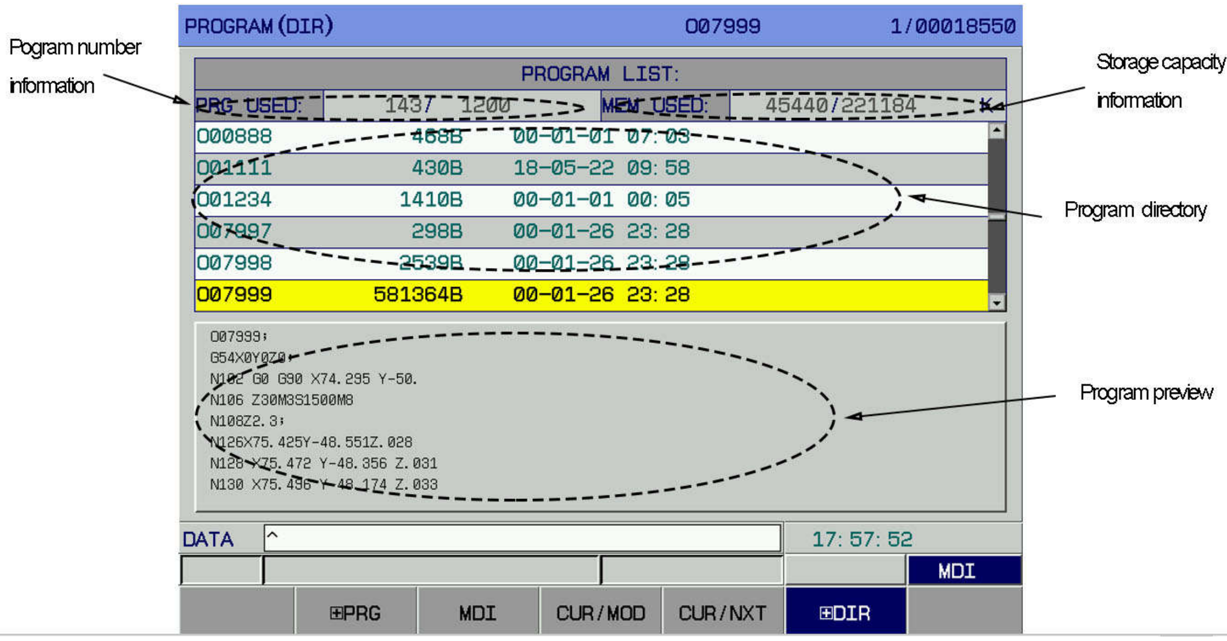


Figure 10-2-1-1

1） Open Program
Open a specified program: O+S/N+Input (or EOB) or S/N + Input (or EOB).
Program will be created if the entered S/N does not exist in Edit mode.
2） Delete a program: 1. In Edit mode, press DEL to delete the program pointed by cursor.
2 In Edit mode: O+S/N+DEL or S/N+DEL.

10.2.2 Number of Stored Programs

The number of stored programs in this system cannot exceed 400. For the number of currently stored programs, see “No. of programs” on the program catalog page in **10.2.1**.

Volume II Operation Instructions

10.2.3 Storage Capacity

For the specific storage capacity, see “Storage capacity” on the program catalog display page in **10.2.1**.

10.2.4 View The Program List

The program catalog page can display up to 6 **CNC** program names at a time. If there are more than 6 **CNC** programs, the names will not be fully displayed in one page. Press the page-turning button, and the LCD will display the **CNC** program names on the next page. Press the page-turning button repeatedly, and the LCD will display all **CNC** program names in cycle.

10.2.5 Lock A Program

To prevent the user program from being modified or deleted by others without authorization, the system has set the program switch. After the program is edited, turn off the program switch to lock the program. Then the user cannot edit the program. For details, see “**3.4.1 Instructions**”.

Chapter XI System Communication

The system supports three communication interfaces: RS232, USB, and Network port transmission, which communicate with the PC or U disk respectively to transfer data.

11.1 Introduction To GSKComm

GSKComm communication software is a communication management software specially designed for users, supporting RS232 serial port and Network transmission connection. It enables file uploading and editing between PC and CNC. It features easy operation, high communication efficiency and reliability. The software uses Windows interface and runs on Win7, WinMe, WinXP and Win2000.

Run the **GSK218MC**Comm.exe program directly. After the program starts, the interface is shown as below:

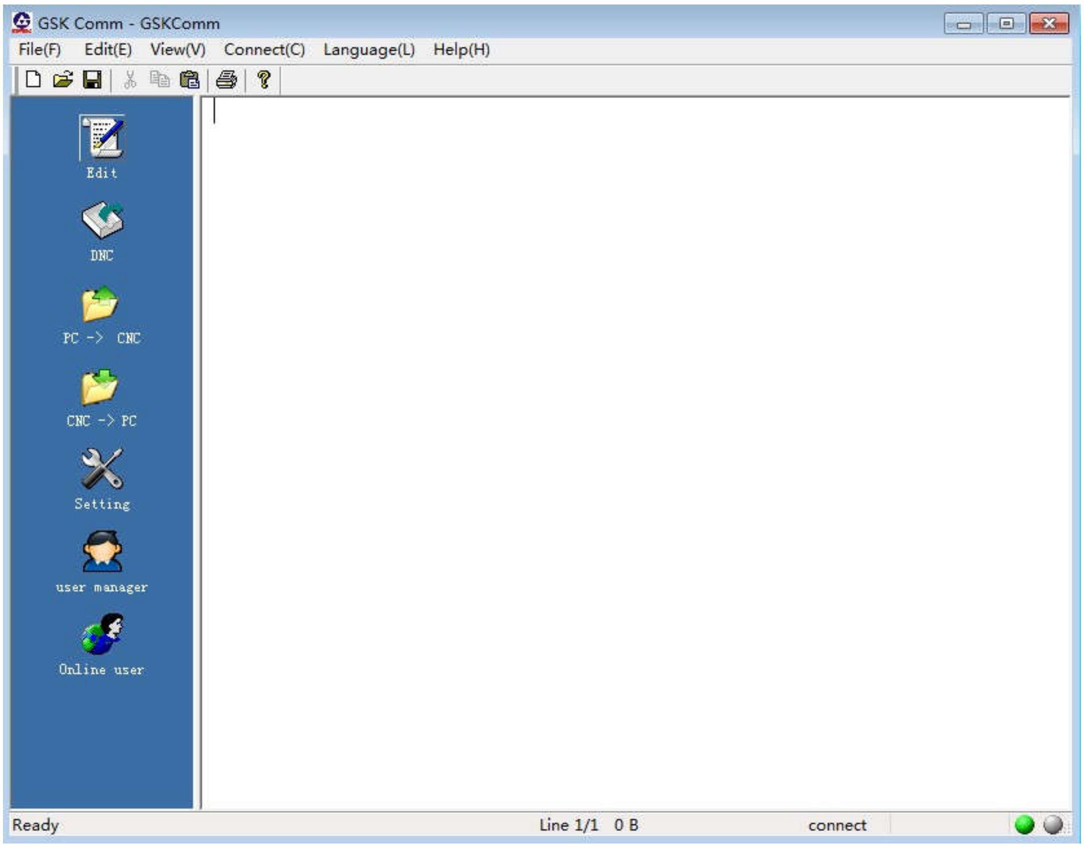


Figure 11-1-1

11.1.1 Functions

1. File Menu
   File menu includes New, Open and Save program files, Print and Print Setting, and Opened Files.
2. Edit Menu
   Edit menu includes Undo, Cut, Copy, Paste, Select All, Find and Replace.
3. View Menu
   Display the toolbar and status bar.

4. Connect menu
   Mainly connect/disconnect the serial port/Network port.
5. Main menu
   Include Edit File, DNC Transmission, PC-CNC Transmission, CNC-PC Transmission, Software and Serial Port Setting, User Management and Count. Click the small black triangle on the main menu to view the menu contents.
6. Help
   Version information of the software.

Note: Network port transmission is only available when the CNC system has a Network port.

11.1.2 Edit

Click  on the main menu to enter the file edit interface. "Edit" allows the user to create a new part program file or open an existing part program file to be edited.

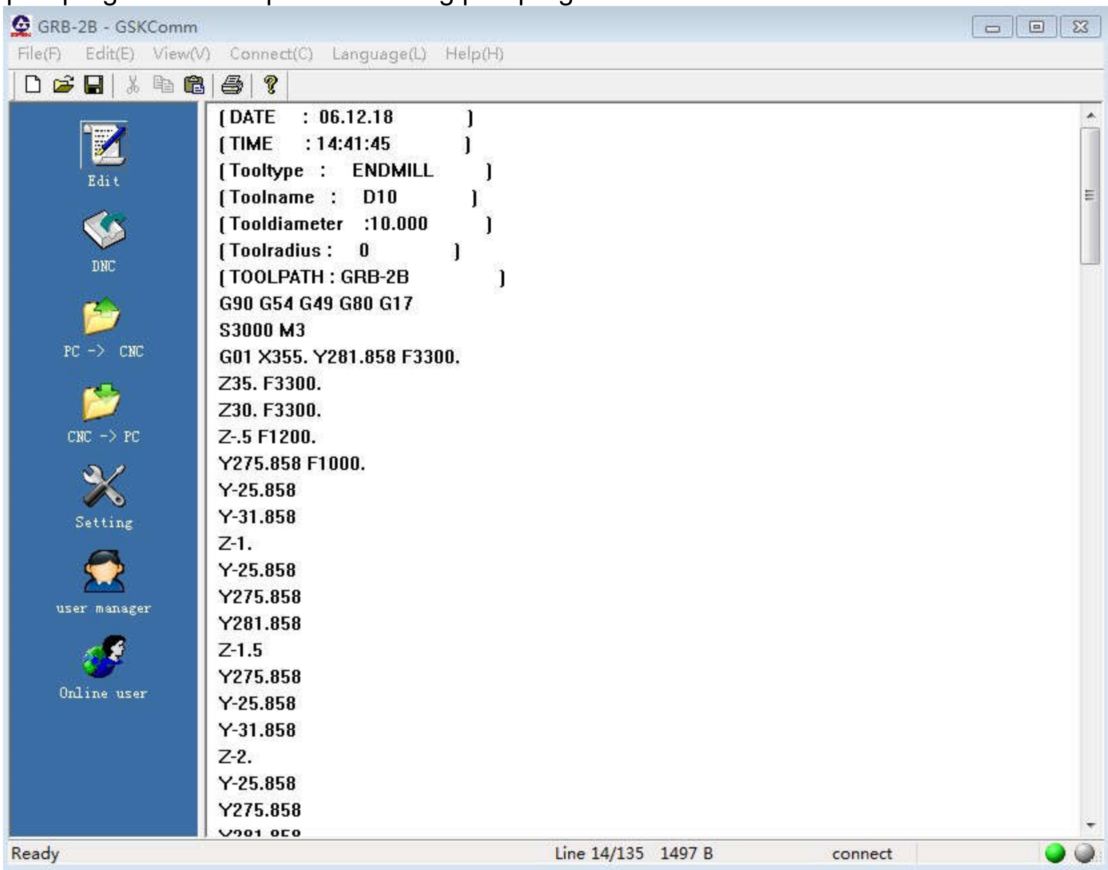
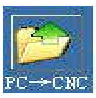


Fig. 11-1-2-1

11.1.3 PC--CNC Send File

Click  on the main menu to enter the file sending interface. The user can use the shortcut menu bar on the right of the "Send File" interface, or move the cursor to the file display field in the middle of the interface, right click, select the corresponding operation in the pop-up context menu or directly press Send.

Press **[Add...]** to add more files. The user can choose to add a single file or several files at once. If the added file name does not match the rule or the file size exceeds 4M, the file list item will show Red, and the second column will display "×". If the file name and the size are both in compliance with the rule, the second column of the list item will display "√" (as shown in Figure 11-1-3-1).

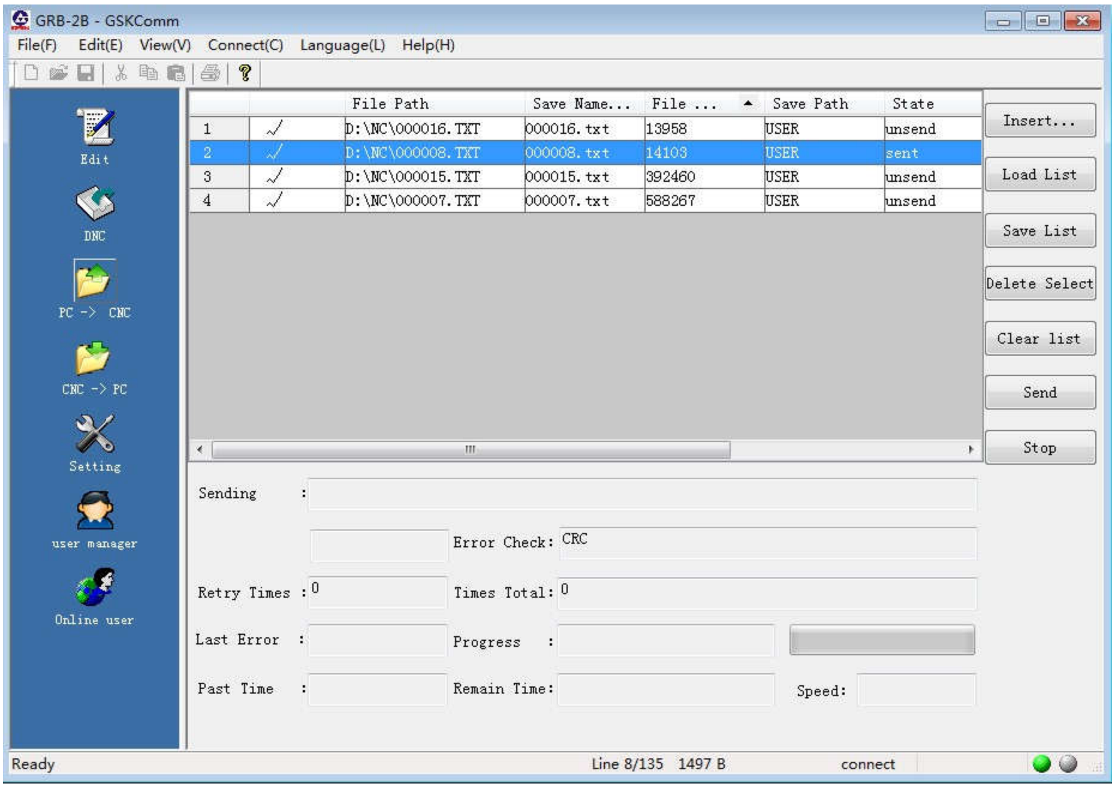


Fig. 11-1-3-1

Press **[Load File List]** to add the list of saved files; press **[Save List]** to save the current file list; press **[Delete from List]** to select a single file or several files at once, and delete the selected list items from PC-CNC file list; press **[Clear List]** to clear the entire PC-CNC file list; press **[Send]** to send the selected file to CNC; press **[Stop]** to stop the ongoing transmission. The user can sort the list of sent files by clicking the header of the sent file list. After sorting, a black triangle will be shown in the head of list: the upper triangle represents ascending order, and the lower triangle represents descending order (as shown in Figure 11-1-3-2).

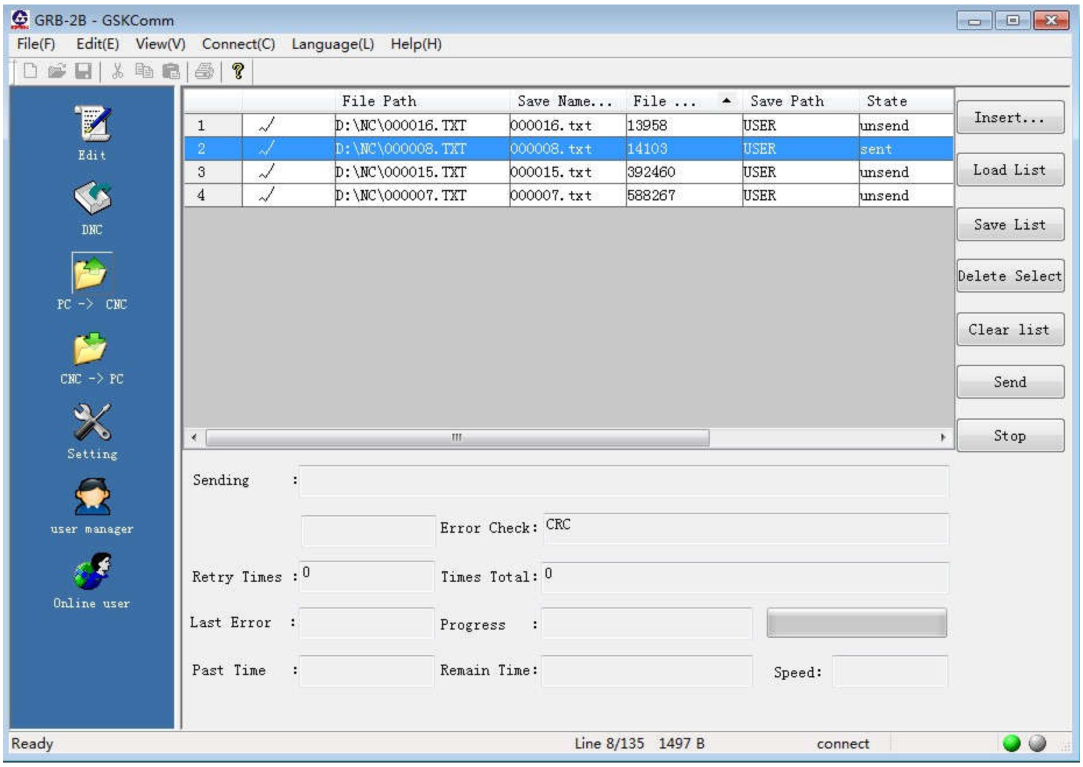


Fig. 11-1-3-2

To modify the file path, the file name saved or storage area in CNC, double-click the list item to be modified, and modify it in the pop-up dialog box as shown in Figure 11-1-3-3.

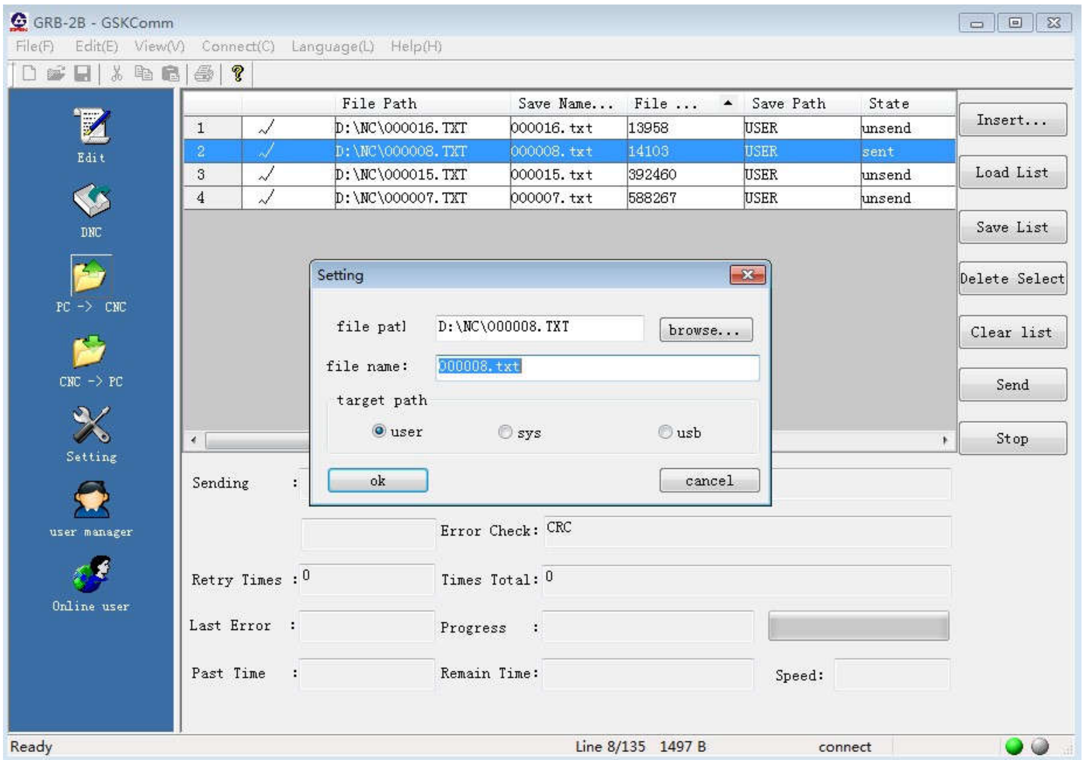


Fig. 11-1-3-3

If the file sent by the user has the same name as that in CNC system, a dialog box as shown in Figure 11-1-3-4 will pop up during the sending process. The user can press “Yes” to directly overwrite it, or press "No" to rename the file, or press "Cancel" to skip the file.

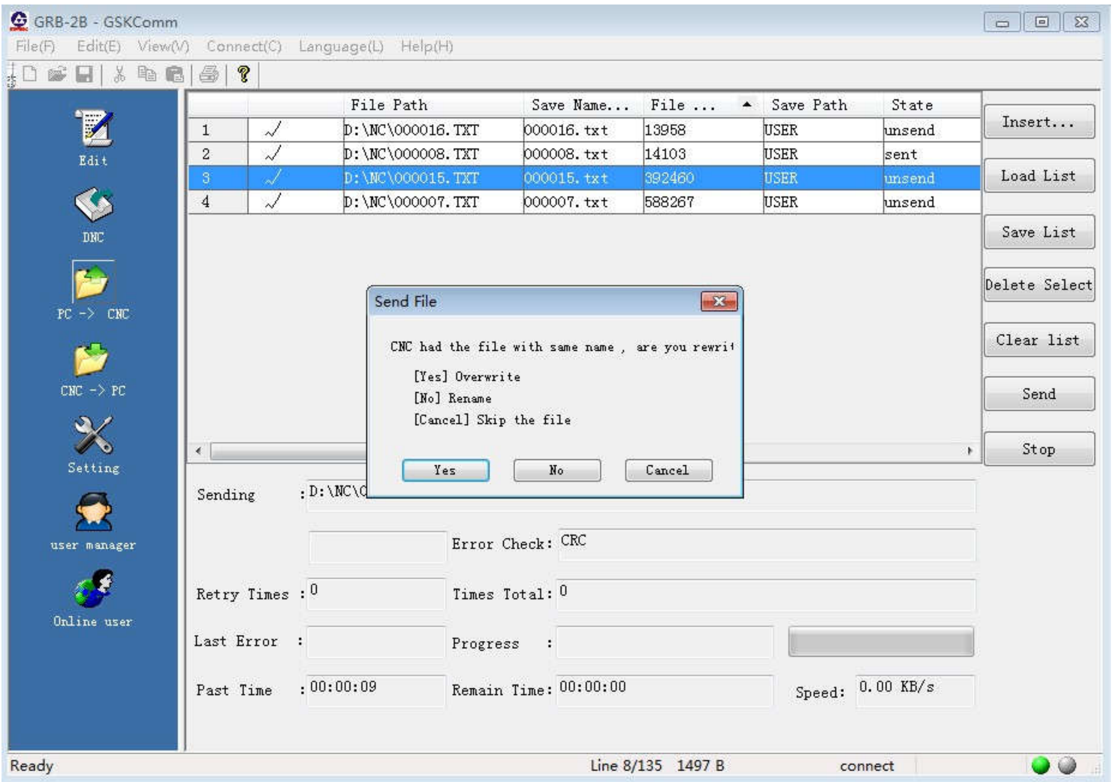


Fig. 11-1-3-4

11.1.4 CNC--PC Receive File

Press **[Obtain CNC List]** to obtain the list of files in CNC system; press **[Only Delete from List]** to delete the selected list items from the list of received files; press **[Delete CNC File]** to delete the selected files from the file list and from CNC system; press **[Receive]**, and a dialog box will pop up (Figure 11-1-4-1) to let the user select the storage location of the received file; press **[Stop]** to stop the on-going transmission.

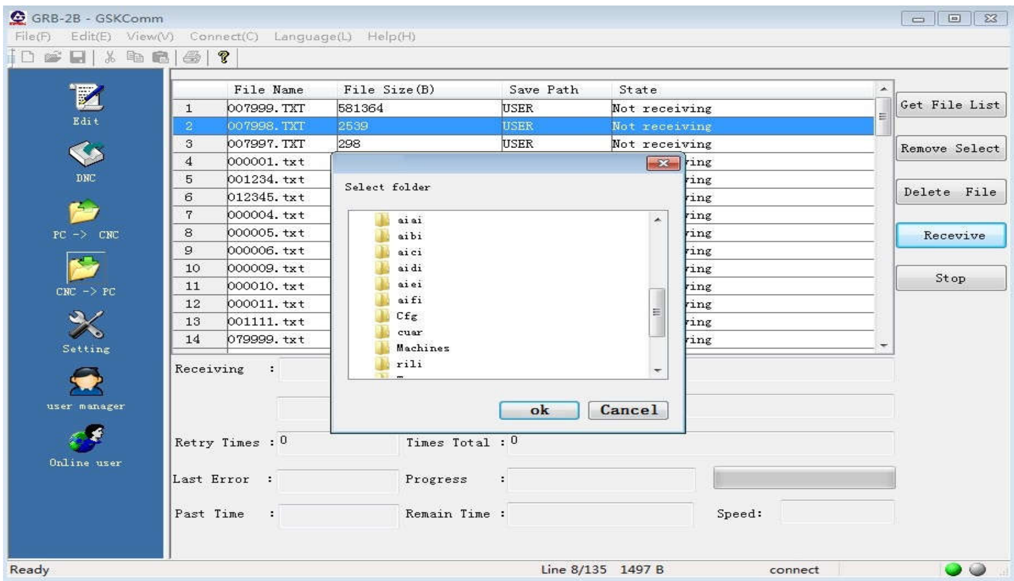


Fig. 11-1-4-1

11.1.5 User Management Setting

Click [Add] to set multiple user names, passwords and file paths; click [Edit] to change the set user names, passwords and file paths; click [User Permssions] to select downloading, uploading or deletion of files through permissions; click [Delete] to delete multiple user names from the list, as shown in Figure 11-1-5.

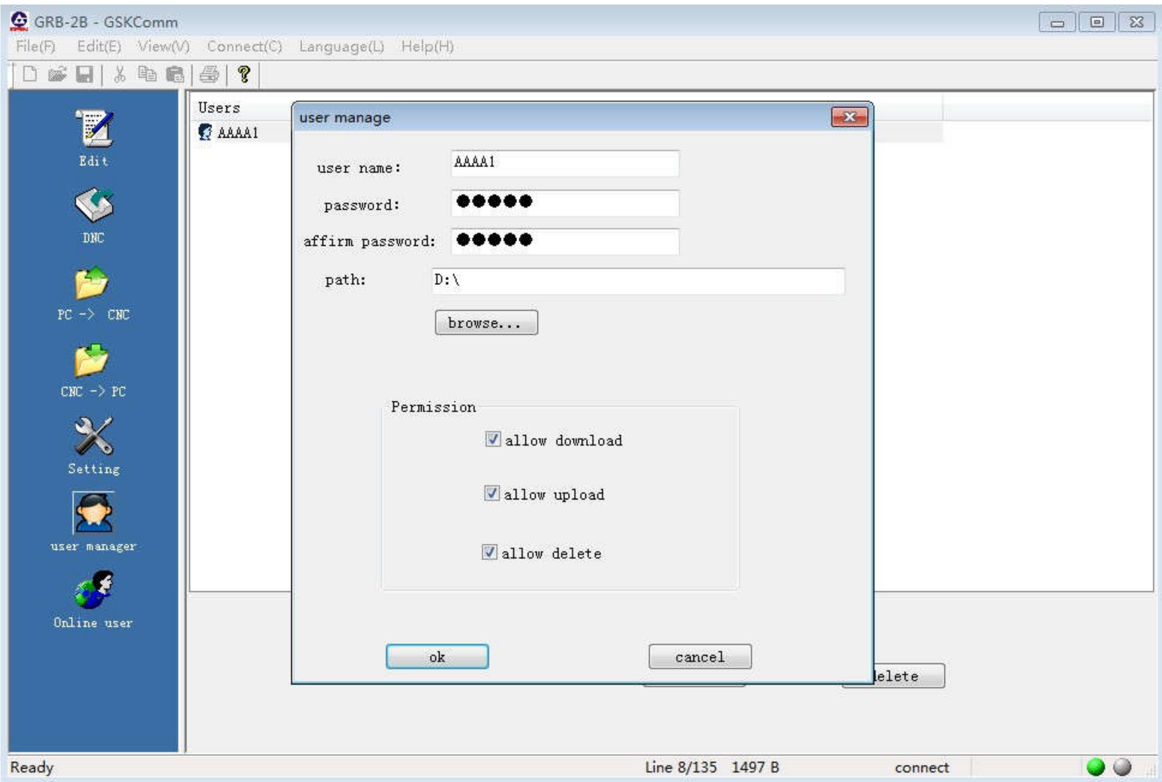


Fig. 11-1-5-1

To set multiple user names, click [Add] on the blank of the user name list; to modify the user name, password or file path, double-click the list user name or click [Edit], and a dialog box as shown in Figure 11-1-5-2 will pop up, on which modification can be made.

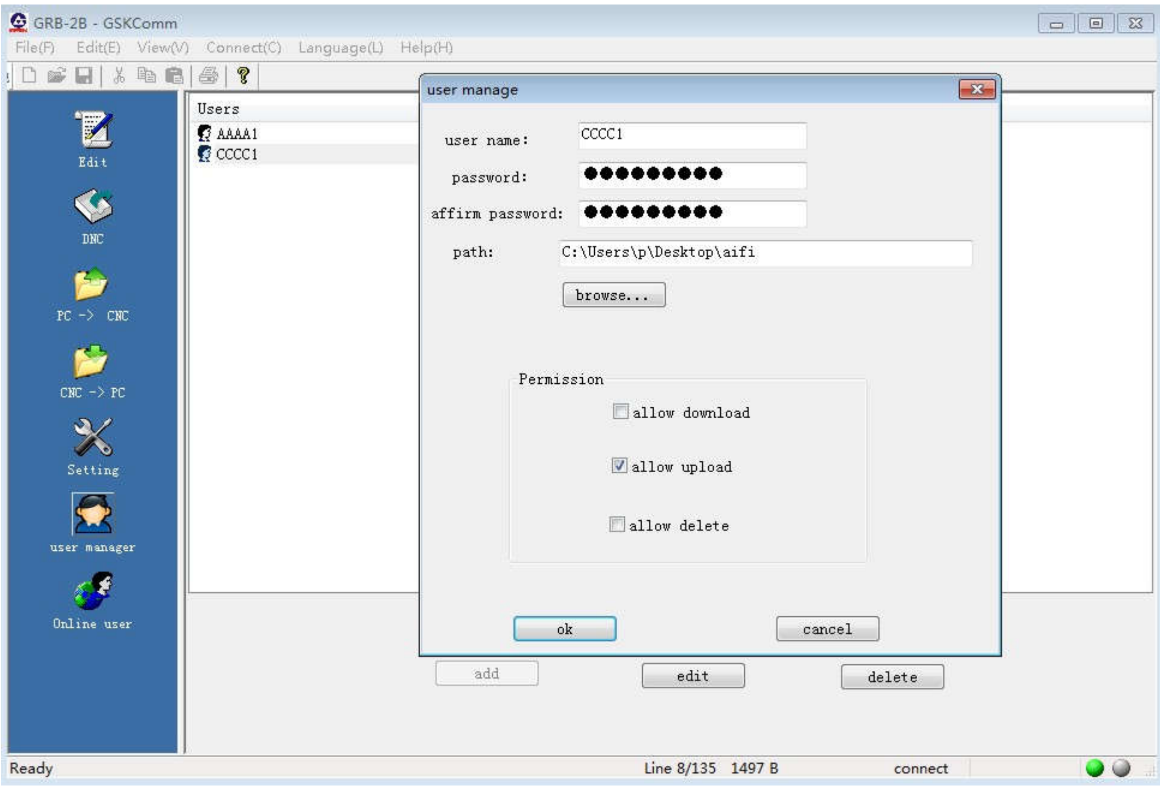


Fig. 11-1-5-2

11.1.6 Software and Serial Port Setting

The Set page is shown in Figure 11-1-6-1, where the use can set software and serial ports.

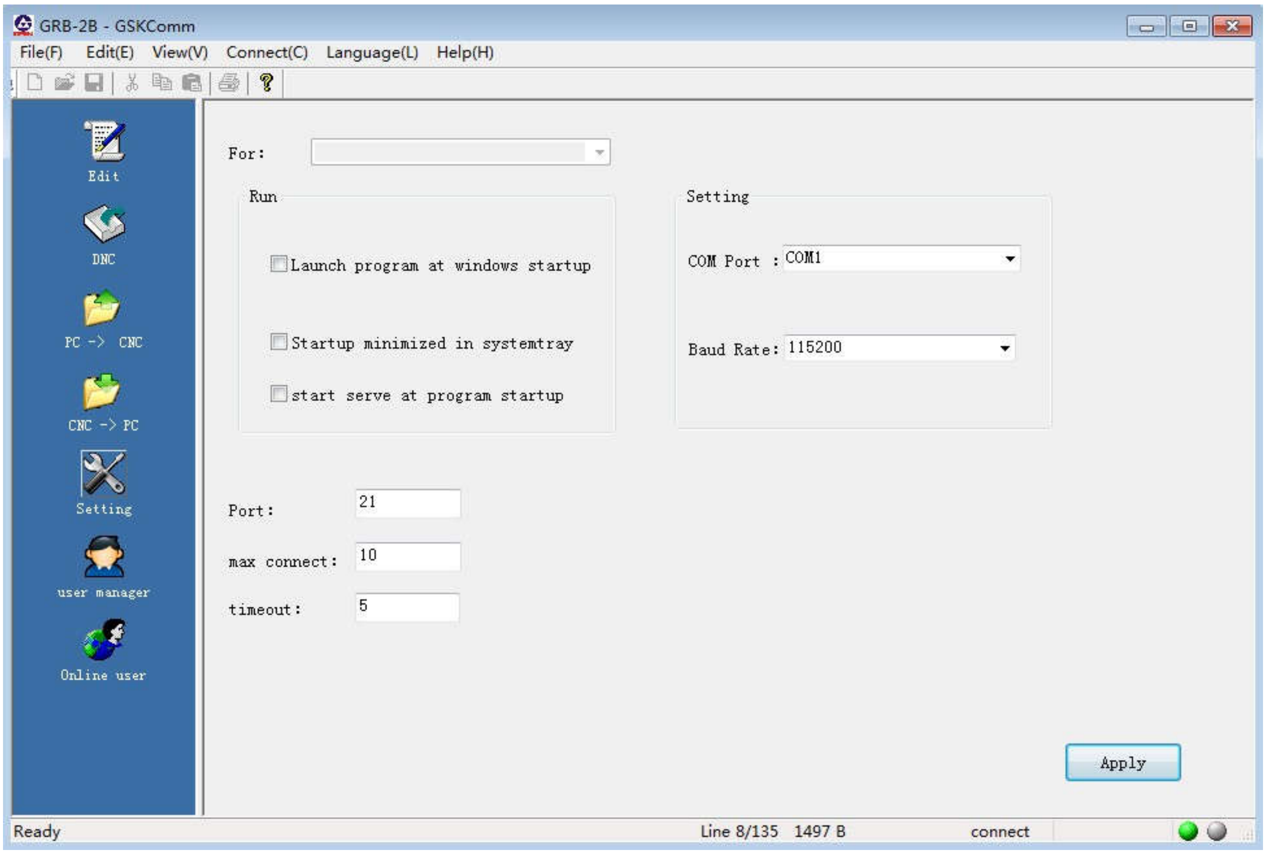


Fig. 11-1-6-1

Program Startup: the user can set whether to automatically run the software at booting, whether to automatically minimize the software to the lower right corner of the screen when it is started; communication: the user can select the serial port, and set baud rate of the serial port. (Click “Apply” after setting is finished, otherwise it will not work)

Note: The system does not support auto starting of the server when the program is running.

11.2 Serial Communication

11.2.1 Preparation For Serial Communication

1. Connect the PC Serial Port (COM port) to RS232 interface of the system with a serial line.
2. Open “GSK Comm” communication software on the PC.
3. Setting of “GSK Comm” communication software:

(1) Baud rate setting:

Click “Set” in the software to enter the Set page to set the serial communication;

Port selection: Select the port for communication through “Serial Port No.” drop-down menu (the computer port that can be used is automatically recognized by the software);

Baud rate setting: Select through “Baud Rate” drop-down menu to ensure the same baud rate for PC and CNC communication. Standard factory setting: Select baud rate 115200 for data transmission (corresponding to the default parameter P0002 of CNC).

Select baud rate 38400 for DNC online processing (corresponding to the default parameter P0001 of CNC).

Finally, click  to save the settings.

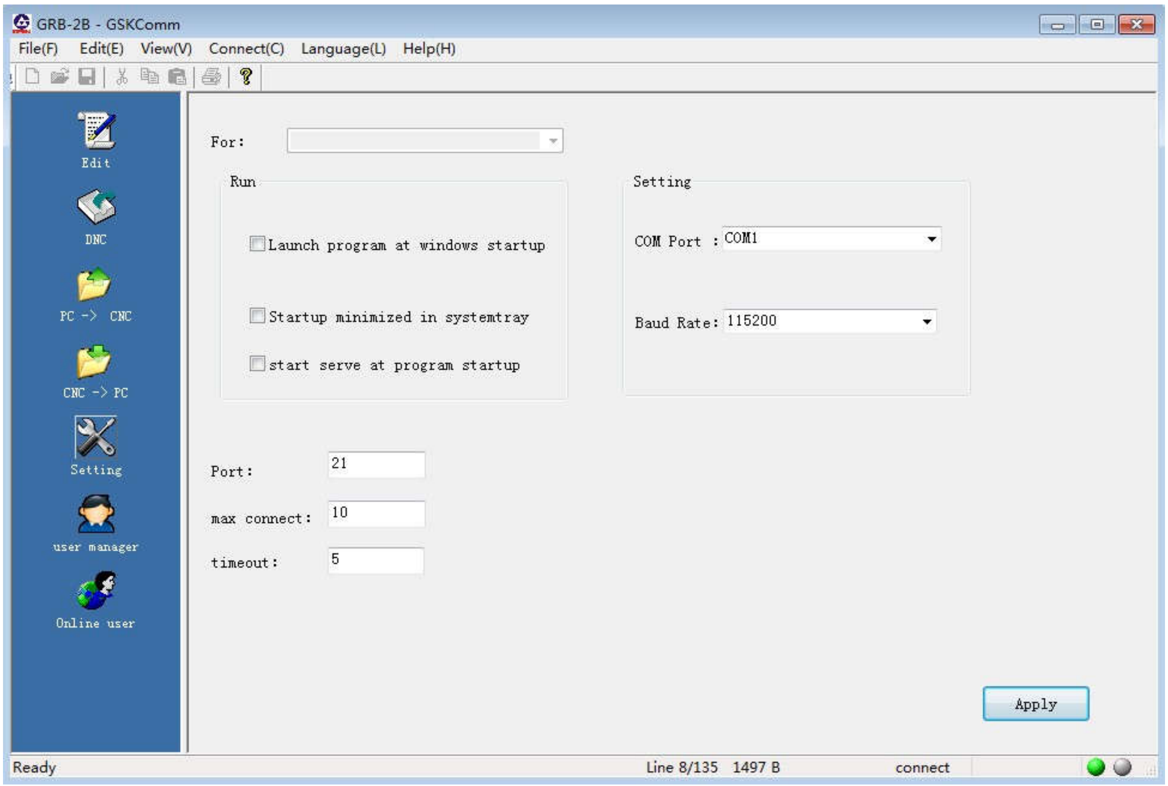


Fig. 11-2-1-1

(2) Click “Connect” menu and select “Via serial port”. If the serial port is successfully opened, the status bar will display “Serial port is open”, and the small icon in the lower right corner will also turn “green” and “gray”, but this only indicates that the local serial port has been opened, but not necessarily means that a connection has been established with the CNC system.

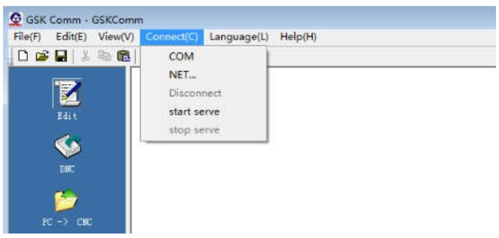


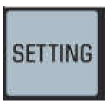
Fig. 11-2-1-2

(3) Click “Connect” menu, select “Disconnect” to disconnect the CNC system.

11.2.2 Serial Data Transmission

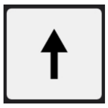
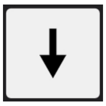
Operating steps:

1) Select <Enter> operation mode;

2) Press  to enter the Set page, and set the I/O channel to 1.

3) Select baud rate 115200 for data transmission

4) Press [Password] to enter the (password) setting page and enter a password of the corresponding permission level. For details, Please refer to "3.3.1.1 Setting and modification of password permission".

5) Press [Data] to enter the (data processing) setting page, press the arrow key  or  to move the cursor to **<CNC Part Program>**.

A. Data output (CNC→PC)

1. Press soft key [Data Output], and the system prompts "Waiting for transmission".

2. Click  on GSK Comm communication software to enter the Receive File page.

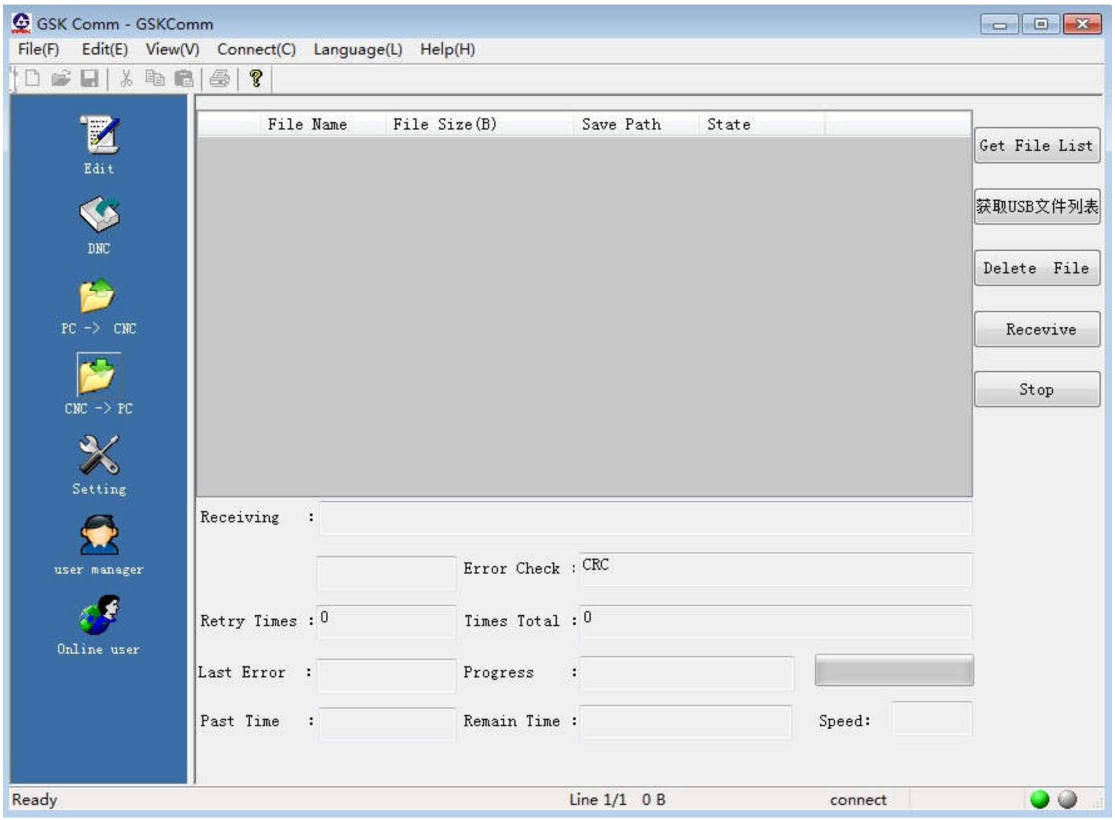


Fig. 11-2-2-1

3. Click  to get obtain the file list on CNC, as shown in Figure 11-2-2-2.

	File Name	File Size(B)	Save Path	State
1	007999.TXT	581364	USER	Not receiving
2	007998.TXT	2539	USER	Not receiving
3	007997.TXT	298	USER	Not receiving
4	000001.txt	10	USER	Not receiving
5	001234.txt	1410	USER	Not receiving

Fig. 11-2-2-2

4. Select the file to be received (multiple files can be received at the same time), then press 

, and a dialog box pops up for the user to select the storage location of the received file, and file receiving begins, as shown in Figure 11-2-2-3.

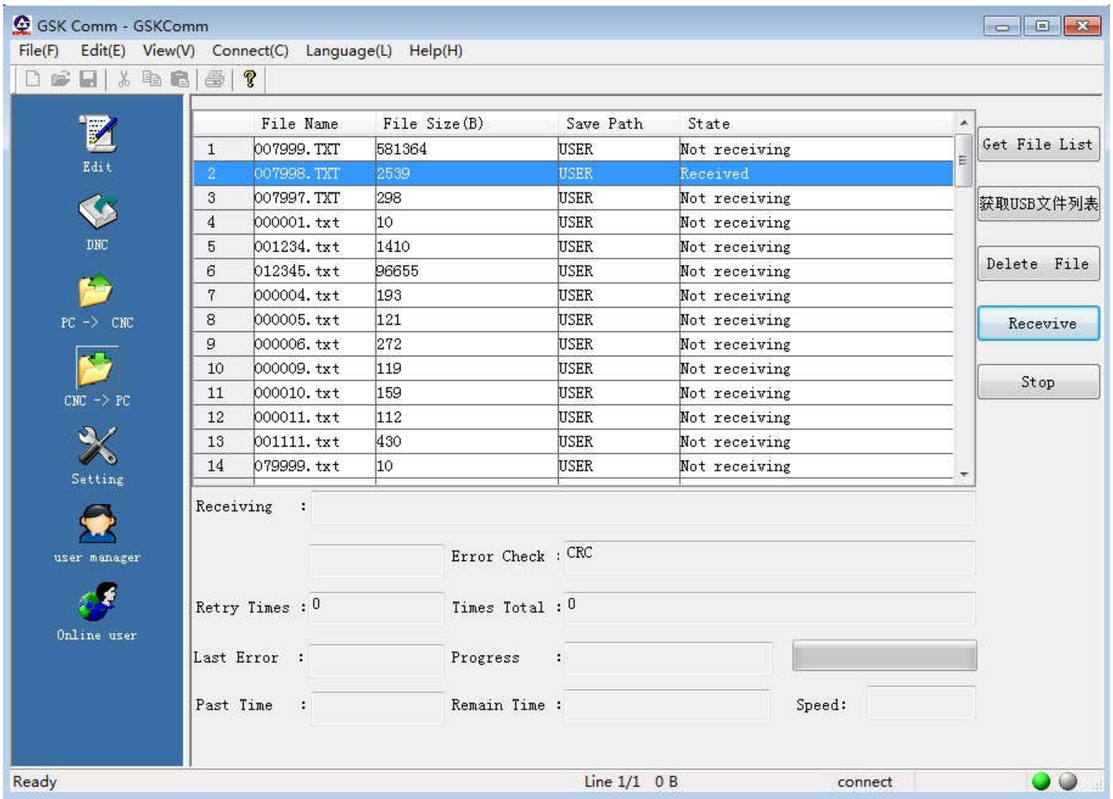
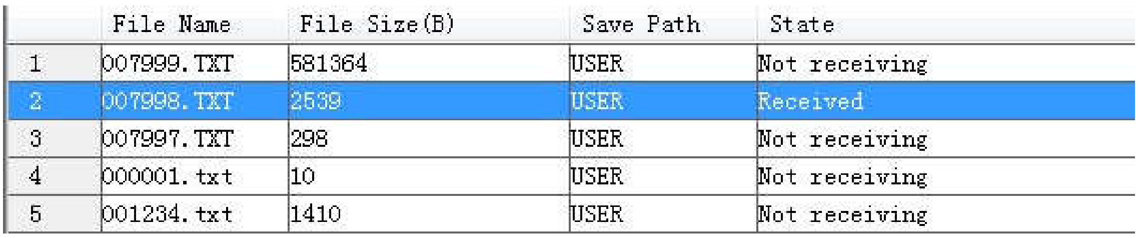


Fig. 11-2-2-3

5. After the file is received, the status bar of the file list item displays "Received", as shown in Figure 11-2-2-4.



	File Name	File Size(B)	Save Path	State
1	007999.TXT	581364	USER	Not receiving
2	007998.TXT	2539	USER	Received
3	007997.TXT	298	USER	Not receiving
4	000001.txt	10	USER	Not receiving
5	001234.txt	1410	USER	Not receiving

Fig. 11-2-2-4

B. Data input (PC→CNC)

1. Press soft key [Data Input], and the system prompts "Waiting for transmission".

2. Click 

on GSK Comm communication software to enter the Send File page.

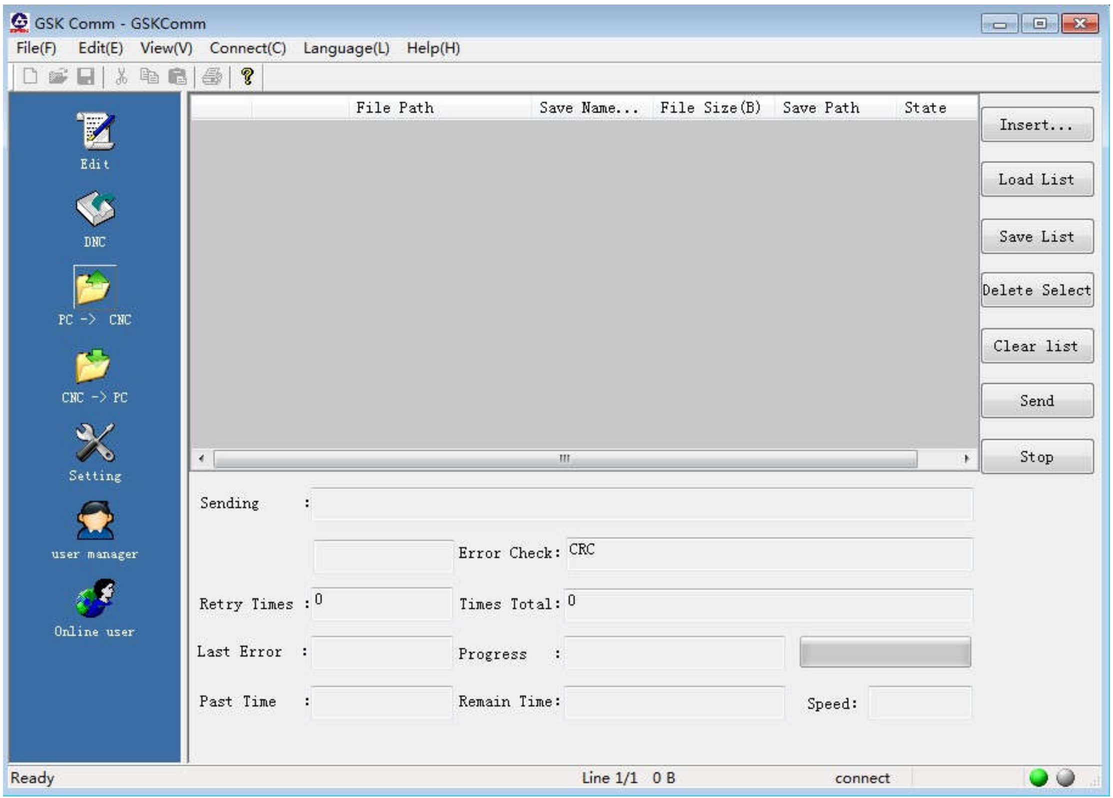


Fig. 11-2-2-5

3. Click Add... to add all files to be sent to CNC. As shown in the figure 11-2-2-6.

		文件路径	存储为(CNC)	文件大小(B)	存储区	状态
1	√	C:\Documents and Sett	007999.txt	577237	USER	未发送
2	√	C:\Documents and Sett	007998.txt	2460	USER	未发送
3	√	C:\Documents and Sett	000251.txt	171649	USER	未发送
4	√	C:\Documents and Sett	000247.txt	896	USER	未发送
5	√	C:\Documents and Sett	000170.txt	621058	USER	未发送
6	√	C:\Documents and Sett	008010.txt	1408	USER	未发送
7	√	C:\Documents and Sett	000001.txt	201	USER	未发送

Fig. 11-2-2-6

4. Double-click the sent file list item to modify the file path, the file name saved or storage area in CNC.

To send CNC part programs and user macro programs, user partition should be selected; when sending ladder diagram (PLC), parameter (PLC), system parameter value, tool offset value, pitch offset values, system macro variable, etc., system partition should be selected.

5. After the partition is selected, select the file to be sent (multiple files can be sent at the same time), click Send, and file sending starts. As shown in the figure 11-2-2-7.

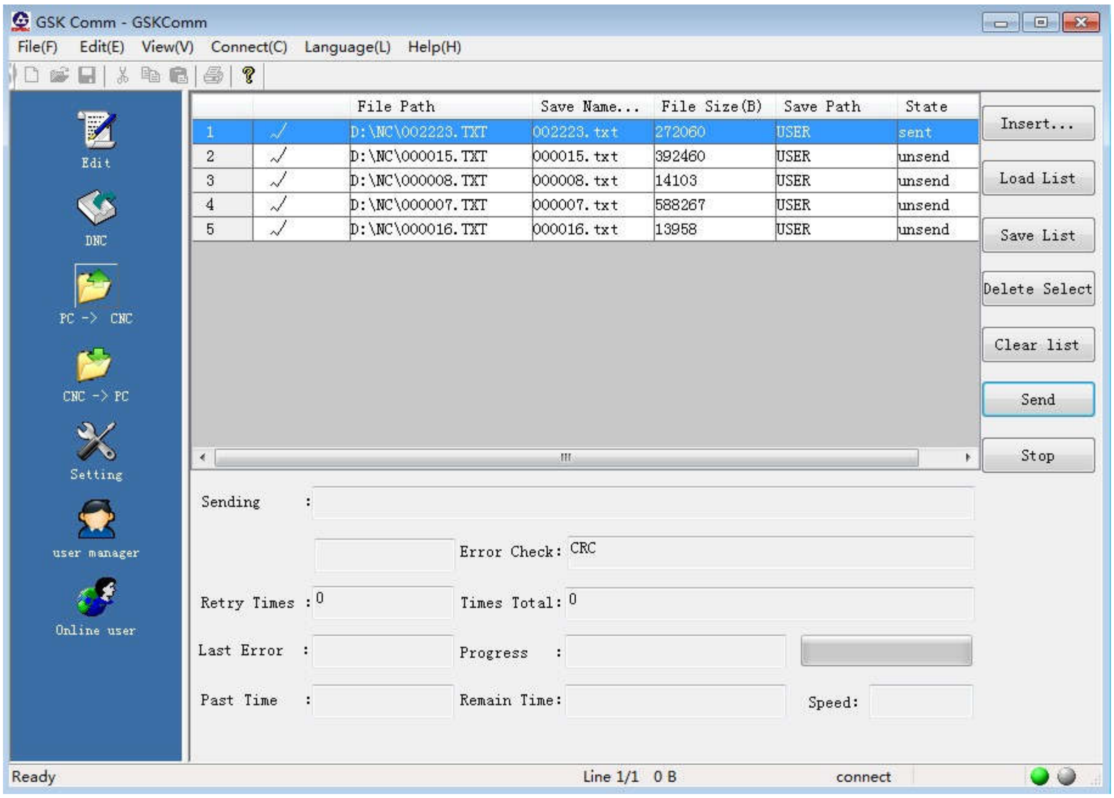


Fig. 11-2-2-7

6. After the file is sent, the status bar of the dialog box displays “Sent”. As shown in the figure 11-2-2-8.

		File Path	Save Name...	File Size(B)	Save Path	State
1	√	D:\NC\002223.TXT	002223.txt	272060	USER	sent
2	√	D:\NC\000015.TXT	000015.txt	392460	USER	unsend
3	√	D:\NC\000008.TXT	000008.txt	14103	USER	unsend
4	√	D:\NC\000007.TXT	000007.txt	588267	USER	unsend
5	√	D:\NC\000016.TXT	000016.txt	13958	USER	unsend

Fig. 11-2-2-8

Note 1: For other functions of the Send File page, please refer to “11.1.4 PC--CNC Send File”; for other functions of the Receive File page, please refer to “11.1.4 CNC--PC Receive File”.

Note 2: Before data transmission, please ensure that the baud rate is set correctly and the serial cable is connected reliably.

Note 3: Do not switch the operation mode or pages of the system during data transmission, otherwise it may cause data transmission error.

Note 4: LADCHI**.TXT file is invalid after transferred to the system, and will become valid after powering off.

11.3 Network Port Transmission

11.3.1 Preparation For Network Port Transmission

1. Open “GSK Comm” communication software on the PC.
2. Setting of “GSK Comm” communication software:

Click “Connect” menu, and select “Via Network port”, as shown in Figure 11-3-1-1.

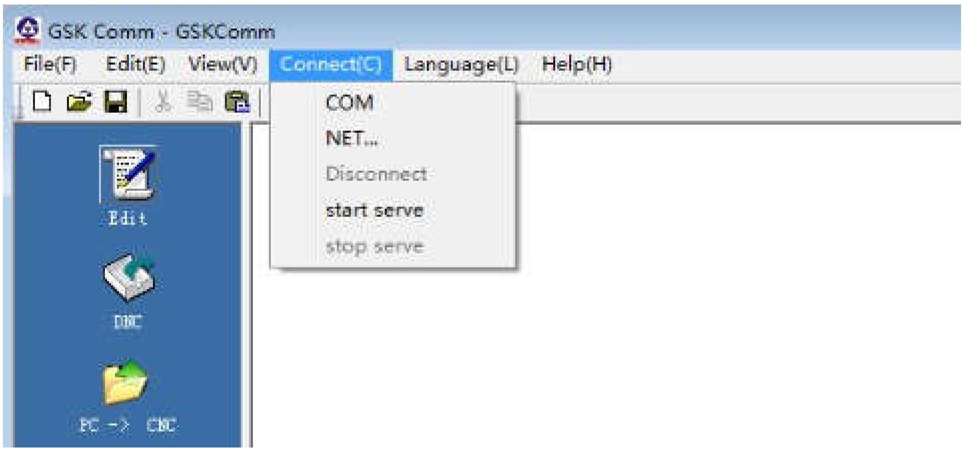


Fig. 11-3-1-1

Then CNC address connection dialog box pops up, where corresponding IP address can be set. As shown in the figure 11-3-1-2.

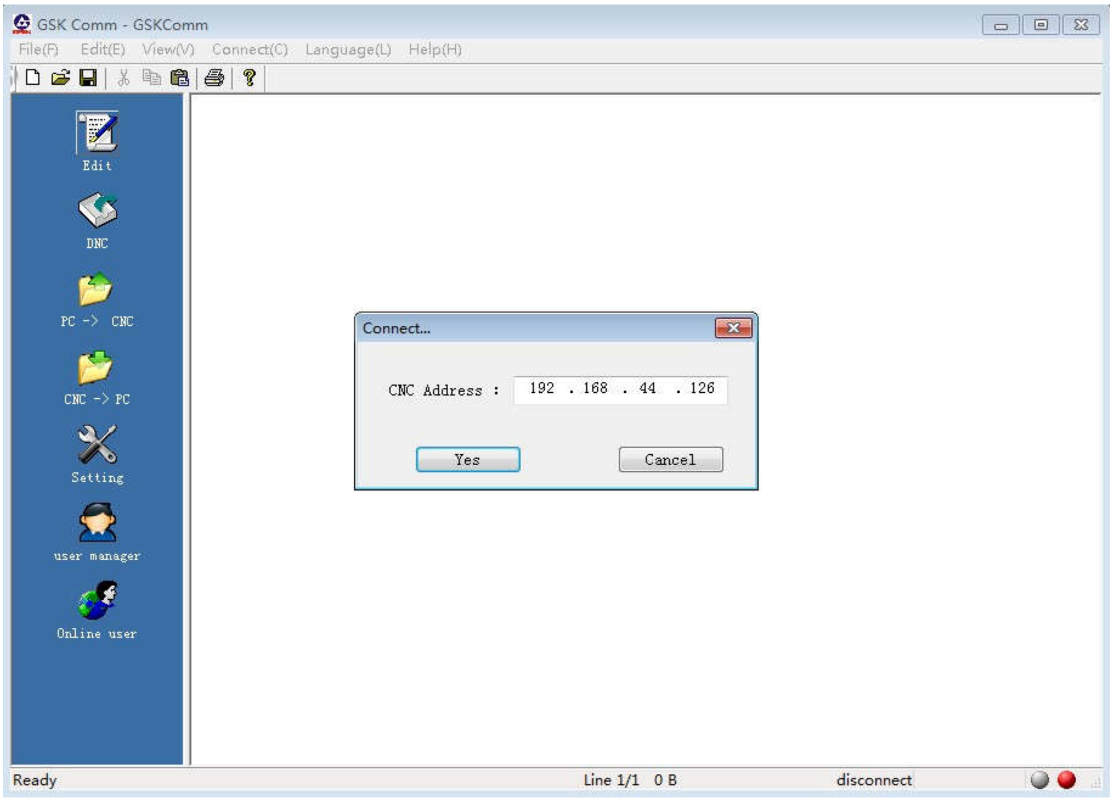
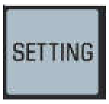


Fig. 11-3-1-2

11.3.2 Date Transmission Via Network Port (Using GSK Comm Communication Software)

Operating steps:

1) Select <Enter> operation mode;

2) Press 

to enter the Set page, and set the I/O channel to 3, as shown in Figure 11-3-2-1.

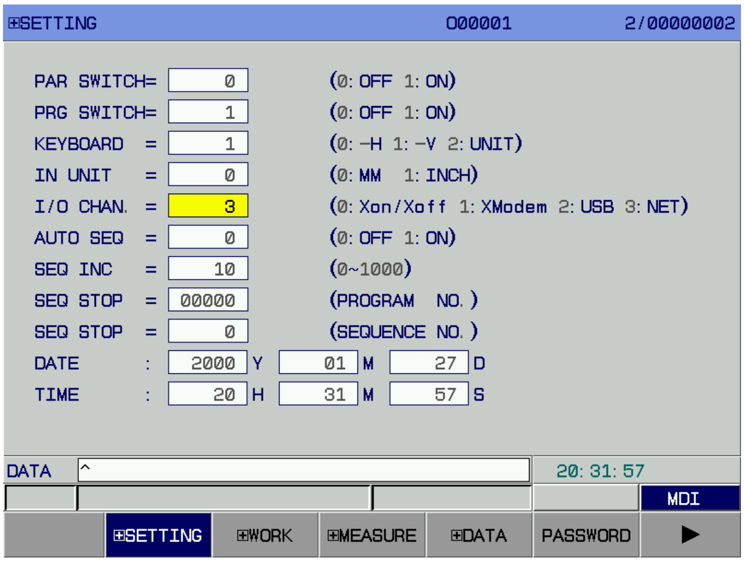
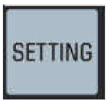
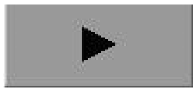


Fig. 11-3-2-1

3) Open the parameter switch, and enter the machine tool manufacturer password. For details, Please refer to "3.4.5 Setting and modification of password permission".

4) Press 

on operation panel to enter the Set interface, press  to enter the system IP setting page, where the IP address is set to the same Network segment address of the PC connected to the CNC. The other settings are the same as PC settings. The "Physical address" can be left unset, as shown in Figure 11-3-2-2.

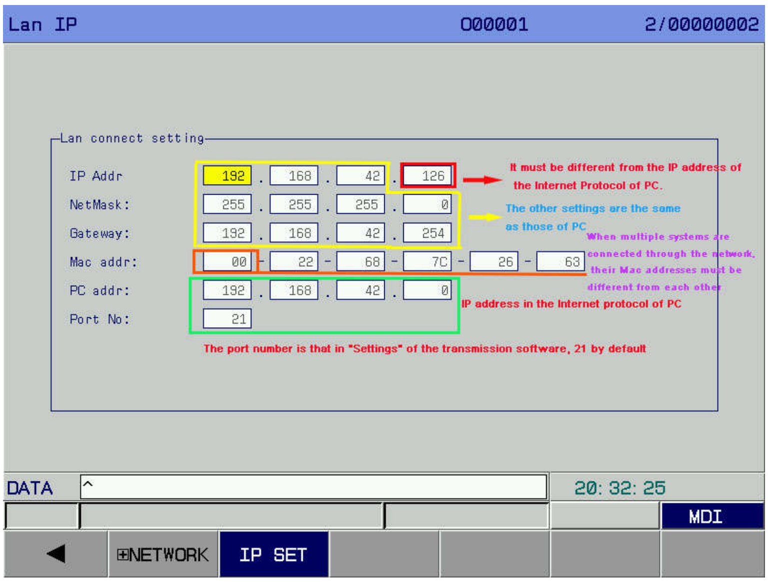


Fig. 11-3-2-2

4) PC settings

1. Click Network Place on the PC to enter the interface, right click on "Local Connection" property, and the Internet protocol appears as shown in Figure 11-3-2-3.

Volume II Operation Instructions

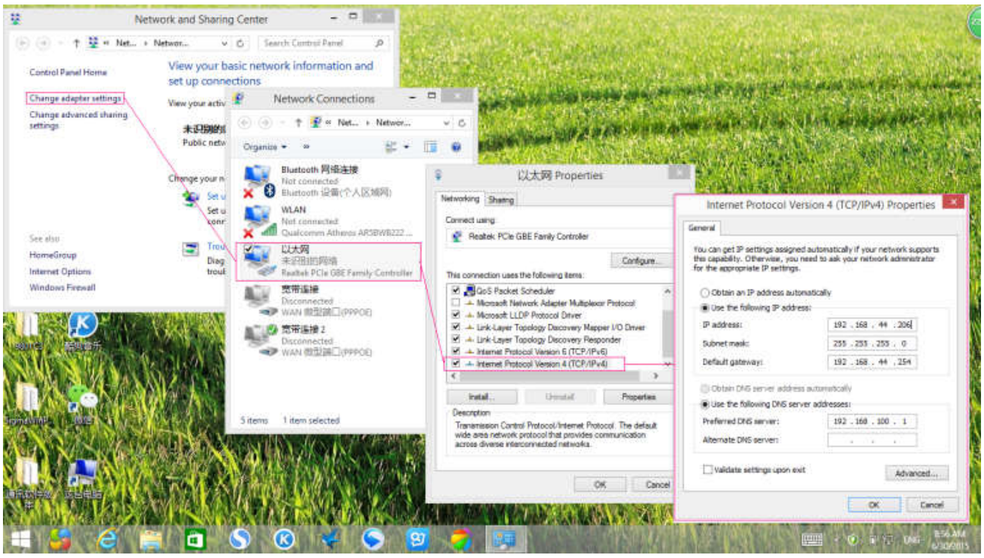


Fig. 11-3-2-3

2. IP address in CNC system cannot be the same as that in the PC. The subnet mask is the same as the default gateway. The IP address of the Network port connection in the communication software needs to be the same as that of the CNC system.
3. For connection via Network port, input the set CNC IP address for connection. Upon successful connection, the lower right corner of the communication software will change to the green ball, and the Ethernet is connected. As shown in the figure 11-3-2-4.

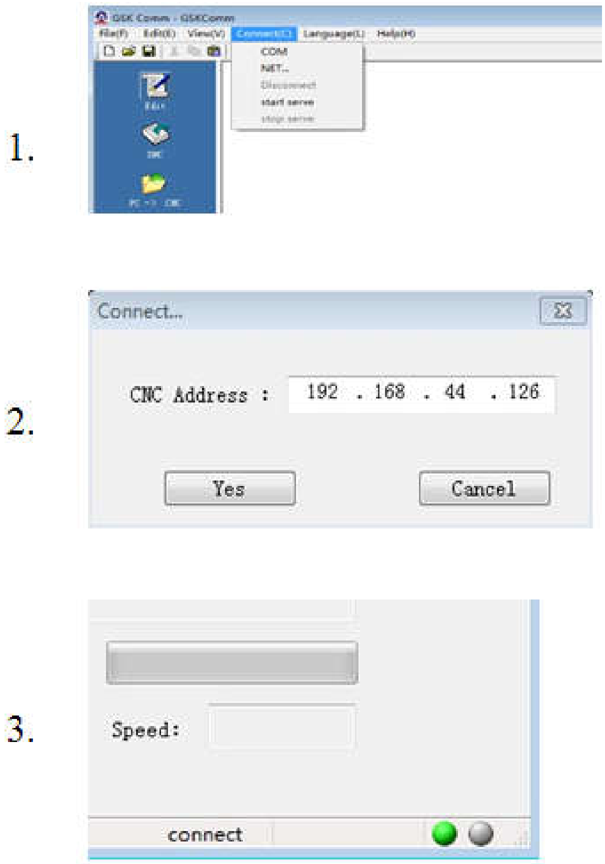


Fig. 11-3-2-4

A. Data output (CNC→PC)

1. Click 

on GSK Comm communication software to enter the Receive File page, as shown in Figure 11-3-2-5.

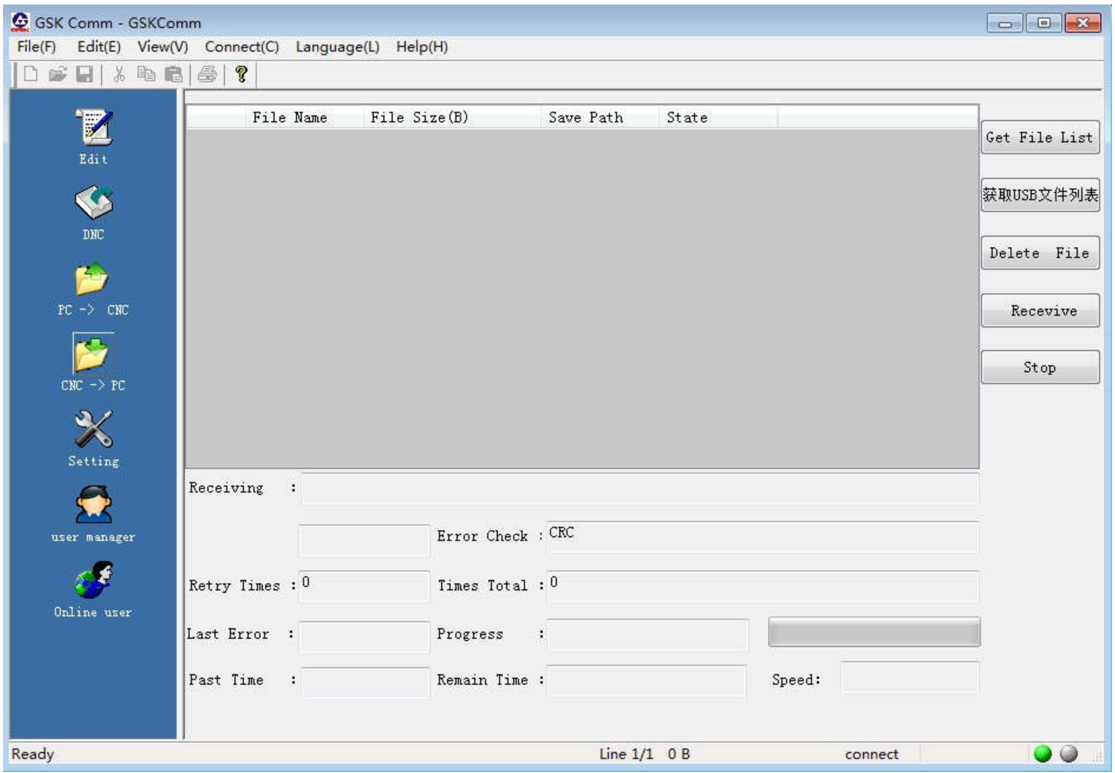


Fig. 11-3-2-5

2. Click Get File List to obtain the file list on CNC, as shown in Figure 11-3-2-6.

	File Name	File Size(B)	Save Path	State
1	007999.TXT	581364	USER	Not receiving
2	007998.TXT	2539	USER	Not receiving
3	007997.TXT	298	USER	Not receiving
4	000001.txt	10	USER	Not receiving
5	001234.txt	1410	USER	Not receiving

Fig. 11-3-2-6

3. Select the file to be received (multiple files can be received at the same time), then press Receive, and a dialog box pops up for the user to select the storage location of the received file, and file receiving begins, as shown in Figure 11-3-2-7.

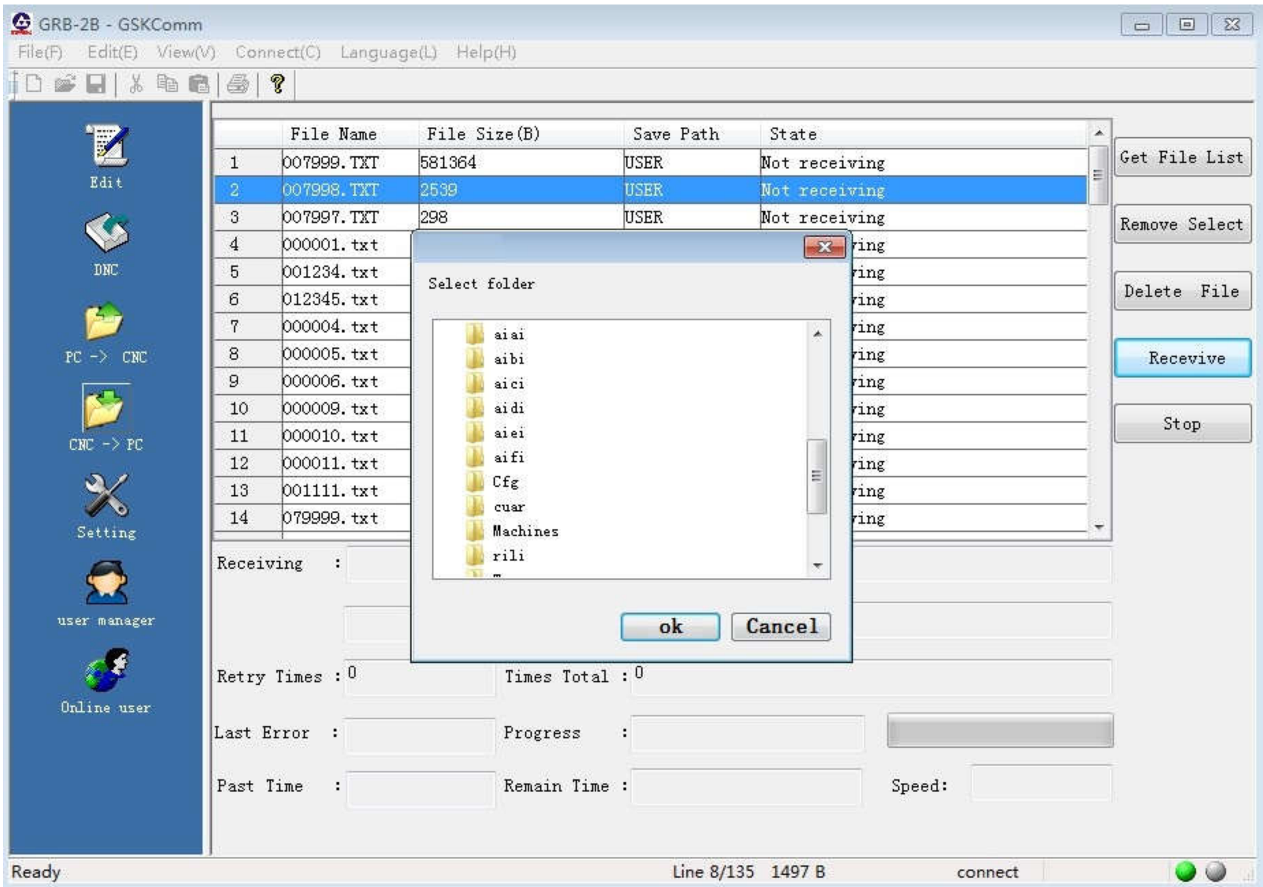


Fig. 11-3-2-7

4. After the file is received, the status bar of the file list item displays “Received”. As shown in the figure 11-3-2-8.

	File Name	File Size(B)	Save Path	State
1	007999.TXT	581364	USER	Not receiving
2	007998.TXT	2539	USER	Received
3	007997.TXT	298	USER	Not receiving
4	000001.txt	10	USER	Not receiving
5	001234.txt	1410	USER	Not receiving

Fig. 11-3-2-8

B. Data input (PC→CNC)

1. Click 

on GSK Comm communication software to enter the Send File page. As shown in the figure 11-3-2-9.

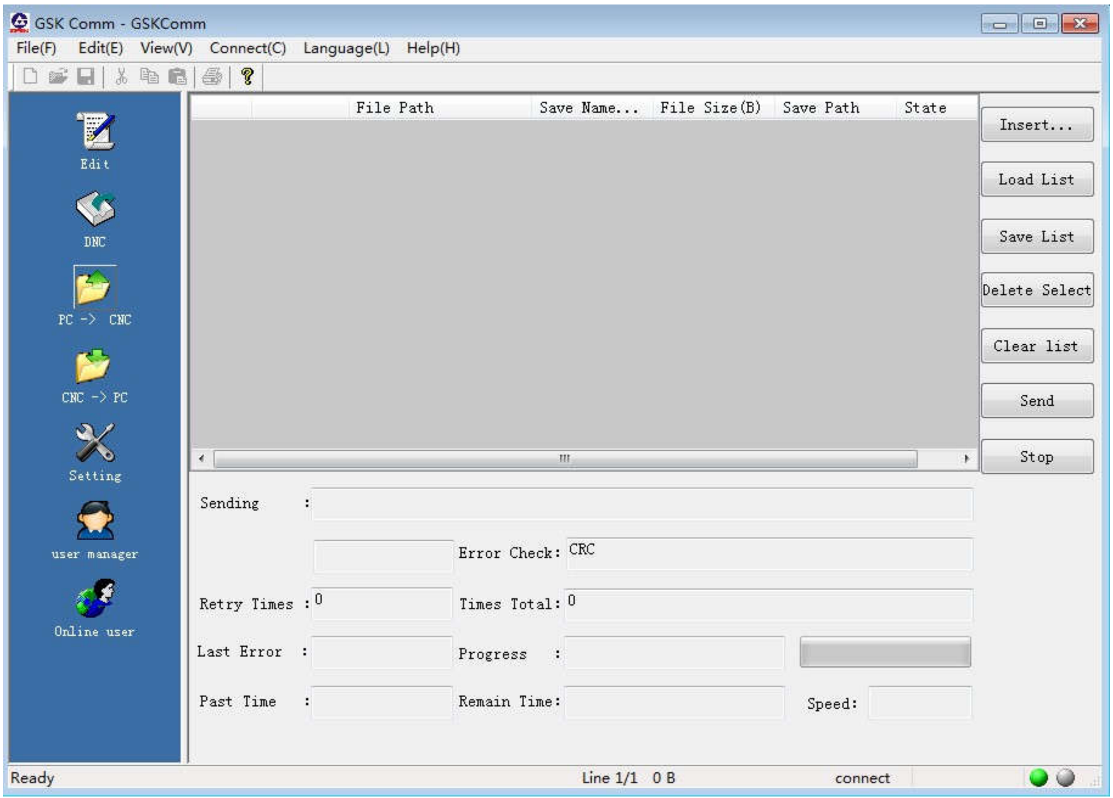


Fig. 11-3-2-9

2. Click **Add...** to add all files to be sent to CNC. As shown in the figure 11-3-2-10.

		File Path	Save Name...	File Size(B)	Save Path	State
1	√	D:\NC\002223.TXT	002223.txt	272060	USER	unsend
2	√	D:\NC\000015.TXT	000015.txt	392460	USER	unsend
3	√	D:\NC\000008.TXT	000008.txt	14103	USER	unsend
4	√	D:\NC\000007.TXT	000007.txt	588267	USER	unsend
5	√	D:\NC\000016.TXT	000016.txt	13958	USER	unsend

Fig. 11-3-2-10

3. Click **Send** , and file sending starts. As shown in the figure 11-3-2-11.

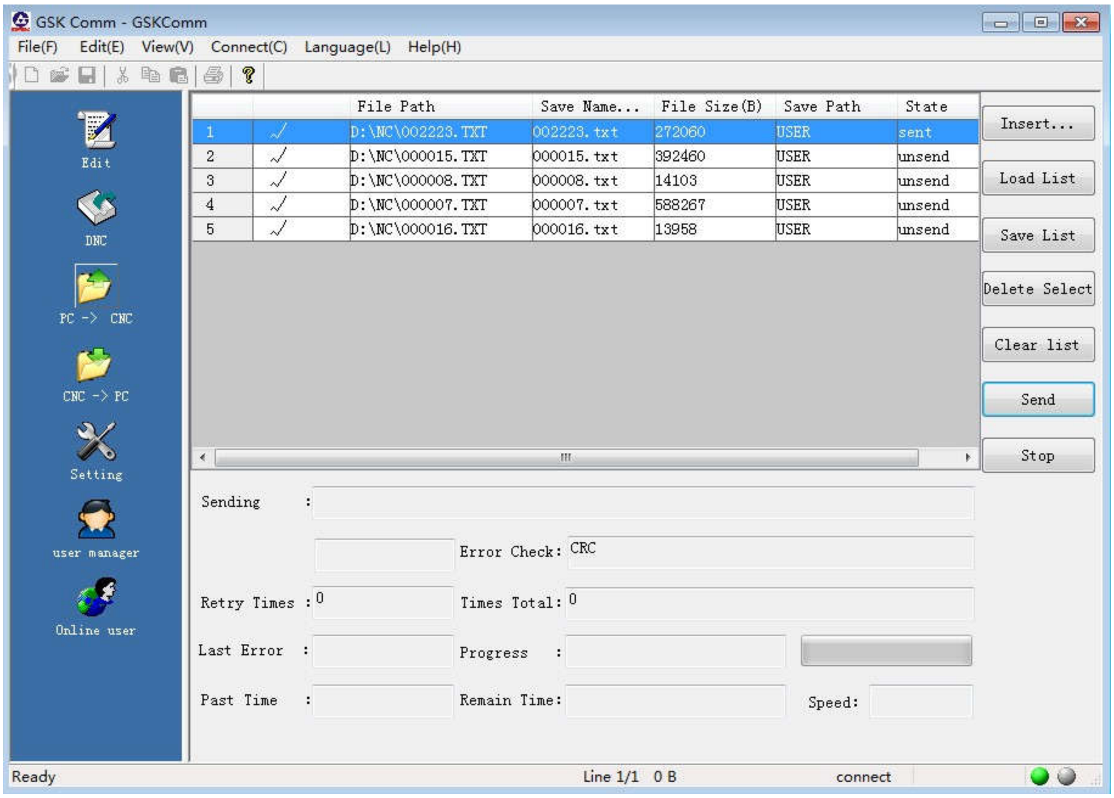


Fig. 11-3-2-11

4. After the file is sent, the status bar of the dialog box displays “Sent”. As shown in the figure 11-3-2-12.

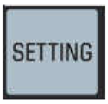
		File Path	Save Name...	File Size(B)	Save Path	State
1	√	D:\NC\002223.TXT	002223.txt	272060	USER	sent
2	√	D:\NC\000015.TXT	000015.txt	392460	USER	unsend
3	√	D:\NC\000008.TXT	000008.txt	14103	USER	unsend
4	√	D:\NC\000007.TXT	000007.txt	588267	USER	unsend
5	√	D:\NC\000016.TXT	000016.txt	13958	USER	unsend

Fig. 11-3-2-12

Note: Upload, download, or delete the program of the system after transmission. Other files such as: ladder diagram, parameter, pitch offset, etc., are handled according to RS232 operation steps.

C. CNC online processing

Operation method:

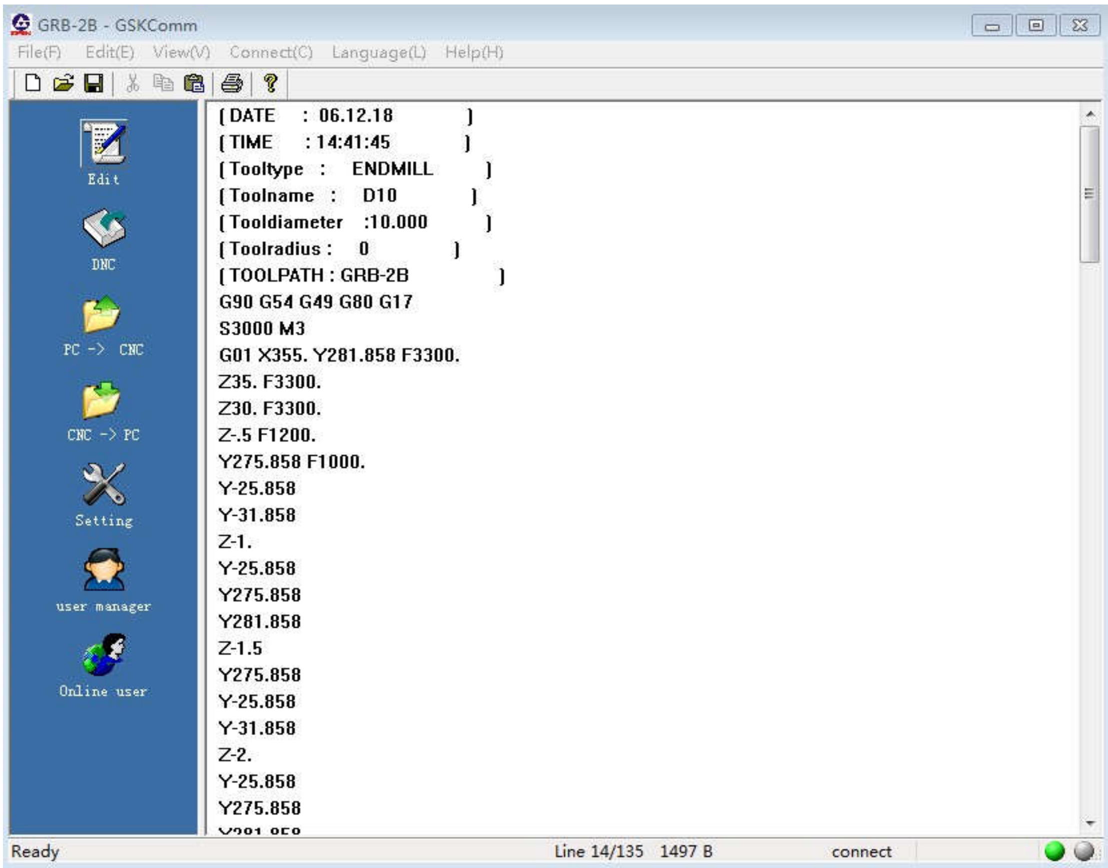
1. Press 

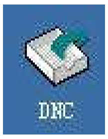
in CNC to enter the Set page, and set the I/O channel to 3.

2. The CNC needs to be in the DNC mode for transmission.

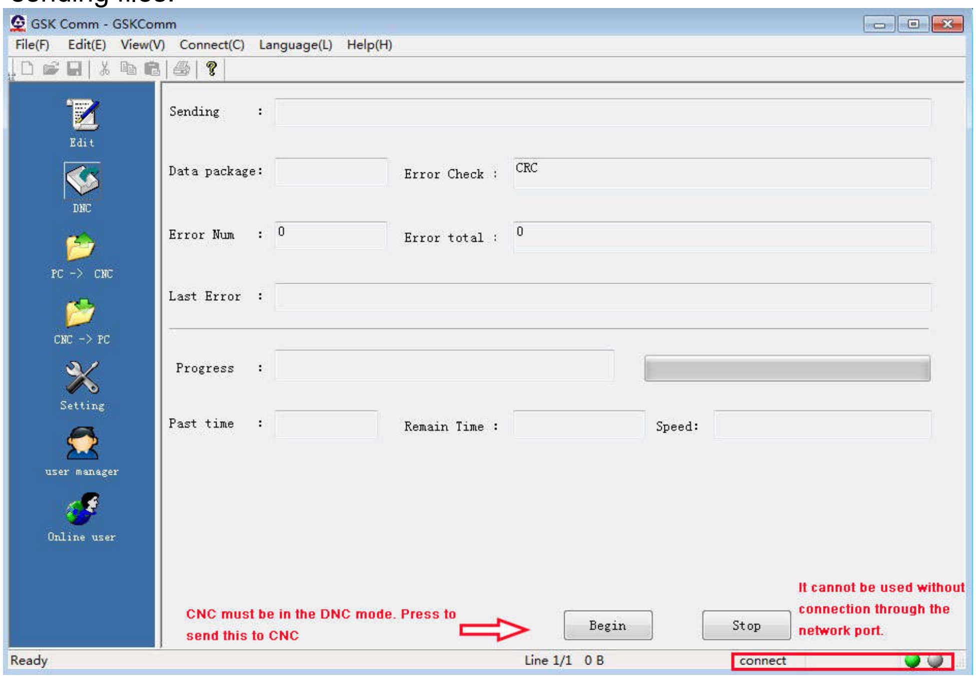
3. Click 

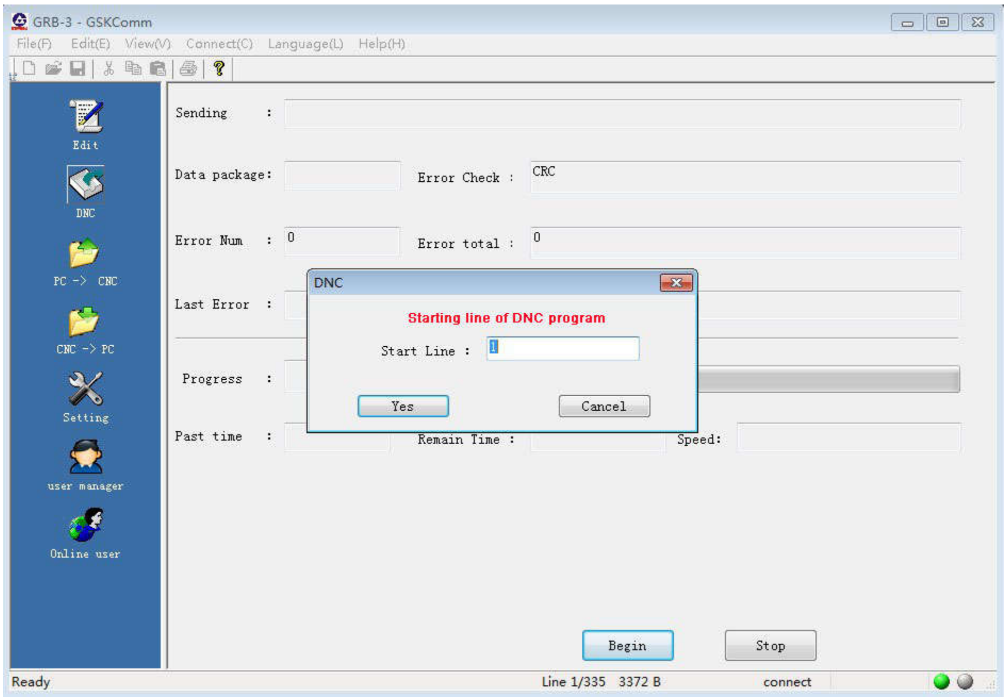
on the main menu on GSK Comm communication software to enter the file editing interface and open the required program.

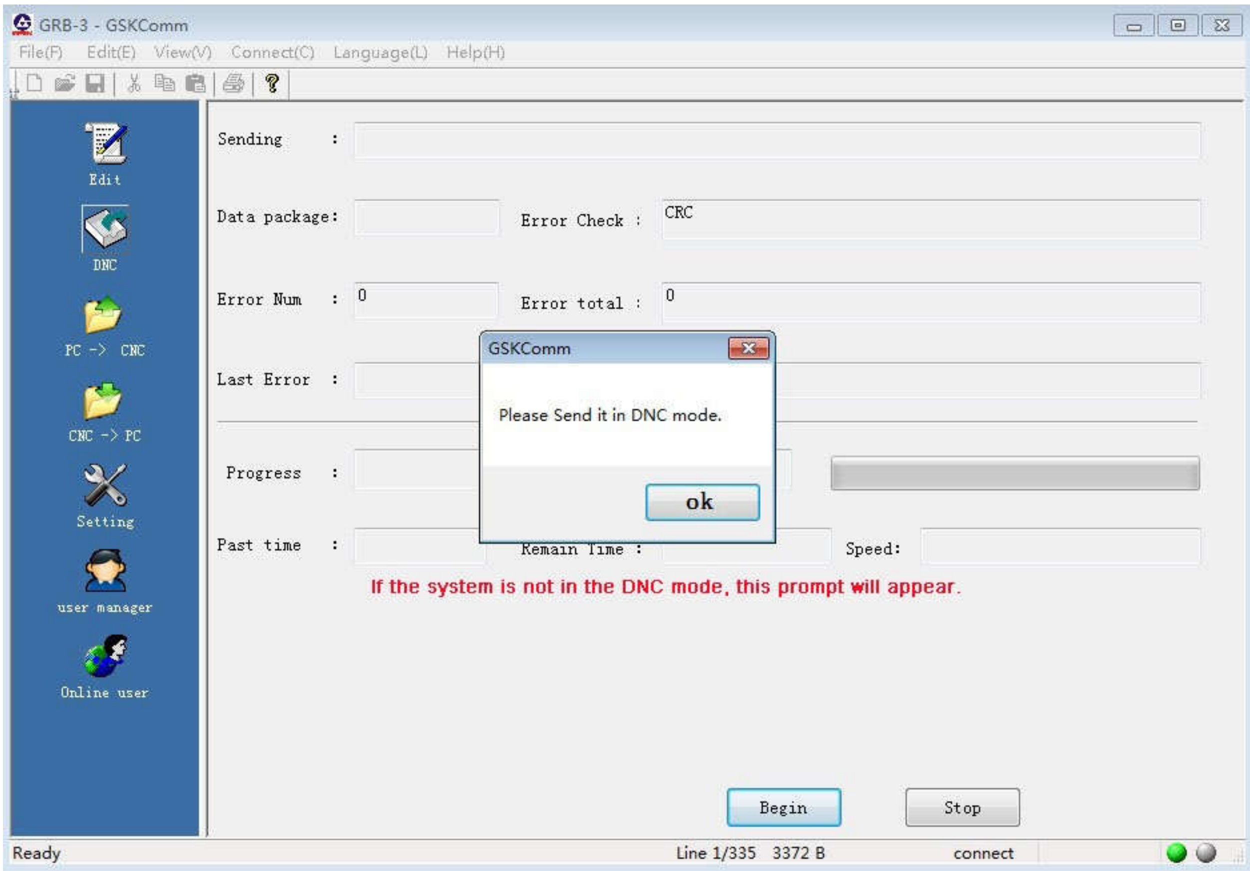


4. Click 

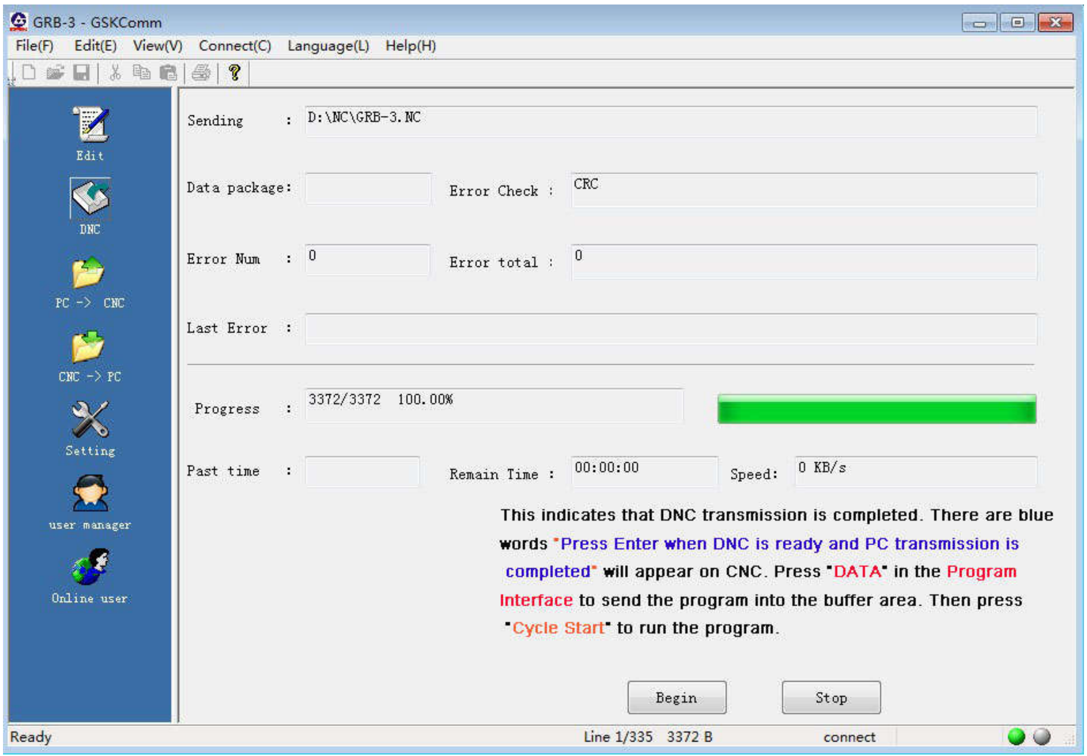
on the main menu on GSK Comm communication software to enter the sending interface in DNC. The CNC system must be in DNC mode. Press “Start” to start sending files.







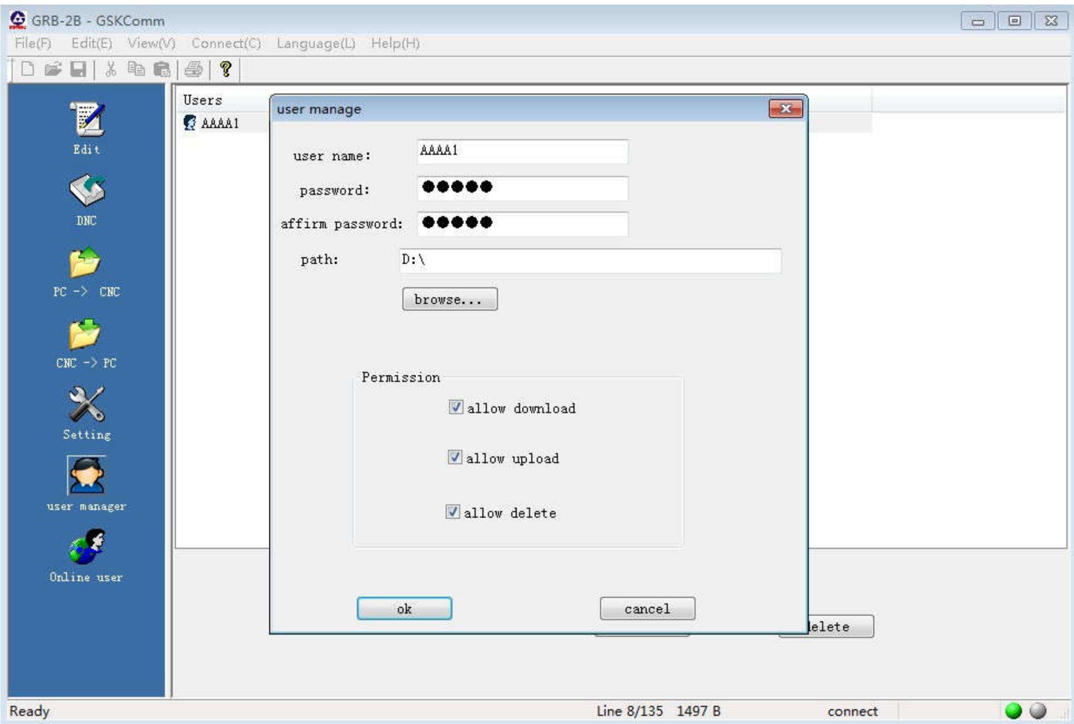
5. After sending is completed, the blue words “DNC is ready, please press ‘Enter” after PC sending” show on CNC.



11.3.3 Data Transmission Via Network (Using 218MC Network Transmission Interface)

1. Open *"GSK Comm" communication software* on the PC.
2. Setting of "GSK Comm" communication software:

1) After opening "User Management", press "Add" to enter the "User Management" setting interface, as shown in Figure 11-3-3-1.



Note: [User Name] and [Password] contain English letters , which must be capitalized.

Fig. 11-3-3-1

2) Click “Connect” menu and select “Enable Server”. Upon successful connection, the lower right corner of the communication software will change to the green ball, and the server is enabled, as shown in 11-3-3-2.

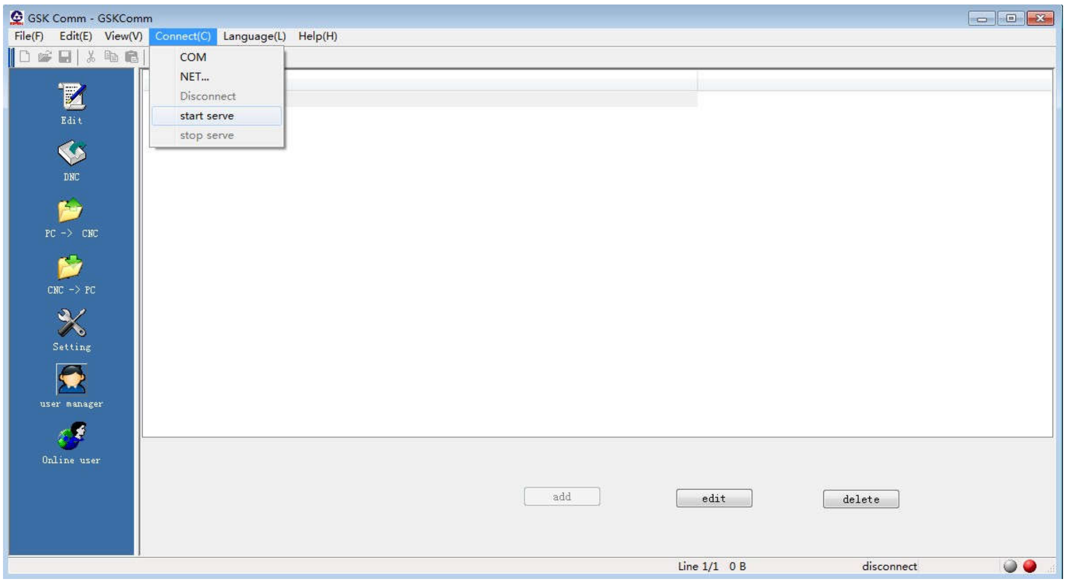
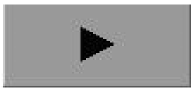


Fig. 11-3-3-2

3. CNC operation steps:

Press 

on the operation panel to enter the Set display interface; press soft key  to enter the Network transmission page. The display interface contains user name, password and login; move the cursor left and right to enter the user name and password; press “Enter” to log in. To exit, press “Enter” again to exit (see Figure 11-3-4-4).

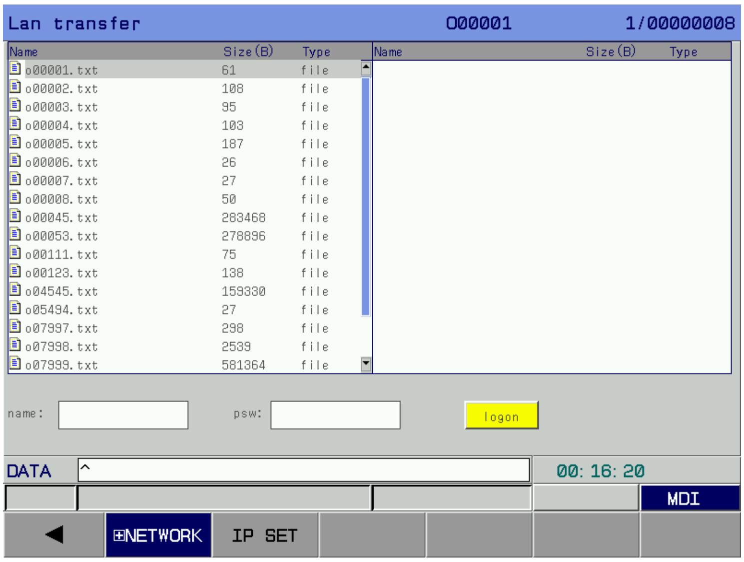


Fig. 11-3-4-4

Note: CNC files are displayed on the left, and PC files on the right.

4. Press the soft key [Network Transmission] for downloading and uploading of CNC and PC files, and DNC processing.

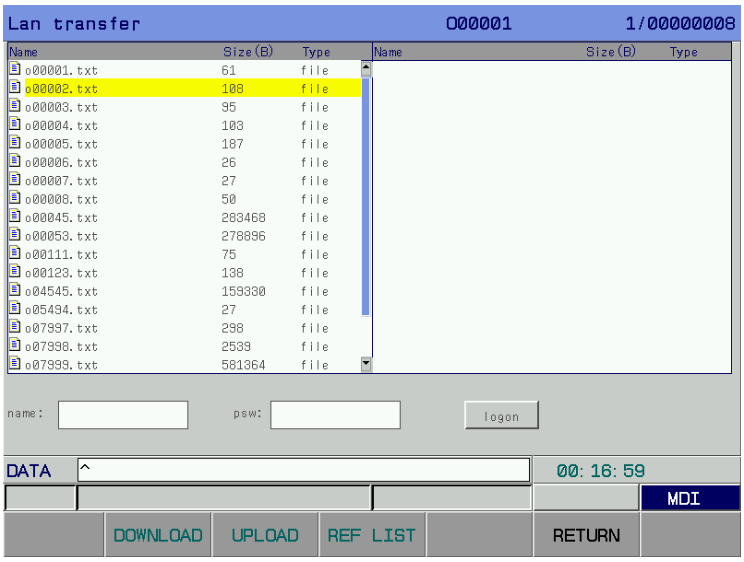
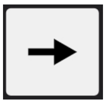
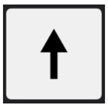
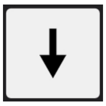


Fig. 11-3-4-5

A、 PC--CNC File Download

a) Press the arrow key  to switch the cursor to the file catalog.

b) Press the arrow key  or  to move the cursor and select the CNC program file in PC disk.

c) Press the soft key [Download], and the system prompts “Transferring.....”.

d) If the name of the downloaded CNC program is duplicated, the system will prompt “Do you want to replace the original file” and press <INPUT> to overwrite the original CNC program file.

If you need to rename the CNC program file, press <Cancel>, type the new program number (such as O10, O100), and then press <INPUT> to download the CNC program file, as shown in Figure 11-3-4-6.

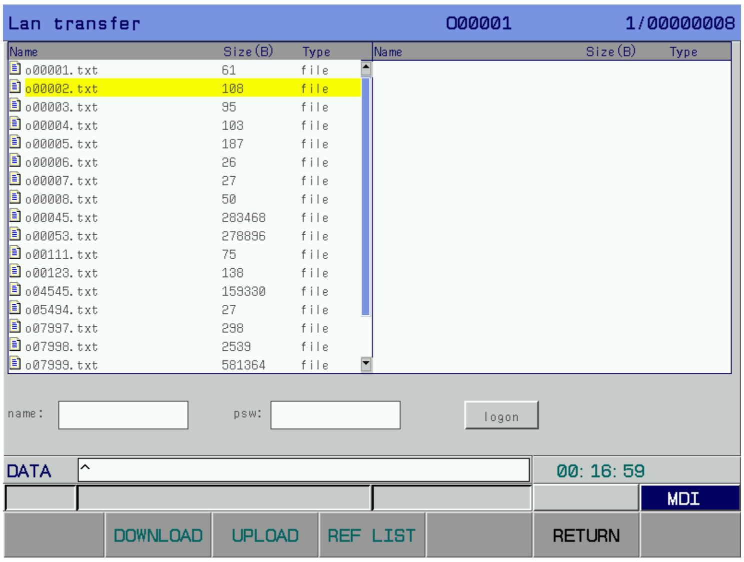
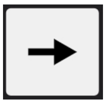
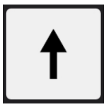
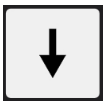


Fig. 11-3-4-6

B、 CNC-PC File Upload

a) Press the arrow key  to switch the cursor to the file catalog.

b) Press the arrow key  or  to move the cursor and select the CNC program file in PC disk.

c) Press the soft key [Upload], and the system prompts “Transferring.....”.

d) If the name of the uploaded CNC program is duplicated, the system will prompt “Do you want to replace the original file” and press <INPUT> to overwrite the original CNC program file.

If you need to rename the CNC program file, press <Cancel>, type the new program number (such as O10, O100), and then press <INPUT> to download the CNC program file, as shown in Figure 11-3-4-7.

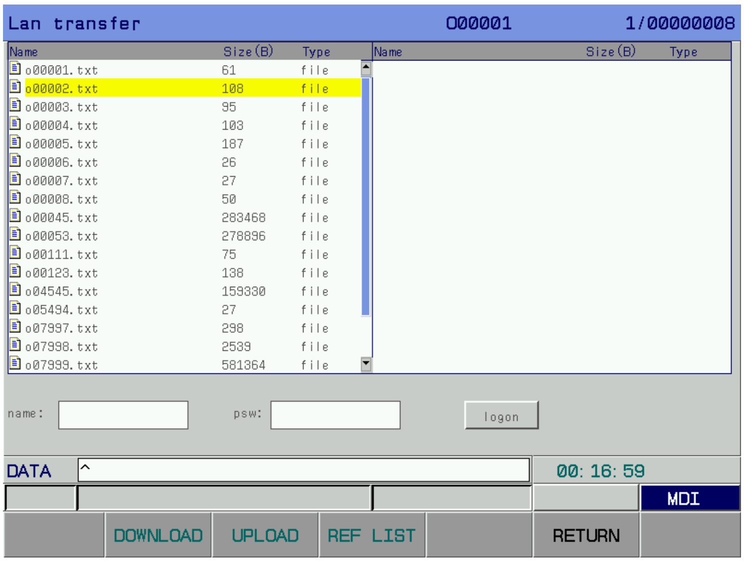
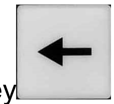
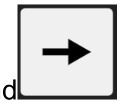
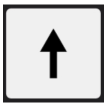
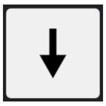
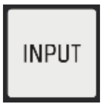


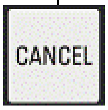
Fig. 11-3-4-7

Volume II Operation Instructions

e) To delete the file in the PC window list, press the arrow key  and  to switch the cursor to the file catalog.

f) Press the arrow key  or  to move the cursor to select the CNC program file on PC side; press 

, and the system will prompt “Are you sure you want to delete that file?” "Press 

to delete it; press 

to exit.

Note 1: If the folder list on PC side is updated, and the PC file window cannot update the file list under the folder on PC side in real time, press “Refresh List” to update the file list on PC side in real time.

Note 2: The Network transmission is enabled through the connection of several computers by the router, but their operation is not affected.

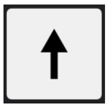
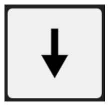
11.4 USB Communication

11.4.1 Overview and Precautions

Notice:

1. Set the I/O channel to 2 on <**Set**> screen.
2. The CNC program must be named in suffix .txt, .nc or .CNC, and stored in the root directory of the U disk, otherwise the system can not read.
3. Do not remove USB flash drive during USB transmission communication, so as to avoid product failure or unexpected consequences.
4. After the U disk communication operation is completed, remove it after the U disk indicator stops flashing (or wait for a little while) to ensure data transmission is completed.

11.4.2 Usb Part Program Operation Steps

After entering [Data] screen under <Input Mode>, press the arrow key  or  to move the cursor to “CNC part program”. Press the soft key [Data Export] or [Data Input] to enter the following operation interface (as shown in Figure 11-4-2-1).

Volume II Operation Instructions

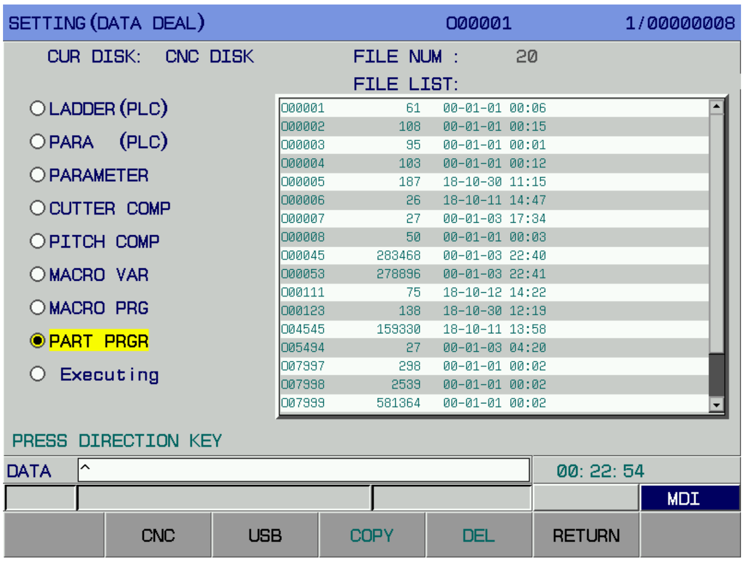
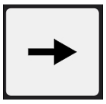
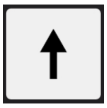
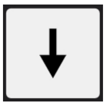


Fig. 11-4-2-1

1. Copy the CNC program file from the system disk to the U disk:

a) Press the arrow key  to switch the cursor to the file catalog.

b) Press the arrow key  or  to move the cursor and select the CNC program file to be copied in the system disk.

c) Press the soft key [Copy], and the system prompts “Copy to U disk? New file name”, as shown below (Figure 11-4-2-2).

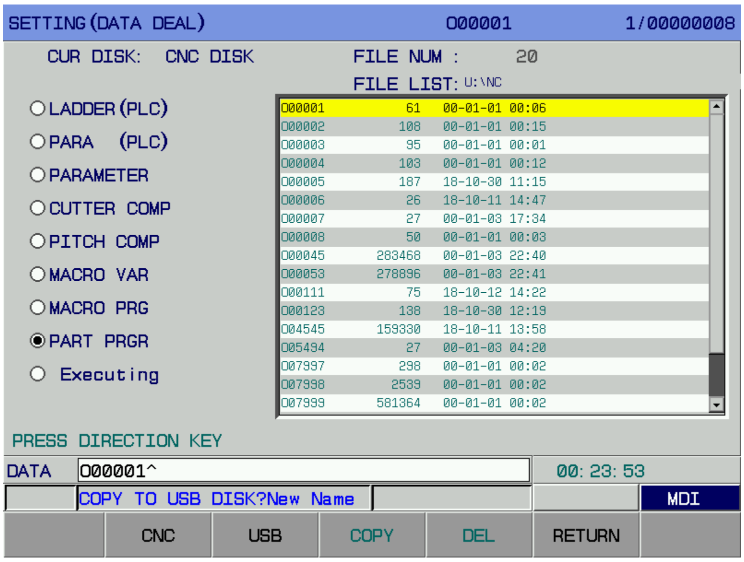


Fig. 11-4-2-2

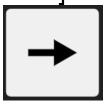
d) If you do not need to rename the CNC program file, press <INPUT> to copy the CNC program file directly.

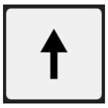
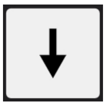
If you need to rename the CNC program file, press <CAN>, type the new program number (such as O10, O100), and then press <INPUT> to copy the CNC program file.

If a program file with the same name exists on the U disk, the system will prompt “Copy to U disk? Please rename it”. Please type the new program number (such as O10, O100), and then press <INPUT> to copy the CNC program file.

2. Copy the CNC program file from the U disk to the system disk:

a) Press the soft key [U disk] to switch to the file catalog display interface on the U disk.

b) Press the arrow key  to switch the cursor to the file catalog.

c) Press the arrow key  or  to move the cursor and select the CNC program file to be copied in the U disk.

Press the soft key [Copy], and the system prompts "Copy to the system disk? New file name, as shown below (Figure 11-4-2-3).

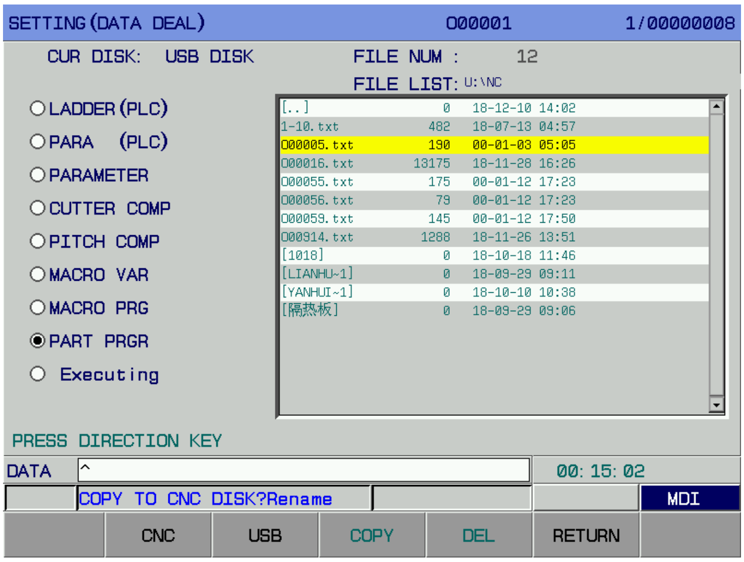


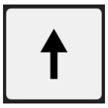
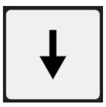
Fig. 11-3-2-3

d) If you do not need to rename the CNC program file, press <INPUT> to copy the CNC program file directly. If you need to rename the CNC program file, press <CAN>, type the new program number (such as O10, O100), and then press <INPUT> to copy the CNC program file.

If a program file with the same name exists on the system disk, the system will prompt "Copy to system disk? Please rename it". Please type the new program number (such as O10, O100), and then press <INPUT> to copy the CNC program file.

Note: The LADCHI**.TXT file is invalid after transferred to the system, and will become valid after powering off.

3. Delete files from the system disk/U disk:

a) Press the arrow key  or  to move the cursor and select the CNC program file to be deleted in the system disk/U disk.

b) Press the soft key [DEL], and a prompt will be shown at the bottom of the interface: "Are you sure to delete the current file?" Press <CAN> to cancel the file deletion; press <INPUT> to delete the file.

11.4.3 Exit The U Disk Operation Interface

Pull out the U disk when the indicator light of the U disk stops flashing.
Press the soft key [Back] to return to [System (Data Processing)] interface.

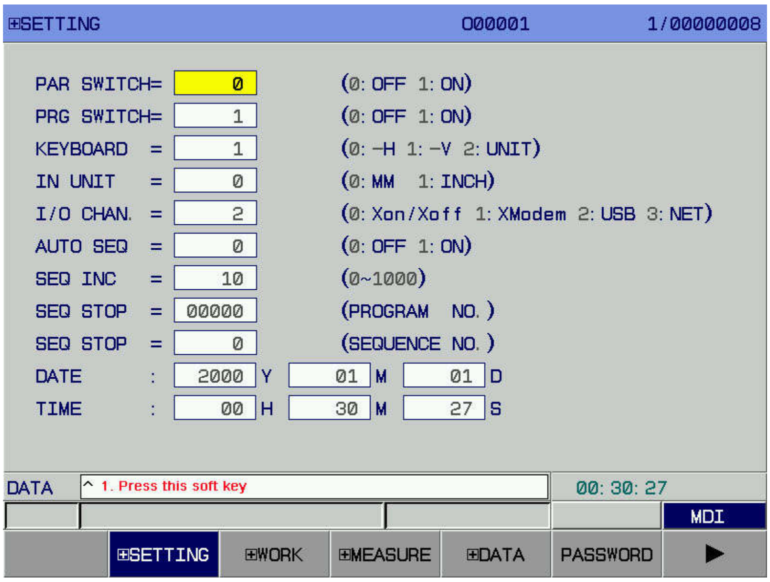
Chapter XII Time-limited Shutdown Function

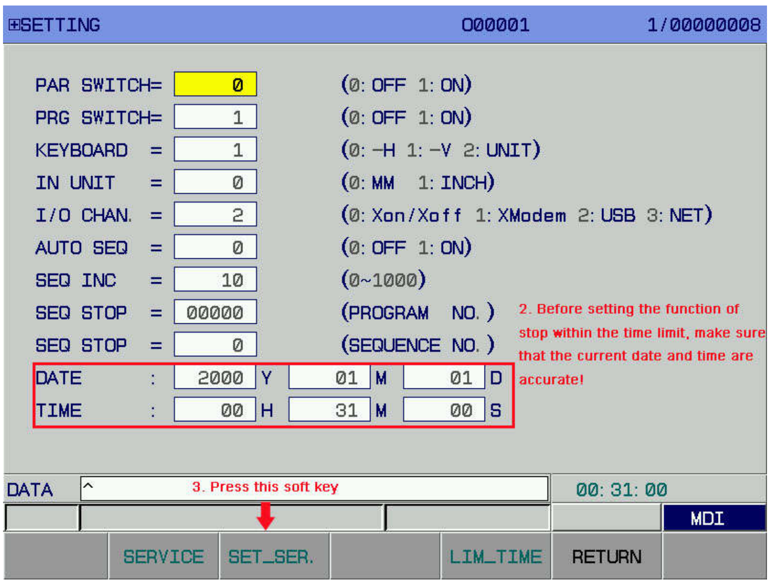
The time-limited shutdown function of GSK 218MC V1.5 version can only be enabled under the system machine manufacturer or higher permission (initial password: ADMIN or 111111).

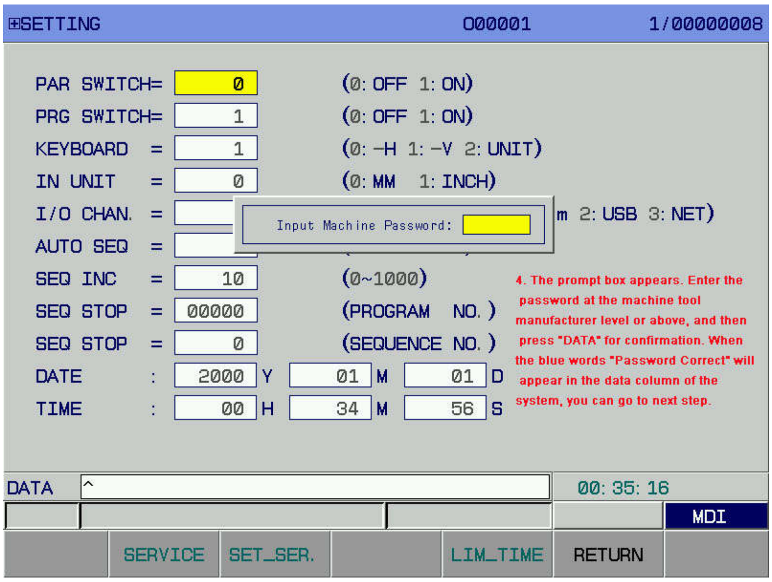
12.1 Set Time-Limited Shutdown Duration

12.1.1 Steps To Set Time-Limited Shutdown In The System

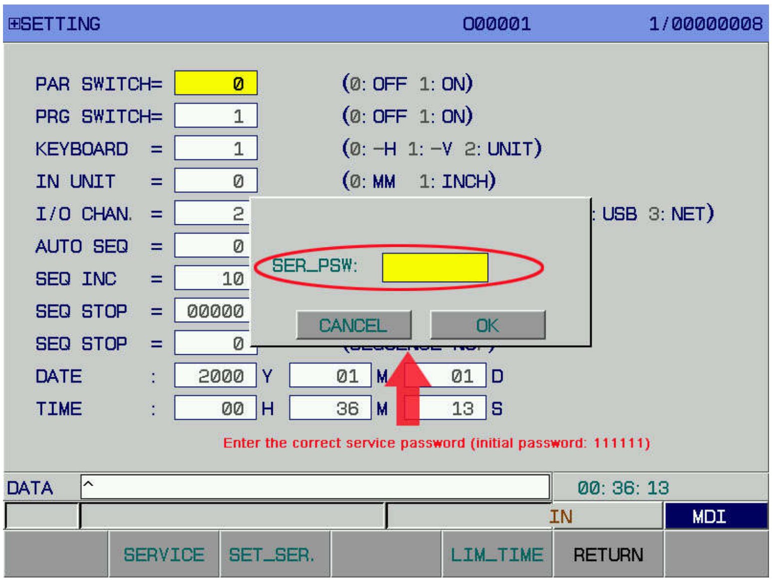
1. Press **[Set]** on the operation panel, and then press the soft key **[田 Set]** to enter. The operation steps are as shown below.





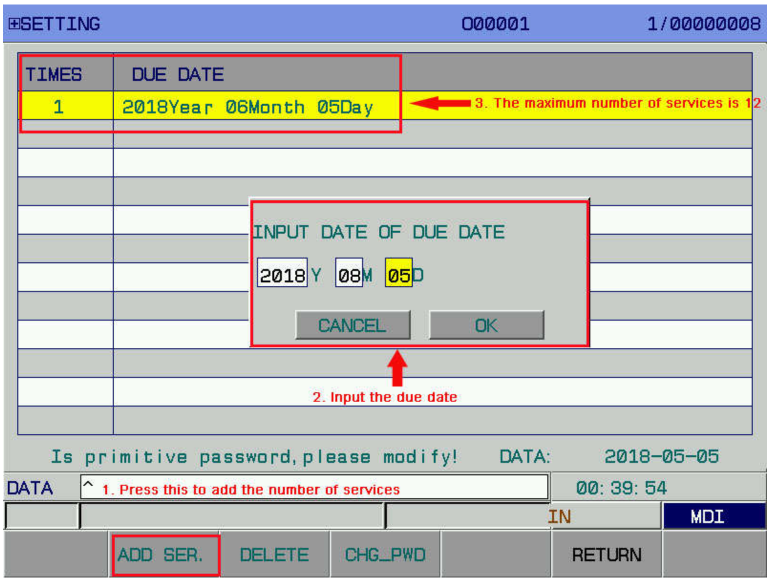


2. Press the soft key [Set Shutdown], and enter the shutdown password (initial password: 111111), as shown below.

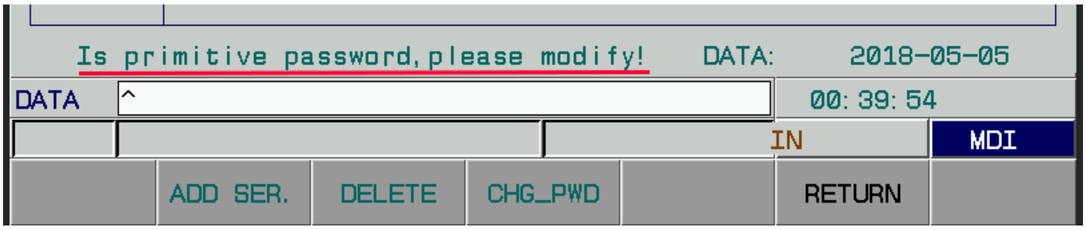


3. Press the soft key [New Phase], and enter the shutdown date as shown below.

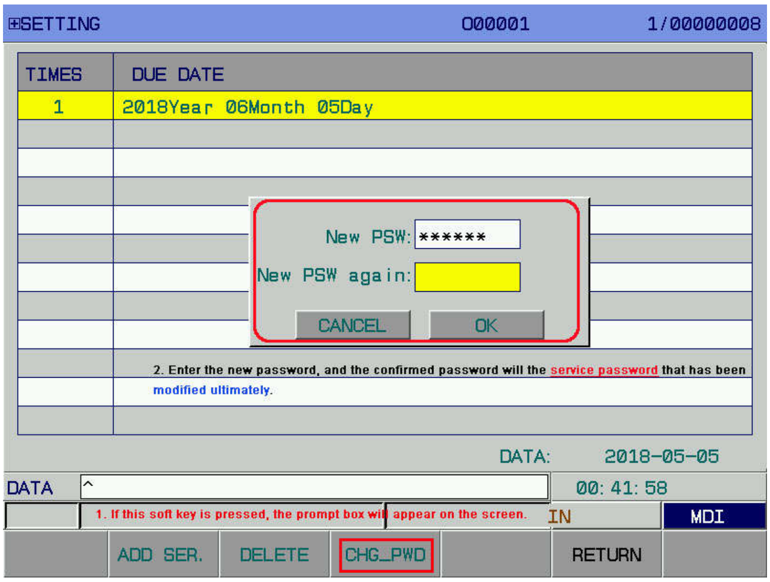
Volume II Operation Instructions



4. If the shutdown password is the initial password, the system will prompt “The current shutdown password is the initial password. Please change it in time!” at the top of the input field. ”.

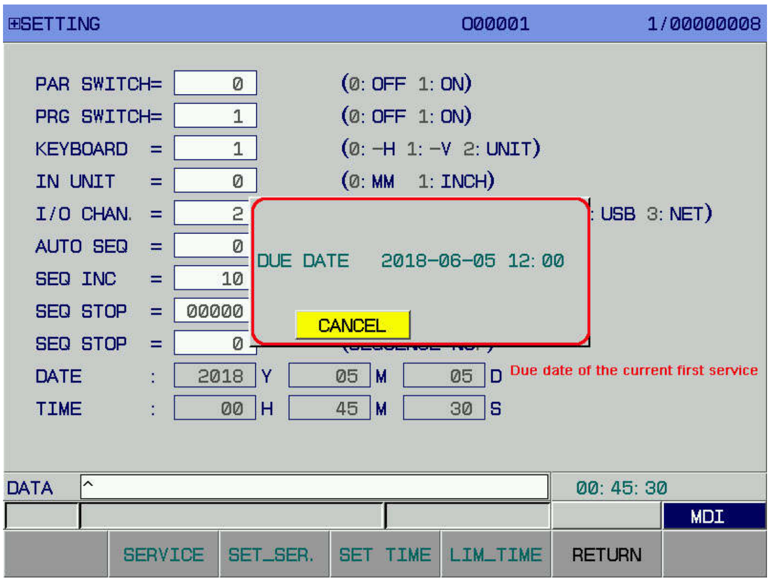


Press [Change Password], enter the new password, and confirm, as shown below.



Note: Remember the last changed shutdown password! You can only replace the mainboard when you forget this shutdown password!

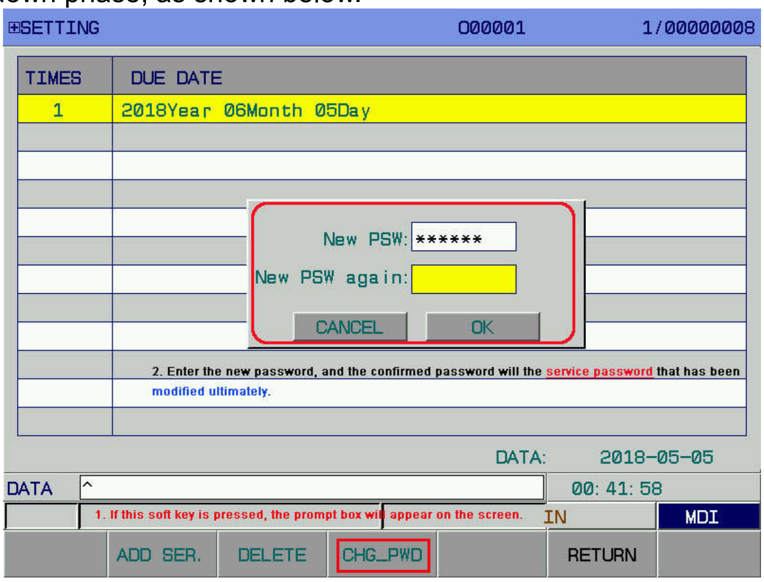
5. When the shutdown phase and shutdown password are set, press [Back] to return to the Set interface, press [田 Set], and then press [Shutdown Time] to view the shutdown date at first phase, as shown below.



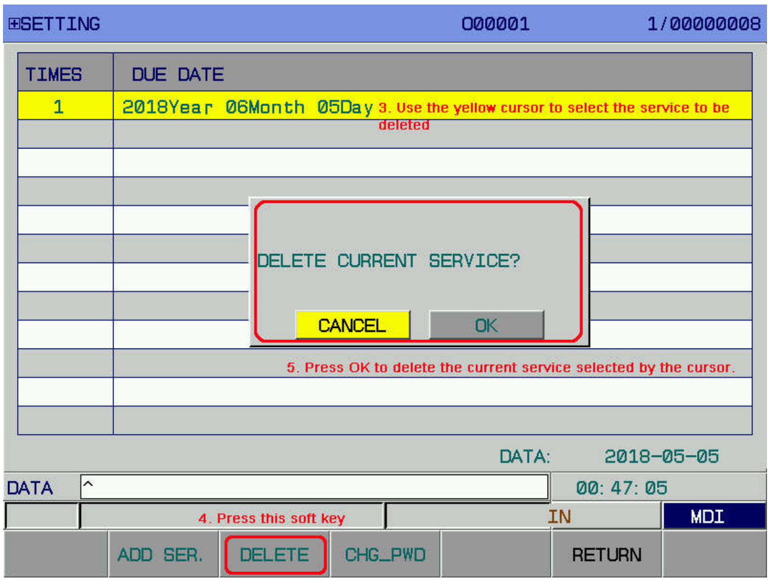
12.2 Cancel Time-Limited Shutdown Time

12.2.1 Cancel Time-Limited Shutdown Time In The System

Delete the shutdown time directly, enter [Set Shutdown] interface, input the shutdown password and delete the shutdown phase, as shown below.



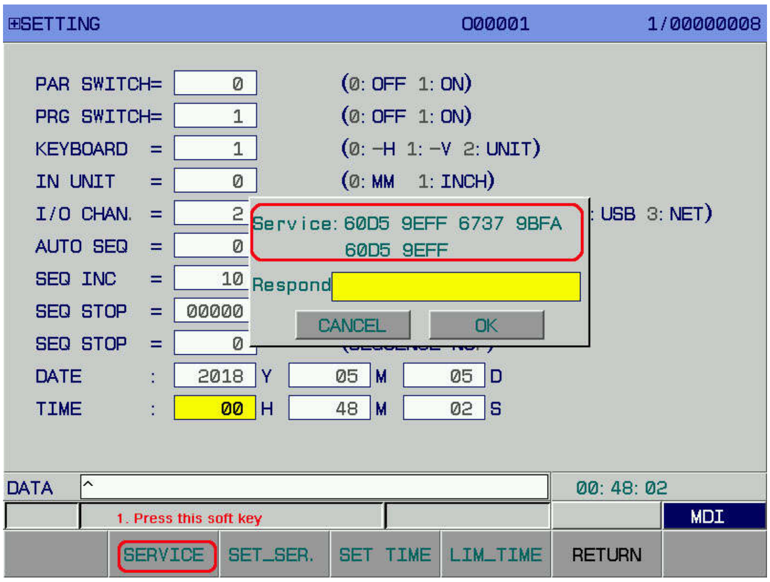
Volume II Operation Instructions



12.2.2 Cancel Time-Limited Shutdown Time Via PC Software

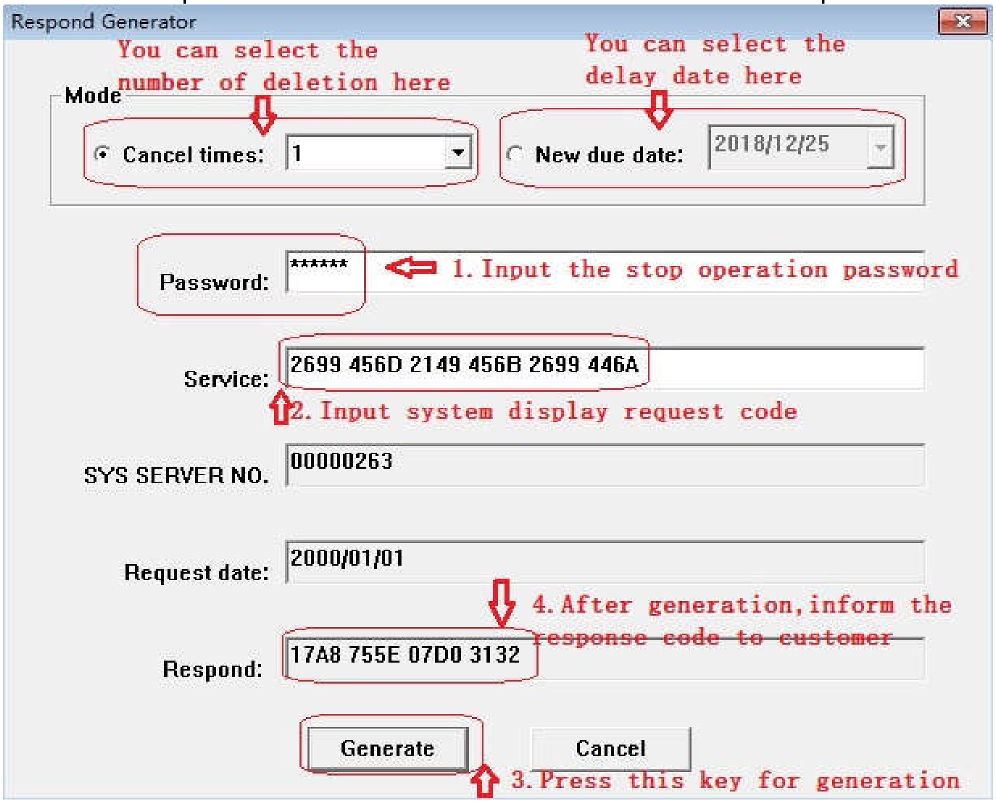
This PC software can be used to cancel time-limited shutdown when the commissioning staff is not at the customer site.

1. When the shutdown time has been set, the system can generate the request code, as shown in the following figure:



Volume II Operation Instructions

2. Run the “Response Code Generator” software on PC side. The operation is as follows:



3. Enter the “System response code" generated in the “Response Code Generator” software into “Response Code”, and confirm. Then the CNC system will prompt “Response code is valid” at the bottom of the input field, indicating that the corresponding shutdown time is successfully cancelled. If the CNC system prompts “Response code is invalid” at the bottom of the input field, the shutdown time has not been cancelled. Please check if the entered shutdown password, request code, and cancelling code is incorrect?
4. To delay the time of the time-limited shutdown, you can modify the time-limited shutdown time in the CNC system, or use the extension function in the “Response Code Generator” software. The operation steps for shutdown input are the same as response code.

APPENDIX

Appendix I Parameters of GSK218MC Series

Parameter Description

Based on the data type, the parameters can be classified into the following categories:

2 data types and valid value range

Data type	Valid data range	Remarks
Bit type	0 or 1	System default values; the user can modify it as needed
Data type	Determined according to the parameter range	System default range and default value; the user can modify it as needed

1. For bit-type parameters, each data consists of 8 bits. Each bit has a different meaning.
2. In the above table, the value range of each data type is a general valid range, and the specific parameter value range is actually not the same. Please refer to the detailed description of each parameter.

[Example]

(1) Meaning of a bit-type parameter

Data No.	BIT7	BIT6	BIT5	BIT4	BIT3	BIT2	BIT1	BIT0

(2) Meaning of a data-type parameter

0 2 1	
Data No.	Data

Note 1: The space bits in the parameter description and the parameter numbers displayed on the screen but not listed in the parameter table are reserved for future extension and must be set to 0.

Note 2: If no meaning is assigned to parameters 0 and 1, 1 represents positive, and 0 represents negative.

Note 3: If INI is set to 0, when inputted in metric system, the unit of linear axis is mm and mm/min, and the basic unit of the rotating is deg and deg/min for parameter setting.

If INI is set to 1, when inputted in inch system, the unit of linear axis is inch and inch/min, and the basic unit of the rotating is deg and deg/min for parameter setting.

1-Bit Parameter

System parameter number

0 0 0	MODE	SVCD	SEQ	MSP		INI	INM	PBUS

PBUS =1: The drive unit transmission mode is the bus type.

=0: The drive unit transmission mode is pulse type.

INM =1: The minimum movement unit of the linear axis is in inch system

=0: The minimum movement unit of the linear axis is in metric system

If **INM** is set to 0, when outputted in metric system: the basic unit of the linear axis is mm and mm/min, and that of the rotating is deg and deg/min.

If **INM** is set to 1, when outputted in inch system: the basic unit of the linear axis is inch and inch/min, and that of the rotating is deg and deg/min.

INI =1: Input in inch system.

=0: Metric system input.

If **INI** is set to 0, when inputted in metric system: the basic unit of the linear axis is mm and mm/min, and that of the rotating is deg and deg/min.
If **INI** is set to 1, when inputted in inch system: the basic unit of the linear axis is inch and inch/min, and that of the rotating is deg and deg/min.

MSP =1: Use dual-spindle control.

=0: Not use dual-spindle control.

SEQ =1: Auto insert sequence number.

=0: Not auto insert sequence number.

SVCD =1: Use bus servo card.

=0: Not use bus servo card.

MODE =1: High-speed high-precision mode, can not modify #15.0 and #17.0, only support four axes three linkages.

=0: Normal mode, when high-speed high-precision mode is set to normal mode, #15.0 is set to 1 by default.

Standard setting: 1 0 0 0   0 0 0 1

System parameter number

0	0	1			SPM2	SPPT	SPEP	SPOM	SPT	SBUS	RASA

RASA =1: Use absolute grating scale.

=0: Not use absolute grating scale.

SBUS =1: Spindle drive unit is under bus control mode

=0: Spindle drive unit is under non-bus control mode

SPT =1: I/O point control.

=0: Frequency conversion or other.

SPOM =1: Burst frequency is selected for output of the 1st spindle speed.

=0: Analog voltage is selected for output of the 1st spindle speed.

SPEP =1: When the bus spindle is used, the feedback interface of spindle encoder is XS32 encoder interface.

=0: When the bus spindle is used, the feedback interface of spindle encoder shares the same interface with output.

SPPT =1: The spindle pulse output mode is AB phase output.

=0: The spindle pulse output mode is pulse + direction.

SPM2 =1: Burst frequency is selected for output of the 2nd spindle speed.

=0: Analog voltage is selected for output of the 2nd spindle speed.

Standard setting: 0 0 0 0   0 0 0 0

System parameter number

0	0	2					DEC5	DEC4	DEC3	DEC2	DEC1

DEC1 =1: When the 1st axis returns to reference point, decelerate if the deceleration signal is 1.

=0: When the 1st axis returns to reference point, decelerate if the deceleration signal is 0.

DEC2 =1: When the 2nd axis returns to reference point, decelerate if the deceleration signal is 1.

Appendix

	=0: When the 2nd axis returns to reference point, decelerate if the deceleration signal is 0.
DEC3	=1: When the 3rd axis returns to reference point, decelerate if the deceleration signal is 1. =0: When the 3rd axis returns to reference point, decelerate if the deceleration signal is 0.
DEC4	=1: When the 4th axis returns to reference point, decelerate if the deceleration signal is 1. =0: When the 4th axis returns to reference point, decelerate if the deceleration signal is 0.
DEC5	=1: When the 5th axis returns to reference point, decelerate if the deceleration signal is 1. =0: When the 5th axis returns to reference point, decelerate if the deceleration signal is 0.

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 reversed.
=0: The 1st axis feed direction is not reversed.
DIR2 =1: The 2nd axis feed direction is reversed.
=0: The 2nd axis feed direction is not reversed.
DIR3 =1: The 3rd axis feed direction is reversed.
=0: The 3rd axis feed direction is not reversed.
DIR4 =1: The 4th axis feed direction is reversed.
=0: The 4th axis feed direction is not reversed.
DIR5 =1: The 5th axis feed direction is reversed.
=0: The 5th axis feed direction is not reversed.

Standard setting: 0 0 0 0 0 0 0 0

System parameter number

0	0	4		SK0					TMSN		TMES

TMES =1: Tool setter installed
=0: Tool setter not installed
TMSN =1: Tool operation interface displays operation steps
=0: The tool setting interface does not display the operation steps
SK0 =1: When the skip signal SKIP is 0, inputted as a signal
=0: When the skip signal SKIP is 1, inputted as a signal

Standard setting: 0 0 0 0 0 0 0 0

System parameter number

0	0	5		DOUS	SPAP					ISC	

ISC =1: The minimum movement unit is 0.0001mm&deg, 0.00001inch.
=0: The minimum movement unit is 0.001mm&deg, 0.0001inch.
SPAP =1: Spindle interface XS23 is used as the 5th axis pulse output interface, and rigid tapping is invalid

	=0: Spindle interface XS23 is not used as the 5th axis pulse output interface
DOUS	=1: The dual-drive tool uses the grating position.
	=0: The 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

ZRN	=1: When the reference point is not created, and any code other than G28 is specified during automatic operation, the system gives an alarm.
	=0: When the reference point is not created, and any code other than G28 is specified during automatic operation, the system does not give an alarm.
ZMOD	=1: Return-to-zero mode selection: Before the stopper.
	=0: Return-to-zero mode selection: Behind the stopper.
JAX	=1: Manually return to the reference point while controlling the axis: Single-axis
	=0: Manually return to the reference point while controlling the axis: Multi-axis
AZR	=1: G28 command when the reference point is not created: Alarm
	=0: G28 command when the reference point is not created: Use stopper
SJZ	=1: Reference point memory
	=0: reference point does not remember
SIOD	=1: The mechanical return-to-zero deceleration signal is operated by PLC logic.
	=0: The mechanical return-to-zero deceleration signal is directly read.
ZPLS	=1: zero returning method selection: there is one-turn signal.
	=0: zero returning method selection: there is no one-turn signal.
MAOB	=1: zero returning method selection if there is no one-turn signal: method B.
	=0: zero returning method selection if there is no one-turn signal: method A.

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 direction of the 1st axis returning to the reference point: negative direction.
	=0: Set the direction of the 1st axis returning to the reference point: positive direction.
ZMI2	=1: Set the direction of the 2nd axis returning to the reference point: negative direction.
	=0: Set the direction of the 2nd axis returning to the reference point: positive direction.
ZMI3	=1: Set the direction of the 3rd axis returning to the reference point: negative direction.
	=0: Set the direction of the 3rd axis returning to the reference point: positive direction.
ZMI4	=1: Set the direction of the 4th axis returning to the reference point: negative direction.
	=0: Set the direction of the 4th axis returning to the reference point: positive

ZMI5	direction. =1: Set the direction of the 5th axis returning to the reference point: negative direction. =0: Set the direction of the 5th axis returning to the reference point: positive direction.

Standard setting: 0 0 0 0 0 0 0 0

System parameter number

0	0	8				ROT5	ROT4	ROT3	ROT2	ROT1

AXS1 =1: The 1st axis is set to rotating axis.
=0: The 1st axis is set to linear axis.

AXS2 =1: The 2nd axis is set to rotating axis.
=0: The 2nd axis is set to linear axis.

AXS3 =1: The 3rd axis is set to rotating axis.
=0: The 3rd axis is set to linear axis.

AXS4 =1: The 4th axis is set to rotating axis.
=0: The 4th axis is set to linear axis.

AXS5 =1: The 5th axis is set to rotating axis.
=0: The 5th axis is set to linear axis.

Standard setting: 0 0 0 0 1 0 0 0

System parameter number

0	0	9	DTO	RAB		ROS5	ROS4	ROS3	ROS2	ROS1

ROS1 =1: Coordinates=B type is selected for the rotating axis type of the 1st axis, and the coordinates are of linear axis type.
=0: Coordinates=A type is selected for the rotating axis type of the 1st axis, and the coordinates are 0-360 degrees.

ROS2 =1: Coordinates=B type is selected for the rotating axis type of the 2nd axis, and the coordinates are of linear axis type.
=0: Coordinates=A type is selected for the rotating axis type of the 2nd axis, and the coordinates are 0-360 degrees.

ROS3 =1: Coordinates=B type is selected for the rotating axis type of the 3rd axis, and the coordinates are of linear axis type.
=0: Coordinates=A type is selected for the rotating axis type of the 3rd axis, and the coordinates are 0-360 degrees.

ROS4 =1: Coordinates=B type is selected for the rotating axis type of the 4th axis, and the coordinates are of linear axis type.
=0: Coordinates=A type is selected for the rotating axis type of the 4th axis, and the coordinates are 0-360 degrees.

ROS5 =1: Coordinates=B type is selected for the rotating axis type of the 5th axis, and the coordinates are of linear axis type.
=0: Coordinates=A type is selected for the rotating axis type of the 5th axis, and the coordinates are 0-360 degrees.

RAB =1: Rotation axis work based on the principal of proximity
=0: Rotation axis does not work based on the principal of proximity

DTO =1: The input type of the rotating axis in cylindrical interpolation is expanded plane distance

=0: The input type of the rotating axis in cylindrical interpolation is angular

Standard setting: 0 0 0 0 0 0 0 1

System parameter number

0	1	0	RCUR	MSL	WCZS		RLC	ZCL	SCBM	

SCBM =1: Stroke detection before movement.

=0: No stroke detection before movement.

ZCL =1: Clear relative coordinate for reference point return.

=0: Not clear relative coordinate for reference point return.

RLC =1: Cancel relative coordinate system after reset.

=0: Not cancel relative coordinate system after reset.

WCZS =1: The zero point of the workpiece coordinate system is the machine coordinate plus the input value.

=0: The zero point of the workpiece coordinate system is the machine coordinate minus the input value.

MSL =1: Upon cycle start of multi-segment MDI, the starting line is the line where the cursor is positioned.

=0: Upon cycle start of multi-segment MDI, the starting line is the first line of the program.

RCUR =1: The cursor returns to the program starting point when reset under non-edit mode

=0: The cursor does not return to the program starting point when reset under non-edit mode

Standard setting: 0 0 0 0 0 0 1 0

System parameter number

0	1	1	BFA	LZR						OUT2

OUT2 =1: Do not enter into the area outside the second stroke limit.

=0: Do not enter into the area in the second stroke limit.

LZR =1: Stroke detection from power-on to manual return to the reference point.

=0: No stroke detection from power-on to manual return to the reference point.

BFA =1: When the over-stroke code is issued, give an alarm after exceeding the stroke.

=0: When the over-stroke code is issued, give an alarm before exceeding the stroke. (The system alarm range is 5MM in front of the boundary of the set prohibited 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 quick operation is valid from power-on to return to the reference point.

=0: Manual quick operation is invalid from power-on to return to the reference point.

LRP =1: Positioning (G00) interpolation type is straight line.

	=0: Positioning (G00) interpolation type non-linear line.
RFO	=1: Rapid feed, stop when rapid feed override is Fo.
	=0: Rapid feed, not stop when rapid feed override is Fo.
TDR	=1: idling is valid during tapping.
	=0: idling is invalid during tapping.
FDR	=1: idling is valid during cutting feed.
	=0: idling is invalid during cutting feed.
RDR	=1: idling is valid during rapid positioning.
	=0: idling is invalid during rapid positioning.

Standard setting: 0 0 0 0 0 0 0 0

System parameter number

0	1	3						HPC2		HPC	NPC

NPC =1: When no position encoder is installed, the feed is valid.
=0: When no position encoder is installed, the feed is Invalid.

HPC =1: The spindle installed position encoder.
=0: The spindle does not installed position encoder.

HPC2 =1: The second spindle installed position encoder.
=0: The second spindle does not installed position encoder.

Standard setting: 0 0 0 0 0 0 1 0

System parameter number

0	1	4		ROVT						DLF	

DLF =1: Manually return to zero point after reference point is created and memorized, and locate the reference point at the manual fast rate.
=0: Manually return to zero point after reference point is created and memorized, and locate the reference point at the quick positioning rate.

ROVT =1: Rapid Override is set to position 6.
=0: Rapid Override is set to position 4.

Standard setting: 0 0 0 0 0 0 0 0

System parameter number

0	1	5		PCKM	RPCK	PIIS		PPCK	ASL	PLAC	STL

STL =1: Select the pre-reading processing method.
=0: Select non-pre-reading processing method.

PLAC =1: Acceleration and deceleration mode after predictive control of interpolation: exponent form.
=0: Acceleration and deceleration mode after predictive control of interpolation: Linear type.

ASL =1: Automatic corner deceleration through predictive control: Speed difference control.
=0: Automatic corner deceleration through predictive control: Angle control.

PPCK =1: In-position check for predictive control.
=0: No in-position check for predictive control.

PIIS =1: Overlapping interpolation of acceleration/deceleration program segment is

Appendix

valid before predictive control.

=0: Overlapping interpolation of acceleration/deceleration program segment is invalid before predictive control.

RPCK =1: Quick positioning and in-position check are valid.

=0: Quick positioning and in-position check are invalid.

PCKM =1: Quick positioning done use the motor in-position flag.

=0: Quick positioning done does not use the motor in-position flag.

Standard setting: 0 1 0 0 0 0 0 0

System parameter number

0	1	6		ALS		JOG					PRT

PRT =1: Quick running acceleration/deceleration type: Constant acceleration.

=0: Quick running acceleration/deceleration type: Constant time.

JOG =1: Manual continuous feed rate for each axis is valid.

=0: Manual continuous feed rate for each axis is invalid.

ALS =1: Automatic corner override is valid.

=0: Automatic corner override is invalid.

Standard setting: 0 0 0 0 0 0 0 0

System parameter number

0	1	7		CPCT	CALT	WLOE			CLLE	CBLS	CBOL

CBOL =1: Cutting feed mode: Acceleration/deceleration after cutting.

=0: Cutting feed mode: Acceleration/deceleration before cutting.

CBLS =1: Acceleration/deceleration before cutting feed: S type.

=0: Acceleration/deceleration before cutting feed: Linear type.

CLLE =1: Acceleration and deceleration after cutting feed: exponent form.

=0: Acceleration and deceleration after cutting feed: Linear type.

WLOE =1: MPG operation selection: exponent form.

=0: MPG operation selection: Linear type.

CALT =1: Control on cutting feed acceleration.

=0: No control on cutting feed acceleration.

CPCT =1: In-position precision is controlled for cutting feed.

=0: In-position precision is not controlled for cutting feed.

Standard setting: 1 0 1 0 0 0 0 1

System parameter number

0	1	8		RVCS	RBK						

RBK =1: Backlash compensation separately for cutting/fast movement.

=0: No backlash compensation separately for cutting/fast movement.

RVCS =1: Backlash compensation mode: lifting speed.

=0: Backlash compensation mode: Fixed frequency.

Standard setting: 0 0 0 0 0 0 0 0

System parameter number

0	1	9			ALS2	ALMS	ALM5	ALM4	ALM3	ALM2	ALM1

ALM1 =1: Alarm will be triggered when the 1st axis drive unit alarm signal is 1.

=0: Alarm will be triggered when the 1st axis drive unit alarm signal is 0.

ALM2 =1: Alarm will be triggered when the 2nd axis drive unit alarm signal is 1.
=0: Alarm will be triggered when the 2nd axis drive unit alarm signal is 0.

ALM3 =1: Alarm will be triggered when the 3rd axis drive unit alarm signal is 1.
=0: Alarm will be triggered when the 3rd axis drive unit alarm signal is 0.

ALM4 =1: Alarm will be triggered when the 4th axis drive unit alarm signal is 1.
=0: Alarm will be triggered when the 4th axis drive unit alarm signal is 0.

ALM5 =1: Alarm will be triggered when the 5th axis drive unit alarm signal is 1.
=0: Alarm will be triggered when the 5th axis drive unit alarm signal is 0.

ALMS =1: Give an alarm when the alarm signal of the 1st spindle drive unit is 1.
=0: Give an alarm when the alarm signal of the 1st spindle drive unit is 0.

ALS2 =1: Give an alarm when the alarm signal of the 2nd spindle drive unit is 1.
=0: Give an alarm when the alarm signal of the 2nd spindle drive unit is 0.

Standard setting: 0 0 1 0 0 0 0 0

System parameter number

0	2	0							DIT	ITX	ITL

ITL =1: All axis interlocking signals are valid.
=0: All axis interlocking signals are invalid.

ITX =1: Interlocking signal for an individual axis is valid.
=0: Interlocking signal for an individual axis is invalid.

DIT =1: Interlocking signals along each axis direction are valid.
=0: Interlocking signals along each axis direction are invalid.

Standard setting: 0 0 0 0 0 0 0 0

System parameter number

0	2	1					APC5	APC4	APC3	APC2	APC1

APC1 =1: The current machine position of the 1st axis is set to machine zero.
=0: The current machine position of the 1st axis is not set to machine zero.

APC2 =1: The current machine position of the 2nd axis is set to machine zero.
=0: The current machine position of the 2nd axis is not set to machine zero.

APC3 =1: The current machine position of the 3rd axis is set to machine zero.
=0: The current machine position of the 3rd axis is not set to machine zero.

APC4 =1: The current machine position of the 4th axis is set to machine zero.
=0: The current machine position of the 4th axis is not set to machine zero.

APC5 =1: The current machine position of the 5th axis is set to machine zero.
=0: The current machine position of the 5th axis is not set to machine zero.

Standard setting: 0 0 0 0 0 0 0 0

System parameter number

0	2	2		PRC	DAL						

DAL =1: Consider tool length compensation for absolute position display.
=0: Not consider tool length compensation for absolute position display.

PRC =1: In the tool offset compensation, input directly workpiece coordinate system offset with PRC signal.
=0: In the tool offset compensation, input directly workpiece coordinate system offset without PRC signal.

Appendix

Standard setting: 0 0 0 0 0 0 0

System parameter number

0	**2**	**3**		**POSM**						

POSM =1: The program monitoring screen displays the modality.
=0: The program monitoring screen does not display the modality.

Standard setting: 0 1 0 0 0 0 0 0

System parameter number

0	**2**	**4**		**NPA**						

NPA =1: Switch to the alarm screen when an alarm is sent out.
=0: Not switch to the alarm screen when an alarm is sent out.

Standard setting: 0 0 0 0 0 0 0 0

System parameter number

0	**2**	**5**	**ALM**	**DGN**	**GRA**	**SET**		**SYS**	**PRG**	**POS**

POS =1: In the Position interface, press “Position” again, and switch the screen.
=0: In the Position interface, press “Position” again, and not switch the screen.

PRG =1: In the Program interface, press “Program” again, and switch the screen.
=0: In the Program interface, press “Program” again, and not switch the screen.

SYS =1: In the System interface, press “System” again, and switch the screen.
=0: In the System interface, press “System” again, and not switch the screen.

SET =1: In the Setting interface, press “Setting” again, and switch the screen.
=0: In the Setting interface, press “Setting” again, and not switch the screen.

GRA =1: In the Graph interface, press “Graph” again, and switch the screen.
=0: In the Graph interface, press “Graph” again, and not switch the screen.

DGN =1: In the Diagnose interface, press “Diagnose” again, and switch the screen.
=0: In the Diagnose interface, press “Diagnose” again, and not switch the screen.

ALM =1: In the Alarm interface, press “Alarm” again, and switch the screen.
=0: In the Alarm interface, press “Alarm” again, and not switch the screen.

Standard setting: 1 1 1 1 0 1 1 1

System parameter number

0	**2**	**6**	**HELP**	**PLC**				**SMDI**		**PETP**

PETP =1: Press “Edit” to automatically jump to the program screen.
=0: Press “Edit” not to automatically jump to the program screen.

SMDI =1: In MDI mode, press <Program> to automatically jump to the input interface.
=0: In MDI mode, press <Program> not to automatically jump to the input interface.

SMDT =1: In MDI mode, press <Program> to automatically jump to current/module interface selection.
=0: In MDI mode, press <Program> to automatically jump to MDI interface selection.

PLC =1: In the PLC interface, press “Program Control” again to switch the screen.
=0: In the PLC interface, press “Program Control” again not to switch the

screen.

HELP =1: In the Help interface, press "Help" again to switch the screen.
=0: In the Help interface, press "Help" again not to switch the screen.

Standard setting: 1 1 0 0 0 0 0 1

System parameter number

0	2	7					NE9				NE8

NE8 =1: Disable edit subroutine 80000 - 89999.
=0: Not disable edit subroutine 80000 - 89999.

NE9 =1: Disable edit subroutine 90000 - 99999.
=0: Edit working subroutine 90000 - 99999.

Standard setting: 0 0 0 1 0 0 0 1

System parameter number

0	2	8		MCL			MKP				

MKP =1: When executing M02, M30 or % in MDI mode, clear the program edited.
=0: When executing M02, M30 or % in MDI mode, not clear the program edited.

MCL =1: In MDI mode, press "Reset" to delete the program edited.
=0: In MDI mode, press "Reset" not to delete the program edited.

Standard setting: 0 0 0 1 0 0 0 0

System parameter number

0	2	9					IWZ	WZO	MCV	GOF	WOF

WOF =1: Do not enter the tool wear offset through MDI keyboard.
=0: Allowed to enter the tool wear offset through MDI keyboard.

GOF =1: Do not enter the tool geometrical offset through MDI keyboard.
=0: Allowed to enter the tool geometrical offset through MDI keyboard.

MCV =1: Do not enter macro program variables through the MDI keyboard.
=0: Allowed to enter macro program variables through the MDI keyboard.

WZO =1: Do not input the workpiece origin offset via the MDI keyboard.
=0: Allow input the workpiece origin offset via the MDI keyboard.

IWZ =1: Do not input the workpiece origin offset via the MDI keyboard during pauses.
=0: Allow input the workpiece origin offset via the MDI keyboard during pauses.

Standard setting: 0 0 0 0 0 0 0 0

System parameter number

0	3	1				G13	G91		G19	G18	G01

G01 =1: G01 mode when the power is turned on or status is cleared.
=0: G00 mode when the power is turned on or status is cleared.

G18 =1: The plane selection command is G18 when the power is turned on or status is cleared.
=0: The plane selection command is not G17 when the power is turned on or status is cleared.

G19 =1: For G19 mode, when G19=1, set G18 to 0.

Appendix

=0: Depend on Parameter No: 31#1。

G19	G18	G17, G18, G19 mode
0	0	G17 mode (X-Y plane)
0	1	G18 mode (Z-X plane)
1	0	G19 mode (Y-Z plane)

G91 =1: Set to G91 mode when the power is turned on or status is cleared.
=0: Set to G90 mode when the power is turned on or status is cleared.

G13 =1: Set G13 method when power on or clearing the status.
=0: Set G12 method when power on or clearing the status.

Standard setting: 0 0 1 0 0 0 0 0

System parameter number

0	3	2			AD2						

AD2 =1: If two or more identical addresses are commanded, the system alarms.
=0: If two or more identical addresses are commanded, the system does not alarm.

Standard setting: 0 1 0 0 0 0 0 0

System parameter number

0	3	3		M3B	MHI	M99	M30		M02		TF

TF =1: Execute the same T code to output the TF signal.
=0: Execute the same T code, but not output the TF signal.

M02 =1: When executing M02, return to the beginning of the program segment.
=0: When executing to M02, not return to the beginning of the program segment.

M30 =1: When executing to M30, return to the beginning of the program segment.
=0: When executing to M30, not return to the beginning of the program segment.

M99 =1: Execute M98M99I, but not output MF and code signals.
=0: Execute M98M99I, and output MF and code signals.

MHI =1: The exchange between M/S/T strobing pulse signal and the ending signal is in high-speed mode.
=0: The exchange between M/S/T strobing pulse signal and the ending signal is in general mode.

M3B =1: Up to three M codes can be commanded in a program.
=0: One M code can be commanded in a program.

Standard setting: 1 0 1 1 0 0 0 0

System parameter number

0	3	4		CFH							DWL

DWL =1: In per revolution feed mode, G04 is paused per revolution.
=0: In per revolution feed mode, G04 is not paused per revolution.

CFH =1: Clear F, H and D code during reset or emergency stop.

Appendix

=0: Reserve F, H and D code upon reset or emergency stop.

Standard setting: 0 0 0 0   0 0 0 0

System parameter number

0	3	5		C07		C05	C04	C03	C02	C01	

C01 =1: Clear G code in Group 01 upon reset or emergency stop.
=0: Reserve G code in Group 01 upon reset or emergency stop.
C02 =1: Clear G code in Group 02 upon reset or emergency stop.
=0: Reserve G code in Group 02 upon reset or emergency stop.
C03 =1: Clear G code in Group 03 upon reset or emergency stop.
=0: Reserve G code in Group 03 upon reset or emergency stop.
C04 =1: Clear G code in Group 04 upon reset or emergency stop.
=0: Reserve G code in Group 04 upon reset or emergency stop.
C05 =1: Clear G code in Group 05 upon reset or emergency stop.
=0: Reserve G code in Group 05 upon reset or emergency stop.
C07 =1: Clear G code in Group 07 upon reset or emergency stop.
=0: Reserve G code in Group 07 upon reset or emergency stop.

Standard setting: 1 0 0 0   0 0 0 0

System parameter number

0	3	6		C15	C14	C13	C12	C11	C10	C09	C08

C08 =1: Clear G code in Group 08 upon reset or emergency stop.
=0: Reserve G code in Group 08 upon reset or emergency stop.
C09 =1: Clear G code in Group 09 upon reset or emergency stop.
=0: Reserve G code in Group 09 upon reset or emergency stop.
C10 =1: Clear G code in Group 10 upon reset or emergency stop.
=0: Reserve G code in Group 10 upon reset or emergency stop.
C11 =1: Clear G code in Group 11 upon reset or emergency stop.
=0: Reserve G code in Group 11 upon reset or emergency stop.
C12 =1: Clear G code in Group 12 upon reset or emergency stop.
=0: Reserve G code in Group 12 upon reset or emergency stop.
C13 =1: Clear G code in Group 13 upon reset or emergency stop.
=0: Reserve G code in Group 13 upon reset or emergency stop.
C14 =1: Clear G code in Group 14 upon reset or emergency stop.
=0: Reserve G code in Group 14 upon reset or emergency stop.
C15 =1: Clear G code in Group 15 upon reset or emergency stop.
=0: Reserve G code in Group 15 upon reset or emergency stop.

Standard setting: 0 0 0 0   0 0 0 1

System parameter number

0	3	7		SCS	CSS	CSZR		SOC	RSC	BDP	SCRW

SCRW =1: Pitch compensation.
=0: No pitch compensation.
BDP =1: Use bidirectional pitch error compensation.
=0: Not use bidirectional pitch error compensation.
RSC =1: When G0 is quickly positioning, the reference coordinate of G96 spindle

speed is the current point.
=0: When G0 is quickly positioning, the reference coordinate of G96 spindle speed is the ending point.

SOC =1: G96 spindle speed is controlled after spindle override.
=0: G96 spindle speed is controlled before spindle override.

CSZR =1: Manually return to reference point when switching to CS contour control axis.
=0: Auto create reference point when switching to CS contour control axis.

CSS =1: CS contour control in each spindle.
=0: No CS contour control in each spindle.

SCS =1: Use CS contour control function.
=0: Not use CS contour control function.

Standard setting: 0 0 0 0 0 0 0 0

System parameter number

0	3	8		PG2	PG1	GTT	FLRE	FLR	EOV	MPP	SAR

SAR =1: Check the spindle speed arrival signal.
=0: Not check the spindle speed arrival signal.

MPP =1: In the multi-spindle control, select spindle via the program command P code.
=0: In the multi-spindle control, not select spindle via the program command P code.

EOV =1: Use each spindle override signal.
=0: Not use each spindle override signal.

FLR =1: The unit of the allowable rate (q) and the change rate (r) set for the spindle speed fluctuation detection is 0.1%.
=0: The unit of the allowable rate (q) and the change rate (r) set for the spindle speed fluctuation detection is 1%.

FLRE =1: Spindle speed fluctuation detection is valid.
=0: Spindle speed fluctuation detection is invalid.

GTT =1: Spindle gear selection mode is T type.
=0: Spindle gear selection mode is M type.

PG2, PG1: Gear ratio of the spindle to the position encoder. 00 is 1:1; 01 is 2:1; 10 is 4:1; 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 tool length compensation: Mode B.
=0: Select the type of tool length compensation: Mode A.

Standard setting: 0 0 0 0 0 0 0 1

System parameter number

0	4	0		ODI					CCN		SUP

SUP =1: Starting type in tool radius compensation: Type B.
=0: Starting type in tool radius compensation: Type A.

CCN =1: When G28, G30 command moves to the middle point, cancel the radius compensation.
=0: When G28, G30 command moves to the middle point, reserve the radius compensation.

ODI =1: The tool radius compensation is set to the diameter value.
=0: The tool radius compensation is set to the radius value.

Standard setting: 1 0 0 0 0 1 0 0

System parameter number

0	4	1		CNI	G39		PUIT			SCRT

SCRT =1: Select Mode A for pitch error compensation processing mode.
=0: Select General Mode for pitch error compensation processing mode.

PUIT =1: The input and display of the digital parameters are determined by the bit parameter NO.0#2 INI.
=0: The digital parameters are inputted and displayed in metric system.

G39 =1: In the radius compensation, the corner arc function is valid.
=0: In the radius compensation, the corner arc function is invalid.

CNI =1: Interference check for radius compensation.
=0: No interference check for radius compensation.

Standard setting: 0 1 1 0 0 0 0 0

System parameter number

0	4	2			RD2	RD1				

RD1 =1: Set G76, G87 tool retracting direction: Negative.
=0: Set G76, G87 tool retracting direction: Forward direction.

RD2 =1: Set G76, G87 tool retracting axis: Shaft 2.
=0: Set G76, G87 tool retracting axis: Shaft 1.

Standard setting: 0 0 0 0 0 0 0 0

System parameter number

0	4	3							QZA	

QZA =1: In deep-hole drilling (G73, G83), an alarm will be sent out if there is no command cutting depth.
=0: In deep-hole drilling (G73, G83), no alarm will be sent out if there is no command cutting depth.

Standard setting: 0 0 0 0 0 0 1 0

System parameter number

0	4	4			PCP	DOV			VGR	

VGR =1: The gear ratio of the spindle and position encoder can be arbitrary.
=0: The gear ratio of the spindle and position encoder is not arbitrary.

DOV =1: rate is valid when rigid tapping retracting.
=0: rate is invalid when rigid tapping retracting.

PCP =1: in high speed deep hole tapping cycle.
=0: in standard deep hole tapping cycle.

Standard setting: 0 0 0 0 0 0 0 0

System parameter number

0	**4**	**5**				**OVS**	**OVU**	**TDR**		**NIZ**

NIZ =1: treated with rigid tapping smoothing.
=0: not treated with rigid tapping smoothing.

TDR =1: rapid rapping feed, use the same time constant when retracting.
=0: rapid rapping feed, do not use the same time constant when retracting.

OVU =1: The unit of rigid tapping retracting rate is 10%.
=0: The unit of rigid tapping retracting rate is 1%.

OVS =1: feed speed rate selection and rate cancel signal are valid during rigid tapping.
=0: feed speed rate selection and rate cancel signal are invalid during rigid tapping.

Standard setting: 0 0 0 0 0 0 0 0

System parameter number

0	**4**	**6**			**ORI**				**SSOG**	

SSOG =1: spindle is controlled in way of servo when tapping starts.
=0: spindle is controlled in way of follow-up when tapping starts.

ORI =1: spindle stop accurately when tapping starts.
=0: spindle does not stop accurately when tapping starts.

Standard setting: 0 0 0 0 0 0 0 0

System parameter number

0	**4**	**7**		**XSC**	**SCL3**	**SCL2**	**SCL1**			**RIN**

RIN =1: Rotating angle of coordinate rotation: G90/G91 command.
=0: Rotating angle of coordinate rotation: Absolute command.

SCL1 =1: Scaling of the 1st axis is valid.
=0: Scaling of the 1st axis is invalid.

SCL2 =1: Scaling of the 2nd axis is valid.
=0: Scaling of the 2nd axis is invalid.

SCL3 =1: Scaling of the 3rd axis is valid.
=0: Scaling of the 3rd axis is invalid.

XSC =1: The scaling ratio for each axis is specified as I, J, and K.
=0: The scaling ratio for each axis is specified as P code.

Standard setting: 0 1 1 1 1 0 0 1

System parameter number

0	**4**	**8**								**MDL**

MDL =1: The unidirectional positioning G code is set to the modality code.
=0: The unidirectional positioning G code is not set to the modality code.

Standard setting: 0 0 0 0 0 0 0 0

System parameter number

0	**4**	**9**								**RPST**

RPST =1: Z axis moves in the way of G01during restart of the program.
=0: Z axis moves in the way of G00 at idling speed during restart of the program.

Standard setting: 0 0 0 0 0 0 0 0

System parameter number

0	5	0		SIM					REL	

REL =1: Display setting of related position of indexing table: Within 360°
=0: Display setting of related position of indexing table: Beyond 360°

SIM =1: sound alarm if indexing code and other control axis code segment are in the same segment.
=0: does not sound alarm if indexing code and other control axis code segment are in the same segment.

Standard setting: 0 1 0 0 0 0 0 0

System parameter number

0	5	1	MDLY		SBM					

SBM =1: Single segments can be used in macroprogram instruction statements.
=0: Single segments cannot be used in macroprogram instruction statements.

MDLY =1: No time delay is allowed in macroprogram instruction statements.
=0: Time delay in allowed in macroprogram instruction statements.

Standard setting: 0 0 0 0 0 0 0 0

System parameter number

0	5	2	CLV	CCV						

CCV =1: Macroprogram public variables #100 - #199 are emptied after reset.
=0: Macroprogram public variables #100 - #199 are not emptied after reset.

CLV =1: Macroprogram local variables #1 - #50 are emptied after reset.
=0: Macroprogram local variables #1 - #50 are not emptied after reset.

Standard setting: 0 0 0 0 0 0 0 0

System parameter number

0	5	3	LADD	GL200			LAD3	LDA2	LAD1	LAD0

LAD0~LAD3 are binary combined parameters. When it is 0, #0 ladder diagram is used; when it is 1~15, #0~15 ladder diagram is used.

GL200 =1: GL200-F bus input/output module is used.
=0: GL200-F bus input/output module is not used.

PLCV =1: Ladder diagrams can only be displayed with authority of machine tool manufacturer.
=0: Ladder diagrams can be displayed without authority of machine tool manufacturer.

Standard setting: 1 0 0 0 0 0 0 1

System parameter number

0	5	4	OPRG	PRGS	P,RT					

PRT =1: Parameters can only be modified with user authority and above.
=0: Parameters can be modified without user authority and above.

PRGS =1: Initial state of program startup is on.
=0: Initial state of program startup is off.

OPRG =1: One-key input/output is valid for part program with commissioning authority and above.

Appendix

=0: One-key input/output is invalid for part program with commissioning authority and above.

Standard setting: 0 1 0 0 0 1 1 1

System parameter number

0	5	5		ABP5	ABP4	ABP3	ABP2	ABP1			CANT

CANT =1: Single-piece machining time is automatically reset.
=0: Single-piece machining time is not automatically reset.

ABP1 =1: Axis 1 servo interface type is pulse.
=0: Axis 1 servo interface type is bus.

ABP2 =1: Axis 2 servo interface type is pulse.
=0: Axis 2 servo interface type is bus.

ABP3 =1: Axis 3 servo interface type is pulse.
=0: Axis 3 servo interface type is bus.

ABP4 =1: Axis 4 servo interface type is pulse.
=0: Axis 4 servo interface type is bus.

ABP5 =1: Axis 5 servo interface type is pulse.
=0: Axis 5 servo interface type is bus.

Standard setting: 0 0 0 0 0 0 0 0

System parameter number

0	5	6		HNG5	HNG4	HNG3	HNG2	HNG1		MANI	HPF

HPF =1: Complete operation is selected for manual pulse generator movement.
=0: Complete operation is not selected for manual pulse generator movement.

MANI =1: Manual intervention is quitted after switching from auto operation to manual operation.
=0: Manual intervention is not quitted after switching from auto operation to manual operation.

HNGD1 =1: Axis 1 moves in the same direction with turning direction of manual pulse generator.
=0: Axes 1 move in different direction from turning direction of manual pulse generator.

HNGD2 =1: Axis 2 moves in the same direction with turning direction of manual pulse generator.
=0: Axes 2 move in different direction from turning direction of manual pulse generator.

HNGD3 =1: Axis 3 moves in the same direction with turning direction of manual pulse generator.
=0: Axes 3 move in different direction from turning direction of manual pulse generator.

HNGD4 =1: Axis 4 moves in the same direction with turning direction of manual pulse generator.
=0: Axes 4 move in different direction from turning direction of manual pulse generator.

HNGD5 =1: Axis 5 moves in the same direction with turning direction of manual pulse generator.

=0: Axes 5 move in different direction from turning direction of manual pulse generator.

Standard setting: 0 0 0 0   0 0 0 1

System parameter number

0	5	8				V5	NEG4	NEG3	NEG2	NEG1

NEG1 =1: axis 1 is ignored.
=0: axis 1 is not ignored.
NEG2 =1: axis 2 is ignored.
=0: axis 2 is not ignored.
NEG3 =1: axis 3 is ignored.
=0: axis 3 is not ignored.
NEG4 =1: axis 4 is ignored.
=0: axis 4 is not ignored.
NEG5 =1: axis 5 is ignored.
=0: axis 5 is not ignored.

Standard setting: 0 0 0 0   0 0 0 0

System parameter number

0	5	9		LEDT						

LEDT =1: External program locks signals validly.
=0: External program locks signals invalidly.

Standard setting: 0 0 0 0   0 0 0 0

System parameter number

0	6	0	PMCA	PMCP	SCL			PMCS	EPW	

EPW =1: Maximum number of position switches is 16.
=1: Maximum number of position switches is 10.
PMCS =0: PMC axis selection is designated by signal G.
=0: PMC axis selection is not designated by signal G.
SCL =1: Use zooming.
=0: Not to use zooming.
PMCP =1: PMC axis zeroing selection --- speed signal
=0: PMC axis zeroing selection --- no-speed signal

Standard setting: 0 0 1 0   0 0 0 0

System parameter number

0	6	1	FALM	LALM	EALM	SALM	AALM			SSC

SSC =1: Use constant peripheral speed control.
=0: Not to use constant peripheral speed control.
AALM =1: Ignore external user alarm.
=0: Do not ignore external user alarm.
SALM =1: Ignore spindle drive unit alarm.
=0: Not ignore spindle drive unit alarm.

Appendix

EALM =1: Ignore e-stop alarm.
=0: Do not ignore e-stop alarm.
LALM =1: Ignore hard limit alarm.
=0: Not ignore hard limit alarm.
FALM =1: Ignore feed axis drive unit alarm.
=0: Not ignore feed axis drive unit alarm.

Standard setting: 0 0 0 0 0 0 0 0

2. Data Parameters

Parameter No.   Parameter value   Default value

0000	I/O channel, I/O device (0:Xon/Xoff 1:XModem 2:USB 3: NET)	2

Setting range: 0～3
Set as 0 or 1 when CNC communicates with PC via RS232 interface and set as 2 when it connects with a USB flash disk.

0001	Communication channel baud rate (DNC)	38400

Setting range: 0～115200 (unit: BPS)

0002	Communication channel baud rate (file transmission)	115200

Setting range: 0～115200 (unit: BPS)

0003	Whether CNC send/receives file via serial port active operation	0

Setting range: 0-1

0005	Number of CNC control axis	4

Setting range: 3～5

0006	System language selection (0: Chinese; 1: English)	0

Setting range: 0-1

0007	Setting of days of reminding in advance of expiration of time-limited shutdown (days)	7

Setting range: 0~99

0008	MDT data package size of Ethernet bus slave station	16

Setting range: 0~20

0009	Maximum retransmission times of Ethernet bus	10

Setting range: 0~30

0010	Axis 1 offset of external workpiece origin	0.0000

Setting range: -9999.9999~9999.9999 (mm)

0011	Axis 2 offset of external workpiece origin	0.0000

Appendix

Setting range: -9999.9999~9999.9999 (mm)

0012	Axis 3 offset of external workpiece origin	0.0000

Setting range: -9999.9999~9999.9999 (mm)

0013	Axis 4 offset of external workpiece origin	0.0000

Setting range: -9999.9999~9999.9999 (mm)

0014	Axis 5 offset of external workpiece origin	0.0000

Setting range: -9999.9999~9999.9999 (mm)

0015	Axis 1 workpiece origin offset of G54	0.0000

Setting range: -9999.9999～9999.9999 (mm)

0016	Axis 2 workpiece origin offset of G54	0.0000

Setting range: -9999.9999～9999.9999 (mm)

0017	Axis 3 workpiece origin offset of G54	0.0000

Setting range: -9999.9999～9999.9999 (mm)

0018	Axis 4 workpiece origin offset of G54	0.0000

Setting range: -9999.9999～9999.9999 (mm)

0019	Axis 5 workpiece origin offset of G54	0.0000

Setting range: -9999.9999～9999.9999 (mm)

0020	Axis 1 workpiece origin offset of G55	0.0000

Setting range: -9999.9999～9999.9999 (mm)

0021	Axis 2 workpiece origin offset of G55	0.0000

Setting range: -9999.9999～9999.9999 (mm)

0022	Axis 3 workpiece origin offset of G55	0.0000

Setting range: -9999.9999～9999.9999 (mm)

0023	Axis 4 workpiece origin offset of G55	0.0000

Setting range: -9999.9999～9999.9999 (mm)

0024	Axis 5 workpiece origin offset of G55	0.0000

Setting range: -9999.9999～9999.9999 (mm)

0025	Axis 1 workpiece origin offset of G56	0.0000

Setting range: -9999.9999～9999.9999 (mm)

0026	Axis 2 workpiece origin offset of G56	0.0000

Setting range: -9999.9999～9999.9999 (mm)

0027	Axis 3 workpiece origin offset of G56	0.0000

Setting range: -9999.9999～9999.9999 (mm)

0028	Axis 4 workpiece origin offset of G56	0.0000

Setting range: -9999.9999～9999.9999 (mm)

0029	Axis 5 workpiece origin offset of G56	0.0000

Setting range: -9999.9999～9999.9999 (mm)

0030	Axis 1 workpiece origin offset of G57	0.0000

Setting range: -9999.9999～9999.9999 (mm)

0031	Axis 2 workpiece origin offset of G57	0.0000

Setting range: -9999.9999～9999.9999 (mm)

0032	Axis 3 workpiece origin offset of G57	0.0000

Setting range: -9999.9999～9999.9999 (mm)

0033	Axis 4 workpiece origin offset of G57	0.0000

Setting range: -9999.9999～9999.9999 (mm)

0034	Axis 5 workpiece origin offset of G57	0.0000

Setting range: -9999.9999～9999.9999 (mm)

0035	Axis 1 workpiece origin offset of G58	0.0000

Setting range: -9999.9999～9999.9999 (mm)

0036	Axis 2 workpiece origin offset of G58	0.0000

Setting range: -9999.9999～9999.9999 (mm)

0037	Axis 3 workpiece origin offset of G58	0.0000

Setting range: -9999.9999～9999.9999 (mm)

0038	Axis 4 workpiece origin offset of G58	0.0000

Setting range: -9999.9999～9999.9999 (mm)

0039	Axis 5 workpiece origin offset of G58	0.0000

Setting range: -9999.9999～9999.9999 (mm)

0040	Axis 1 workpiece origin offset of G59	0.0000

Setting range: -9999.9999～9999.9999 (mm)

0041	Axis 2 workpiece origin offset of G59	0.0000

Setting range: -9999.9999～9999.9999 (mm)

0042	Axis 3 workpiece origin offset of G59	0.0000

Setting range: -9999.9999～9999.9999 (mm)

0043	Axis 4 workpiece origin offset of G59	0.0000

Setting range: -9999.9999～9999.9999 (mm)

0044	Axis 5 workpiece origin offset of G59	0.0000

Setting range: -9999.9999～9999.9999 (mm)

0045	Machine tool coordinate values of axis 1 of reference point 1	0.0000

Setting range: -9999.9999～9999.9999 (mm)

0046	Machine tool coordinate values of axis 2 of reference point 1	0.0000

Setting range: -9999.9999～9999.9999 (mm)

0047	Machine tool coordinate values of axis 3 of reference point 1	0.0000

Setting range: -9999.9999～9999.9999 (mm)

0048	Machine tool coordinate values of axis 4 of reference point 1	0.0000

Setting range: -9999.9999～9999.9999 (mm)

0049	Machine tool coordinate values of axis 5 of reference point 1	0.0000

Setting range: -9999.9999～9999.9999 (mm)

0050	Machine tool coordinate values of axis 1 of reference point 2	0.0000

Setting range: -9999.9999～9999.9999 (mm)

0051	Machine tool coordinate values of axis 2 of reference point 2	0.0000

Setting range: -9999.9999～9999.9999 (mm)

0052	Machine tool coordinate values of axis 3 of reference point 2	0.0000

Setting range: -9999.9999～9999.9999 (mm)

0053	Machine tool coordinate values of axis 2 of reference point 4	0.0000

Setting range: -9999.9999～9999.9999 (mm)

0054	Machine tool coordinate values of axis 5 of reference point 2	0.0000

Setting range: -9999.9999～9999.9999 (mm)

0055	Machine tool coordinate values of axis 1 of reference point 3	0.0000

Setting range: -9999.9999～9999.9999 (mm)

0056	Machine tool coordinate values of axis 2 of reference point 3	0.0000

Setting range: -9999.9999~9999.9999 (mm)

0057	Machine tool coordinate values of axis 3 of reference point 3	0.0000

Setting range: -9999.9999～9999.9999 (mm)

0058	Machine tool coordinate values of axis 4 of reference point 3	0.0000

Setting range: -9999.9999～9999.9999 (mm)

0059	Machine tool coordinate values of axis 5 of reference point 3	0.0000

Setting range: -9999.9999～9999.9999 (mm)

0060	Machine tool coordinate values of axis 1 of reference point 4	0.0000

Setting range: -9999.9999～9999.9999 (mm)

0061	Machine tool coordinate values of axis 2 of reference point 4	0.0000

Setting range: -9999.9999~9999.9999 (mm)

0062	Machine tool coordinate values of axis 3 of reference point 4	0.0000

Setting range: -9999.9999~9999.9999 (mm)

0063	Machine tool coordinate values of axis 4 of reference point 4	0.0000

Setting range: -9999.9999~9999.9999 (mm)

0064	Machine tool coordinate values of axis 5 of reference point 4	0.0000

Setting range: -9999.9999~9999.9999 (mm)

0066	Boundary coordinate values in negative direction of axis 1 in travel detection 1	-9999

Setting range: -9999.9999~9999.9999 (mm)

0067	Boundary coordinate values in positive direction of axis 1 in travel detection 1	9999

Setting range: -9999.9999~9999.9999 (mm)

0068	Boundary coordinate values in negative direction of axis 2 in travel detection 1	-9999

Setting range: -9999.9999~9999.9999 (mm)

0069	Boundary coordinate values in positive direction of axis 2 in travel detection 1	9999

Setting range: -9999.9999~9999.9999 (mm)

0070	Boundary coordinate values in negative direction of axis 3 in travel detection 1	-9999

Setting range: -9999.9999~9999.9999 (mm)

0071	Boundary coordinate values in positive direction of axis 3 in travel detection 1	9999

Setting range: -9999.9999~9999.9999 (mm)

0072	Boundary coordinate values in negative direction of axis 4 in travel detection 1	-9999

Setting range: -9999.9999~9999.9999 (mm)

0073	Boundary coordinate values in positive direction of axis 4 in travel detection 1	9999

Setting range: -9999.9999~9999.9999 (mm)

0074	Boundary coordinate values in positive direction of axis 5 in travel detection 1	-9999

Setting range: -9999.9999~9999.9999 (mm)

0075	Boundary coordinate values in positive direction of axis 5 in travel detection 1	9999

Setting range: -9999.9999~9999.9999 (mm)

0076	Boundary coordinate values in negative direction of axis 1 in travel detection 2	-9999

Setting range: -9999.9999~9999.9999 (mm)

0077	Boundary coordinate values in positive direction of axis 1 in travel detection 2	9999

Setting range: -9999.9999~9999.9999 (mm)

0078	Boundary coordinate values in negative direction of axis 2 in travel detection 2	-9999

Setting range: -9999.9999~9999.9999 (mm)

0079	Boundary coordinate values in positive direction of axis 2 in travel detection 2	9999

Setting range: -9999.9999~9999.9999 (mm)

0080	Boundary coordinate values in negative direction of axis 3 in travel detection 2	-9999

Setting range: -9999.9999~9999.9999 (mm)

0081	Boundary coordinate values in positive direction of axis 3 in travel detection 2	9999

Setting range: -9999.9999~9999.9999 (mm)

0082	Boundary coordinate values in negative direction of	-9999

	axis 4 in travel detection 2	

Setting range: -9999.9999~9999.9999 (mm)

0083	Boundary coordinate values in positive direction of axis 4 in travel detection 2	9999

Setting range: -9999.9999~9999.9999 (mm)

0084	Boundary coordinate values in negative direction of axis 5 in travel detection 2	-9999

Setting range: -9999.9999~9999.9999 (mm)

0085	Boundary coordinate values in positive direction of axis 5 in travel detection 2	9999

Setting range: -9999.9999~9999.9999 (mm)

0086	Idling speed	5000

Setting range: 0～9999 (mm/min)

0087	The cutting feed speed when the power is switched on	300

Setting range: 0～9999 (mm/min)

0088	1st axis G0 rapid operation speed	5000

Setting range:

Metric system: 0～30000 (mm/min)
Imperial: 0～30000/ 25.4 (inch/min)
Rotation axis: 0～30000（deg/min）

0089	2nd axis G0 rapid operation speed	5000

Setting range:

Metric system: 0～30000 (mm/min)
Imperial: 0～30000/ 25.4 (inch/min)
Rotation axis: 0～30000（deg/min）

0090	3rd axis G0 rapid operation speed	5000

Setting range:

Metric system: 0～30000 (mm/min)
Imperial: 0～30000/ 25.4 (inch/min)
Rotation axis: 0～30000（deg/min）

0091	4th axis G0 rapid operation speed	5000

Setting range:

Metric system: 0～30000 (mm/min)
Imperial: 0～30000/ 25.4 (inch/min)
Rotation axis: 0～30000（deg/min）

0092	5th axis G0 rapid operation speed	5000

Setting range:

Metric system: 0～30000 (mm/min)
Imperial: 0～30000/ 25.4 (inch/min)

Rotation axis: 0～30000（deg/min）

0093	Fo speed of quick running ratio of each axis (for all axes)	30

Setting range: 0～1000 (mm/min)

0094	Highest control speed of quick positioning (for all axes)	8000

Setting range: 300～30000(mm/min)

0095	Lowest control speed of quick positioning (for all axes)	0

Setting range: 0～300 (mm/min)

0096	Highest control speed of cutting feed (for all axes)	6000

Setting range: 300～9999 (mm/min)

0097	Lowest control speed of cutting feed (for all axes)	0

Setting range: 0～300 (mm/min)

0098	Speed of manual (JOG) continuous feed of each axis	2000

Setting range: 0～9999 (mm/min)

0099	Speed of obtaining Z pulse signal (FL) (for all axes)	40

Setting range: 1～60 (mm/min)

0100	Fast speed of axis 1 returning to reference point	4000

Setting range: 0～9999 (mm/min)

0101	Fast speed of axis 2 returning to reference point	4000

Setting range: 0～9999 (mm/min)

0102	Fast speed of axis 3 returning to reference point	4000

Setting range: 0～9999 (mm/min)

0103	Fast speed of axis 4 returning to reference point	4000

Setting range: 0～9999 (mm/min)

0104	Fast speed of axis 5 returning to reference point	4000

Setting range: 0～9999 (mm/min)

0110	1st axis rapid positioning acceleration (mm/s^2)	2000

Setting range: 1～9999 (mm/s^2)

0111	2nd axis rapid positioning acceleration (mm/s^2)	2000

Setting range: 1～9999 (mm/s^2)

0112	3rd axis rapid positioning acceleration (mm/s^2)	2000

Setting range: 1～9999 (mm/s^2)

0113	4th axis rapid positioning acceleration (mm/s^2)	2000

Setting range: 1～9999(mm/s^2)

0114	5th axis rapid positioning acceleration (mm/s^2)	2000

Setting range: 1～9999(mm/s^2)

0115	S-type acceleration/deceleration time constant T1 of axis 1 quick positioning	70

Setting range: 0～400 (ms)

0116	S-type acceleration/deceleration time constant T1 of axis 2 quick positioning	70

Setting range: 0～400 (ms)

0117	S-type acceleration/deceleration time constant T1 of axis 3 quick positioning	70

Setting range: 0～400 (ms)

0118	S-type acceleration/deceleration time constant T1 of axis 4 quick positioning	70

Setting range: 0～400 (ms)

0119	S-type acceleration/deceleration time constant T1 of axis 5 quick positioning	70

Setting range: 0～400 (ms)

0120	S-type acceleration/deceleration time constant T2 of axis 1 quick positioning	30

Setting range: 0～400 (ms)

0121	S-type acceleration/deceleration time constant T2 of axis 2 quick positioning	30

Setting range: 0～400 (ms)

0122	S-type acceleration/deceleration time constant T2 of axis 3 quick positioning	30

Setting range: 0～400 (ms)

0123	S-type acceleration/deceleration time constant T2 of axis 4 quick positioning	30

Setting range: 0～400 (ms)

0124	S-type acceleration/deceleration time constant T2 of axis 5 quick positioning	30

Setting range: 0～400 (ms)

0125	Acceleration/deceleration L-type time constant before cutting feed	100

Setting range: 3～400 (ms)

0126	Acceleration/deceleration S-type time constant before cutting feed	100

Setting range: 3～400 (ms)

0127	Acceleration/deceleration L-type time constant after cutting feed	80

Setting range: 3～400 (ms)

0128	Acceleration/deceleration E-type time constant before cutting feed	60

Setting range: 3～400 (ms)

0129	FL speed of exponential acceleration/deceleration	10

Setting range: 0～9999 (mm/min)

0130	Maximum merging program segments of pre-interpolation	0

Setting range: 0~10

0131	Positioning accuracy of cutting feed	0.03

Setting range: 0.001～0.5 (mm)

0132	Arc interpolation control accuracy	0.03

Setting range: 0～0.5 (mm)

0133	Contour control accuracy of pre-interpolation	0.01

Setting range: 0.01～0.5 (mm)

0134	Accelerated speed of linear acceleration/deceleration before interpolation in predicted control mode	250

Setting range: 0～2000 (mm/s²)

0135	S-type pre-acceleration/deceleration constant in predicted control mode	100

Setting range: 0～400 (ms)

0136	Linear acceleration/deceleration time constant of post-acceleration/deceleration in predicted control	80

Setting range: 0～400 (ms)

0137	Exponential acceleration/deceleration time constant of post-acceleration/deceleration in predicted control	60

Setting range: 0～400 (ms)

0138	Exponential acceleration/deceleration FL speed of feeding cut in predicted control mode	10

Setting range: 0～400 (ms)

Appendix

0139	Contour control accuracy of predicted control mode	0.01

Setting range: 0～0.5 (mm)

0140	Merging segment number of predicted control mode	0

Setting range: 0~10

0141	Positioning accuracy of predicted control mode	0.05

Setting range: 0～0.5 (mm)

0142	Axis rapid positioning maximum acceleration (mm/s^2)	3500

Setting range: 1～9999 (mm/s^2)

0143	Quick positioning acceleration/deceleration FL speed (mm/min)	30

Setting range: 0～9999 (mm/min)

0144	Predicted control mode, critical included angle of two programs in automatic corner deceleration	5

Setting range: 2~178°

0145	Predicted control mode, lowest feeding speed in automatic corner deceleration	120

Setting range: 10～1000 (mm/min)

0146	Predicted control mode, allowed offset for each axis in speed difference-based deceleration	80

Setting range: 60～1000

0147	Predicted control mode, cutting process accuracy level	2

Setting range: 0～8

0148	External accelerated speed limit of arc interpolation	1,000

Setting range: 100～5000 (mm/s^2)

0149	Low speed lower limit external accelerated speed clamping of arc interpolation	200

Setting range: 0～2000 (mm/min)

0150	Accelerated speed clamp time constant of cutting feed	50

Setting range: 0～1000(ms)

0151	Highest clamp speed of incomplete operation mode of manual pulse generator	4000

Setting range: 0～6000 (mm/min)

0152	Linear acceleration/deceleration time constant of manual pulse generator	120

Setting range: 0～400 (ms)

Appendix

0153	Exponential acceleration/deceleration time constant of manual pulse generator	80

Setting range: 0～400 (ms)

0154	Acceleration clamp time constant of manual pulse generator	100

Setting range: 0～400 (ms)

0155	Highest clamp speed of single-step feed	1,000

Setting range: 0～3000 （mm/min）

0156	Linear acceleration/deceleration time constant of JOG feed of each axis	50

Setting range: 0～400 (ms)

0157	S-type acceleration/deceleration time constant of JOG feed of each axis	40

Setting range: 0～400 (ms)

0158	Accelerated speed clamp constant of incomplete operation mode of manual pulse generator	50

Setting range: 0～1000 (ms)

0160	Axis 1 instruction clock multiplier factor (CMR)	1

Setting range: 1～65536

0161	Axis 2 instruction clock multiplier factor (CMR)	1

Setting range: 1～65536

0162	Axis 3 instruction clock multiplier factor (CMR)	1

Setting range: 1～65536

0163	Axis 4 instruction clock multiplier factor (CMR)	1

Setting range: 1～65536

0164	Axis 5 instruction clock multiplier factor (CMR)	1

Setting range: 1～65536

0165	Axis 1 instruction clock division factor (CMD)	1

Setting range: 1～65536

0166	Axis 2 instruction clock division factor (CMD)	1

Setting range: 1～65536

0167	Axis 3 instruction clock division factor (CMD)	1

Setting range: 1～65536

0168	Axis 4 instruction clock division factor (CMD)	1

Setting range: 1～65536

0169	Axis 5 instruction clock division factor (CMD)	1

Setting range: 1～65536

0170	1st axis manual rapid positioning speed	5000

Setting range: 0～30000

0171	2nd axis manual rapid positioning speed	5000

Setting range: 0～30000

0172	3rd axis manual rapid positioning speed	5000

Setting range: 0～30000

0173	4th axis manual rapid positioning speed	5000

Setting range: 0～30000

0174	5th axis manual rapid positioning speed	5000

Setting range: 0～30000

0175	Axis 1program name	0

Setting range: 0～8 (0:X 1:Y 2:Z 3:A 4:B 5:C 6:U 7:V 8:W)

0176	Axis 2 program name	1

Setting range: 0～8 (0:X 1:Y 2:Z 3:A 4:B 5:C 6:U 7:V 8:W)

0177	Axis 3 program name	2

Setting range: 0～8 (0:X 1:Y 2:Z 3:A 4:B 5:C 6:U 7:V 8:W)

0178	Axis 4 program name	3

Setting range: 0～8 (0:X 1:Y 2:Z 3:A 4:B 5:C 6:U 7:V 8:W)

0179	Axis 5 program name	4

Setting range: 0～8 (0:X 1:Y 2:Z 3:A 4:B 5:C 6:U 7:V 8:W)

0180	1st axis grid offset or reference point offset	0

Setting range: -50～50

0181	2nd axis grid offset or reference point offset	0

Setting range: -50～50

0182	3rd axis grid offset or reference point offset	0

Setting range: -50～50

0183	4th axis grid offset or reference point offset	0

Setting range: -50～50

0184	Axis 5 grid offset or reference point offset	1

Setting range: -50～50

0185	Machine tool axis Z frictional compensation mode	0

Setting range: 0~3　0: invalid; 1: up; 2: down; 3: up and down

0186	Machine tool axis Z frictional compensation (mm)	0.5

Setting range: 0～0.5

0187	Axis Z reverse backlash compensation condition (default as 1)	1

Setting range: 0～50

0188	Accumulative distance of axis Z reverse backlash compensation (default as 0.02)	0.02

Setting range: 0～0.5

0189	Reverse accuracy determined in reserve backlash compensation (X0.0001)	0.0100

Setting range: 0.0001～1.0000 (mm)

After setting $\alpha = p(189) \times 0.0001$ and reverse feeding, single-servo periodic feed is larger than α. Determine reverse backlash and start compensation.
Thus, when machining contour of excircle with large radius, small accuracy should be set in order to prevent compensation position from deviating from pass-quadrant position. When machining curved surface, large accuracy should be set in order to ensure that each path compensates reverse backlash at fixed position to form a raised ridge and that backlash compensation is evenly distributed within certain width.

0190	Axis 1 reverse backlash compensation	0.0000

Setting range:
　Metric system: -0.5～0.5 (mm)
　Imperial: -0.5～0.5/25.4 (inch)
　Rotation axis: -0.5～0.5000 (deg)

0191	Axis 2 reverse backlash compensation	0.0000

Setting range:
　Metric system: -0.5～0.5 (mm)
　Imperial: -0.5～0.5/25.4 (inch)
　Rotation axis: -0.5～0.5 (deg)

0192	Axis 3 reverse backlash compensation	0.0000

Setting range:
　Metric system: -0.5～0.5 (mm)
　Imperial: -0.5～0.5/25.4 (inch)
　Rotation axis: -0.5～0.5 (deg)

0193	Axis 4 reverse backlash compensation	0.0000

Setting range:
Metric system: -0.5～0.5 (mm)
Imperial: -0.5～0.5/25.4 (inch)
Rotation axis: -0.5～0.5(deg)

0194	Axis 5 reverse backlash compensation	0.0000

Setting range:
Metric system: -0.5～0.5(mm)
Imperial: -0.5～0.5/25.4 (inch)
Rotation axis: -0.5～0.5 (deg)

0195	Step size of compensation in fixed frequency of axis 1 backlash	0.0030

Setting range: 0～0.5 (mm)

0196	Step size of compensation in fixed frequency of axis 2 backlash	0.0030

Setting range: 0～0.5 (mm)

0197	Step size of compensation in fixed frequency of axis 3 backlash	0.0030

Setting range: 0～0.5 (mm)

0198	Step size of compensation in fixed frequency of axis 4 backlash	0.0030

Setting range: 0～0.5 (mm)

0199	Step size of compensation in fixed frequency of axis 5 backlash	0.0030

Setting range: 0～0.5 (mm)

0200	Time constant of compensation in lifting speed of reverse backlash	20

Setting range: 0～400 (ms)

0201	Reverse backlash compensation mode	0

Setting range: 0~2 0: mode A; 1: mode B; 2: mode C

0202	Receivable width of M, S and T completion signals	0

Setting range: 0～9999 (ms)

0203	Output time of reset signal	200

Setting range: 50～400 (ms)

0204	Allowed digit number of M code	2

Setting range: 1-2

0205	Allowed digit number of S code	5

Setting range: 1～6

0206	Allowed digit number of T code	4

Setting range: 1～4

0209	Condition of frictional compensation of machine tool axis Z (default: 1.0）	0

Setting range: 0～0

0210	Increment of number during automatic insertion of sequence number	10

Setting range: 0~1000

0211	Entering starting number of tool offset via MDI is prohibited	0

Setting range: 0~9999

0212	Entering number of tool offset via MDI is prohibited	0

Setting range: 0~9999

0214	Arc radius error limit	0.05

Setting range: 0.0001～0.1000 (mm)

0216	Pitch error compensation number for the 1st axis reference point	0

Setting range: 0~9999

0217	Pitch error compensation number for the 2nd axis reference point	0

Setting range: 0~9999

0218	Pitch error compensation number for the 3rd axis reference point	0

Setting range: 0~9999

0219	Pitch error compensation number for the 4th axis reference point	0

Setting range: 0~9999

0220	Pitch error compensation number for the 5th axis reference point	0

Setting range: 0~9999

0221	Screw pitch compensation for zero point when axis 1 moves from opposite direction of zeroing to zero point (absolute value)	0

Setting range: -0.9999～0.9999

0222	Screw pitch compensation for zero point when axis 2 moves from opposite direction of zeroing to zero	0

	point (absolute value)	

Setting range: -0.9999～0.9999

0223	Screw pitch compensation for zero point when axis 3 moves from opposite direction of zeroing to zero point (absolute value)	0

Setting range: -0.9999～0.9999

0224	Screw pitch compensation for zero point when axis 4 moves from opposite direction of zeroing to zero point (absolute value)	0

Setting range: -0.9999～0.9999

0225	Screw pitch compensation for zero point when axis 5 moves from opposite direction of zeroing to zero point (absolute value)	0

Setting range: -0.9999～0.9999

0226	1st axis pitch error compensation interval	5

Setting range: 0～9999.9999

0227	2nd axis pitch error compensation interval	5

Setting range: 0～9999.9999

0228	3rd axis pitch error compensation interval	5

Setting range: 0～9999.9999

0229	4th axis pitch error compensation interval	5

Setting range: 0～9999.9999

0230	5th axis pitch error compensation interval	5

Setting range: 0～9999.9999

0231	Reverse backlash compensation during quick movement of axis 1	0

Setting range: -0.5～0.5

0232	Reverse backlash compensation during quick movement of axis 2	0

Setting range: -0.5～0.5

0233	Reverse backlash compensation during quick movement of axis 3	0

Setting range: -0.5～0.5

0234	Reverse backlash compensation during quick movement of axis 4	0

Setting range: -0.5～0.5

0235	Reverse backlash compensation during quick movement of axis 5	0

Setting range: -0.5～0.5

0240	Gain adjustment data of main axis 1 speed analog output	1

Setting range: 0.98～1.02

0241	Gain adjustment data of main axis 2 speed analog output	1

Setting range: -0.2～0.2

0242	Gain adjustment data of main axis 3 speed analog output	1

Setting range: 0～9999 (r/min)

0243	Gain adjustment data of main axis 4 speed analog output	1

Setting range: 0～9999 (r/min)

0244	Compensation value of bias voltage of main axis 1 speed analog output	0

Setting range: 0～9999 (r/min)

0245	Compensation value of bias voltage of main axis 2 speed analog output	0

Setting range: 0～9999 (r/min)

0246	Compensation value of bias voltage of main axis 3 speed analog output	0

Setting range: 0～99999 (r/min)

0247	Compensation value of bias voltage of main axis 4 speed analog output	0

Setting range: 0～99999 (r/min)

0248	Main axis rotation speed during orientation or inching of main axis 1	50

Setting range: 0～99999 (r/min)

0249	Main axis rotation speed during orientation or inching of main axis 2	50

Setting range: 0～99999 (r/min)

0250	Main axis rotation speed during orientation or inching of main axis 3	50

Setting range: 0～1000 (r/min)

0251	Main axis rotation speed during orientation or inching of main axis 4	50

Setting range: 0～99999 (r/min)

0252	Maximum rotation speed of 1st spindle corresponding to 1st gear	6000

Setting range: 0～99999 (r/min)

0253	Correspond to highest speed of main axis 2 of gear 1	6000

Setting range: 0～99999 (r/min)

0254	Correspond to highest speed of main axis 3 of gear 1	6000

Setting range: 0～99999 (r/min)

0255	Maximum rotation speed of 1th spindle corresponding to 4th gear	6000

Setting range: 0～99999 (r/min)

0256	Maximum rotation speed of 1th spindle corresponding to 2th gear	6000

Setting range: 0～99999 (r/min)

0257	Maximum rotation speed of 2th spindle corresponding to 2th gear	6000

Setting range: 0～99999 (r/min)

0258	Maximum rotation speed of 3th spindle corresponding to 2th gear	6000

Setting range: 0～99999 (r/min)

0259	Maximum rotation speed of 4th spindle corresponding to 2th gear	6000

Setting range: 0～99999 (r/min)

0260	Maximum rotation speed of 1th spindle corresponding to 3th gear	6000

Setting range: 0～99999 (r/min)

0261	Maximum rotation speed of 2th spindle corresponding to 3th gear	6000

Setting range: 0～99999 (r/min)

0262	Maximum rotation speed of 3th spindle corresponding to 3th gear	6000

Setting range: 0～99999 (r/min)

0263	Maximum rotation speed of 4th spindle corresponding to 3th gear	6000

Setting range: 0～99999 (r/min)

0264	Maximum rotation speed of 1th spindle corresponding to 4th gear	6000

Setting range: 0～99999 (r/min)

0265	Maximum rotation speed of 2th spindle corresponding to 4th gear	6000

Setting range: 0～99999 (r/min)

0266	Maximum rotation speed of 3th spindle corresponding to 4th gear	6000

Setting range: 0～99999 (r/min)

0267	Maximum rotation speed of 4th spindle corresponding to 4th gear	6000

Setting range: 0～99999 (r/min)

0268	Lowest clamp speed of main axis 1 motor	0

Setting range: 0~99999

02689	Lowest clamp speed of main axis 2 motor	0

Setting range: 0~99999

0270	Lowest clamp speed of main axis 3 motor	0

Setting range: 0~99999

0271	Lowest clamp speed of main axis 4 motor	0

Setting range: 0~99999

0272	Lowest clamp speed of main axis 1 motor	6000

Setting range: 0~99999

0273	Lowest clamp speed of main axis 2 motor	6000

Setting range: 0~99999

0274	Lowest clamp speed of main axis 3 motor	6000

Setting range: 0~99999

0275	Lowest clamp speed of main axis 4 motor	6000

Setting range: 0~99999

0278	Upper speed limit of main axis 1	6000

Setting range: 0~99999

0279	Upper speed limit of main axis 2	6000

Setting range: 0~99999

0280	Upper speed limit of main axis 3	6000

Setting range: 0~99999

0281	Upper speed limit of main axis 4	6000

Setting range: 0~99999

0282	Encoder line number of main axis 1	1024

Setting range: 1000～9999

0283	Encoder line number of main axis 2	1024

Setting range: 1000～9999

0284	Encoder line number of main axis 3	1024

Setting range: 1000～9999

0285	Encoder line number of main axis 4	1024

Setting range: 1000～9999

0286	Upper limit of main axis during tapping cycle	2000

Setting range: 1～5000

0287	Acceleration/deceleration time constant of main axis in main axis pre-stop mode of flexible tapping	100

Setting range: 1～9999

0290	Vector limit value is ignored when moving along external side of corner during tool radius compensation C	0

Setting range: 0~999

0291	Maximum value of tool wear compensation	400

Setting range: 0~9999

0292	Maximum error allowed during tool radius compensation C	0.001

Setting range: 0.0001～0.01

0293	Factor of helix entry radius during groove cycle	1.5

Setting range: 0.01～3

0294	Tool retraction amount of high-speed deep hole cycle G73	2

Setting range: 0~9999

0295	Tool retraction or reserve amount of fixed cycle G83	2

Setting range: 0~9999

0296	Shortest drilling pause time	250

Setting range: 0~1000

0297	Longest drilling pause time	9999

Setting range: 1000～9999

0298	Tool retraction ratio during rigid tapping	100

Setting range: 0～100

0300	Retraction or reserve amount during deep hold tapping cycle	2

Setting range: 0～100

0301	Delay alarm time setting of rigid tapping boot timeout (ms)	1,000

Setting range: 0~9999

0302	Smoothing time constant of rigid tapping speed	20

Setting range: 0～200

0303	Highest accelerated speed during rigid tapping	300

Setting range: 0~1000

0304	Rigid tapping algorithm selection (1, 2, 3)	3

Setting range: 1～3

0305	Compensation for whole circle reverse jumping of G17 plane	0

Setting range: 0～5

0306	Compensation for whole circle reverse jumping of G18 plane	0

Setting range: 0～5

0307	Compensation for whole circle reverse jumping of G19 plane	0

Setting range: 0～5

0329	Rotation angle used when there is no rotation angle instruction during rotation of coordinates	0

Setting range: 0～9999.9999

0330	Zoom ratio used when there is no zoom ratio instruction	1

Setting range: 0.0001～9999.9999

0331	Axis 1 zoom ratio	1

Setting range: 0.0001～9999.9999

0332	Axis 2 zoom ratio	1

Setting range: 0.0001～9999.9999

0333	Axis 3 zoom ratio	1

Setting range: 0.0001～9999.9999

0334	Pause time of unidirectional positioning	0

Setting range: 0～10(S)

0335	Unidirectional positioning direction and overrun of axis 1	0

Setting range: -99.9999～99.9999

0336	Unidirectional positioning direction and overrun of axis 2	0

Setting range: -99.9999～99.9999

0337	Unidirectional positioning direction and overrun of axis 3	0

Setting range: -99.9999～99.9999

0338	Unidirectional positioning direction and overrun of axis 4	0

Setting range: -99.9999～99.9999

0339	Unidirectional positioning direction and overrun of axis 5	0

Setting range: -99.9999～99.9999

0340	Decoding pre-treatment segment number	20

Setting range: 1～64

0341	Buffer zone size of ARM interpolation point	36

Setting range: 0～99999

0342	1st axis zero returning low speed	200

Setting range: 0~1000

0343	2nd axis zero returning low speed	200

Setting range: 0~1000

0344	3rd axis zero returning low speed	200

Setting range: 0~1000

0345	4th axis zero returning low speed	200

Setting range: 0~1000

0346	5th axis zero returning low speed	200

Setting range: 0~1000

0347	Torque limit ratio of axis 1 torque limit skipping instruction (%)	100

Setting range: 1～200

0348	Torque limit ratio of axis 2 torque limit skipping instruction (%)	100

Setting range: 1～200

Appendix

0349	Torque limit ratio of axis 3 torque limit skipping instruction (%)	100

Setting range: 1～200

0350	Torque limit ratio of axis 4 torque limit skipping instruction (%)	100

Setting range: 1～200

0351	Torque limit ratio of axis 5 torque limit skipping instruction (%)	100

Setting range: 1～200

0352	Zero returning high speed acceleration/deceleration time constant	100

Setting range: 3～400

0353	Zero returning low speed acceleration/deceleration time constant	30

Setting range: 3～400

0354	Zero returning stop return to the low speed of machine tool zero point	200

Setting range: 0～999999

0355	Successful system booting times	0

Setting range: 0～999999

0356	Number of machined parts	0

Setting range: 0~9999

0357	Total number of parts to be machined	0

Setting range: 0~9999

0358	Accumulative power-on hours	0

Setting range: 0～99999

0359	Accumulative power-on days	0

Setting range: 0～99999

0360	Accumulative cutting hours	0

Setting range: 0～99999

0361	Coordinate axis of axis Z ATC tool change point at mechanical coordinate system	0

Setting range: 0～99999

0362	Axis Z G0 quick positioning speed (ATC)	5000

Setting range: 0～300000

0371	Axis 1 reverse positioning allowance	0.0150

Setting range: 0～99.9999 (mm)

0372	Axis 2 reverse positioning allowance	0.0150

Setting range: 0～99.9999 (mm)

0373	Axis 3 reverse positioning allowance	0.0150

Setting range: 0～99.9999 (mm)

0374	Axis 4 reverse positioning allowance	0.0150

Setting range: 0～99.9999 (mm)

0375	Axis 5 reverse positioning allowance	0.0150

Setting range: 0～99.9999 (mm)

When reverse backlash compensation set for certain axis (P0190---P0193) is larger than reverse positioning allowance set for the same axis (P0371---P0374), one single-node endpoint speed should be reduced to the lowest speed before reverse backlash compensation for such axis to limit movements of other axes within compensation period and ensure small deviation of synthetic profile from actual profile.

0376	Order of each axis moving to program restart position	12345

Setting range: 0～99999

0379	Set main control axis (0-50) 0: out of sync; 1: axis X; 2: axis Y; 3: axis Z; 4: 4^{th}; 5: 5^{th}	0

Setting range: 0～5

0380	Set main control axis (0-50) 0: out of sync; 1: axis X; 2: axis Y; 3: axis Z; 4: 4^{th}; 5: 5^{th}	0

Setting range: 0～5

0381	Maximum allowed error between synchronizing axes	200

Setting range: 0～10000

0382	Set double-drive reference position difference	0.0000

Setting range: 0.0000～2000.0000

0383	Maximum length of double-drive correction	1.0000

Setting range: 1.0000～2.0000

0384	Incremental distance between two Rs of grating (mm)	0.0010.

Setting range: 1.0000～1.0000

0385	Grating correction accuracy (mm)	0.0000.

Setting range: 1.0000～1.0000

Appendix

0387	Call M code of subprogram 9001	0.0000.

Setting range: 0~65535

0388	Call M code of user macroprogram 9020	0.0000.

Setting range: 0~65535

0389	Call M code of user macroprogram 9021	0.0000.

Setting range: 0~65535

0392	Moving distance during servo optimization	50

Setting range: 0～100

0393	Moving rate during servo optimization	2000

Setting range: 0～5000

0394	Backup of axis 1 in coordinate system	0

Setting range: -9999.9999～9999.9999

0395	Backup of axis 2 in coordinate system	0

Setting range: -9999.9999～9999.9999

0396	Backup of axis 3 in coordinate system	0

Setting range: -9999.9999～9999.9999

0397	Backup of axis 4 in coordinate system	0

Setting range: -9999.9999～9999.9999

0398	Backup of axis 5 in coordinate system	0

Setting range: -9999.9999～9999.9999

0399	Interpolation step size multiple	1.5

Setting range: 1.0000～10.0000

0400	Shape matching parameter	20

Setting range: 0.0020～99.0000
Shape matching parameter (#400) analyzes shape error, optimize shape and keep error within allowed range based on initial spline.
The larger the parameter, the larger the error. The smaller the parameter, the smaller the error.

0401	Shape matching limit	15

Setting range: 1.0000～999.000

Shape matching limit parameter (#401) limits increase in shape error due to curvature optimization during speed matching calculation.

0402	Speed matching parameter	1

Appendix

Setting range: 0.0020～99.0000

Speed matching parameter (#402) optimizes curvature to smooth speed with curvature radiating radially along normal direction of every point on the curve.
The larger the parameter, the lower the curvature optimization degree, the larger the accelerated speed and the shorter the machining time.
The smaller the parameter, the high the curvature optimization degree and the longer the machining time.

0403	Small line fitting segment number	7

Setting range: 0.0020～999.0000

This parameter (#403) determines the number of cutter location points of the spline curve to be fitted. Such parameter should be kept within certain range.
#403 = 1~10 The larger the parameter, the larger the calculated amount and smaller the shape error.
The smaller the parameter, the smaller the calculated amount and the larger the shape error.

0404	Spline coefficient n1	30

Setting range: 1.0000～199.0000

0405	Spline coefficient n2	30

Setting range: 1.0000～199.0000

0406	Spline coefficient n3	30

Setting range: 1.0000～199.0000

Fit a initial cubic spline curve based on spline coefficients n1, n2 and n3 (#404, #405 and #406). The larger the spline coefficients n1 and n2 (#404 and #405), the larger the curve error. But speed is smooth and machine tool runs stably. The smaller the coefficient, the smaller the curve error. But speed is not smooth and machine tool vibrates. Spline coefficient n3 (#406) is the opposite.

0407	High-speed interpolation cutting feed accelerated speed upper limit	0.6000

Setting range: 0.0020～99.0000

0408	High-speed interpolation cutting feed accelerated speed lower limit	0.6000

Setting range: 0.0020～99.0000

0409	Pre-read smoothing control	2.0000

Setting range: 0.0000～30.0000

Pre-read smoothing control parameter (#409) pre-reads the shape to be machined in advance and automatically calculate trend of overall shape in order to reduce machining scars due to program error caused by CAM.
0: turn off pre-read smoothing function
1: smooth based on length
2: smooth based on comprehensive relation between length and angle

0410	Accuracy-smoothness balance coefficient	6.0000

Setting range: 0.0000～10.0000

To enable high-accuracy control, users only need to set accuracy-smoothness balance coefficient (#410) which controls level of machining effect. There are 11 effect levels ranging from 0 to 10:
#410 = 0 : Indicates high-accuracy control and strict control of positioning accuracy with no requirement for smoothness; effective for programs with detailed requirements of corner angles during machining.
=1-10: returning to high-speed and high-accuracy control. The lower the level, the higher the accuracy; the higher the level, the higher the smoothness.
Ideal effect can be obtained by adjusting this parameter based on actual machining situation.

0411	Spline shape control factor	10.0000

Setting range: 0.0000～10.0000

0412	Small line fitting accuracy control	-1.0000

Setting range: -10.0000～50.0000

0413	Roundness and smoothness control factor n1	3.0000

Setting range: 0.0000～50.0000

0414	Roundness and smoothness control factor n2	0.0000

Setting range: 0.0000～50.0000

0420	Max speed in straight line	0.0000

Setting range: 0.0000～0.0000

0421	Highest rotating axis speed	0.0000

Setting range: 0.0000～0.0000

0422	Rotating axis shape matching	0.0000

Setting range: 0.0000～0.0000

0423	Fitting segment number of rotating axis subline	0.0000

Setting range: 0.0000～0.0000

0424	Internal system parameter 3	0.0000

Setting range: 0.0000～0.0000

0425	Internal system parameter 4	0.0000

Setting range: 0.0000～0.0000

0426	Central bias of rotating disk	0.0000

Setting range: 0.0000～0.0000

0427	Central bias of rotating disk	0.0000

Setting range: 0.0000～0.0000

0428	Azimuth direction	0.0000

Setting range: 0.0000～0.0000

0429	Elevation direction	0.0000

Setting range: 0.0000～0.0000

0430	Workpiece offset height	0.0000

Setting range: 0.0000～0.0000

0440	M code for activating Cs contour control axis function	0

Setting range: 0~999

0441	M code for deactivating Cs contour control axis function	0

Setting range: 0~999

0444	Maximum allowed offset of machine tool coordinate and absolute encoder position of each axis	50

Setting range: 0～500

0445	Axis 1 configuration grating accuracy	0.0010

Setting range: 0~10

0446	Axis 2 configuration grating accuracy	0.0010

Setting range: 0~10

0447	Axis 3 configuration grating accuracy	0.0010

Setting range: 0~10

0448	Axis 4 configuration grating accuracy	0.0010

Setting range: 0~10

0449	Axis 5 configuration grating accuracy	0.0010

Setting range: 0~10

0451	Measure feeding speed during automatic measurement (M series of tool length (for XAE1 signal)	100

Setting range: 0～9000

0452	Measure feeding speed during automatic measurement (M series of tool length (for XAE2 signal)	100

Setting range: 0～9000

0453	Measure feeding speed during automatic measurement (M series of tool length (for XAE3 signal)	100

Setting range: 0～9000

0454	γ value of automatic measurement of tool length (M series) (for XAE1 signal)	0.000

Setting range: -9999.0000～9999.9999

0455	γ value of automatic measurement of tool length (M series) (for XAE2 signal)	0.000

Setting range: -9999.0000～9999.9999

0456	γ value of automatic measurement of tool length (M series) (for XAE3 signal)	0.000

Setting range: -9999.0000～9999.9999

0457	ε value of automatic measurement of tool length (M series) (for XAE1 signal)	0.000

Setting range: -9999.0000～9999.9999

0458	ε value of automatic measurement of tool length (M series) (for XAE1 signal)	0.000

Setting range: -9999.0000～9999.9999

0459	ε value of automatic measurement of tool length (M series) (for XAE1 signal)	0.000

Setting range: -9999.0000～9999.9999

0463	Minimum dividing angle of axis 4 dividing axis	0.0000

Setting range: 0.0000～0.0000

0464	Minimum dividing angle of axis 5 dividing axis	0.0000

Setting range: 0.0000～0.0000

0480	Control axis executing position 01 switching function	0

Setting range: 0～5

0481	Control axis executing position 02 switching function	0

Setting range: 0～5

0482	Control axis executing position 03 switching function	0

Setting range: 0～5

0483	Control axis executing position 04 switching function	0

Setting range: 0～5

0484	Control axis executing position 05 switching function	0

Setting range: 0～5

Appendix

0485	Control axis executing position 06 switching function	0

Setting range: 0～5

0486	Control axis executing position 07 switching function	0

Setting range: 0～5

0487	Control axis executing position 08 switching function	0

Setting range: 0～5

0488	Control axis executing position 09 switching function	0

Setting range: 0～5

0489	Control axis executing position 10 switching function	0

Setting range: 0～5

0490	Control axis executing position 11 switching function	0

Setting range: 0～5

0491	Control axis executing position 12 switching function	0

Setting range: 0～5

0492	Control axis executing position 13 switching function	0

Setting range: 0～5

0493	Control axis executing position 14 switching function	0

Setting range: 0～5

0494	Control axis executing position 15 switching function	0

Setting range: 0～5

0495	Control axis executing position 16 switching function	0

Setting range: 0～5

0500	Maximum value of position 01 switching action range	0.0000

Setting range: -9999.9999～9999.9999

0501	Maximum value of position 02 switching action range	0.0000

Setting range: -9999.9999～9999.9999

0502	Maximum value of position 03 switching action range	0.0000

Setting range: -9999.9999～9999.9999

0503	Maximum value of position 04 switching action range	0.0000

Setting range: -9999.9999～9999.9999

0504	Maximum value of position 05 switching action range	0.0000

Setting range: -9999.9999～9999.9999

0505	Maximum value of position 06 switching action range	0.0000

Setting range: -9999.9999～9999.9999

0506	Maximum value of position 07 switching action range	0.0000

Setting range: -9999.9999～9999.9999

0507	Maximum value of position 08 switching action range	0.0000

Setting range: -9999.9999～9999.9999

0508	Maximum value of position 09 switching action range	0.0000

Setting range: -9999.9999～9999.9999

0509	Maximum value of position 10 switching action range	0.0000

Setting range: -9999.9999～9999.9999

0510	Maximum value of position 11 switching action range	0.0000

Setting range: -9999.9999～9999.9999

0511	Maximum value of position 12 switching action range	0.0000

Setting range: -9999.9999～9999.9999

0512	Maximum value of position 13 switching action range	0.0000

Setting range: -9999.9999～9999.9999

0513	Maximum value of position 14 switching action range	0.0000

Setting range: -9999.9999～9999.9999

0514	Maximum value of position 15 switching action range	0.0000

Setting range: -9999.9999～9999.9999

0515	Maximum value of position 16 switching action range	0.0000

Setting range: -9999.9999～9999.9999

0520	Minimum value of position 01 switching action range	0.0000

Setting range: -9999.9999～9999.9999

0521	Minimum value of position 02 switching action range	0.0000

Setting range: -9999.9999～9999.9999

0522	Minimum value of position 03 switching action range	0.0000

Setting range: -9999.9999～9999.9999

0523	Minimum value of position 04 switching action range	0.0000

Setting range: -9999.9999～9999.9999

0524	Minimum value of position 05 switching action range	0.0000

Setting range: -9999.9999～9999.9999

0525	Minimum value of position 06 switching action range	0.0000

Setting range: -9999.9999～9999.9999

0526	Minimum value of position 07 switching action range	0.0000

Setting range: -9999.9999～9999.9999

0527	Minimum value of position 08 switching action range	0.0000

Setting range: -9999.9999～9999.9999

0528	Minimum value of position 09 switching action range	0.0000

Setting range: -9999.9999～9999.9999

0529	Minimum value of position 10 switching action range	0.0000

Setting range: -9999.9999～9999.9999

0530	Minimum value of position 11 switching action range	0.0000

Setting range: -9999.9999～9999.9999

0531	Minimum value of position 12 switching action range	0.0000

Setting range: -9999.9999～9999.9999

0532	Minimum value of position 13 switching action range	0.0000

Setting range: -9999.9999～9999.9999

0533	Minimum value of position 14 switching action range	0.0000

Setting range: -9999.9999～9999.9999

0534	Minimum value of position 15 switching action range	0.0000

Setting range: -9999.9999～9999.9999

0535	Minimum value of position 16 switching action range	0.0000

Setting range: -9999.9999～9999.9999

0540	Call starting code of code G of user macroprogram	101

Setting range: 0~9999

0541	Starting program number of user macrogrogram called via code G	89101

Setting range: 0～99999

0542	Number of code G of user macroprogram called	100

Setting range: 0~9999

0555	Positional deviation limit value during movement of axis 1 (mm)	50.0000

Setting range: 0.000～99999.9999

Appendix

0556	Positional deviation limit value during movement of axis 2 (mm)	50.0000

Setting range: 0.000～99999.9999

0557	Positional deviation limit value during movement of axis 3 (mm)	50.0000

Setting range: 0.000～99999.9999

0558	Positional deviation limit value during movement of axis 4 (mm)	10.0000

Setting range: 0.000～99999.9999

0559	Positional deviation limit value during movement of axis 5 (mm)	10.0000

Setting range: 0.000～99999.9999

0560	Positional deviation limit value during movement of axis 1 (mm)	50.0000

Setting range: 0.000～99999.9999

0560	Positional deviation limit value during movement of axis 1 (mm)	50.0000

Setting range: 0.000～99999.9999

0561	Positional deviation limit value during movement of axis 2 (mm)	50.0000

Setting range: 0.000～99999.9999

0562	Positional deviation limit value during movement of axis 3 (mm)	50.0000

Setting range: 0.000～99999.9999

0563	Positional deviation limit value during movement of axis 4 (mm)	0.5000

Setting range: 0.000～99999.9999

0564	Positional deviation limit value during movement of axis 5 (mm)	0.5000

Setting range: 0.000～99999.9999

600	Number of teeth of main axis side gear of axis 1 (first-speed gear)	1

Setting range: 1～999

601	Number of teeth of main axis side gear of axis 2 (first-speed gear)	1

Setting range: 1～999

602	Number of teeth of main axis side gear of axis 3 (first-speed gear)	1

Setting range: 1～999

603	Number of teeth of main axis side gear of axis 4 (first-speed gear)	1

Setting range: 1～999

604	Number of teeth of main axis side gear of axis 1 (second-speed gear)	1

Setting range: 1～999

605	Number of teeth of main axis side gear of axis 2 (second-speed gear)	1

Setting range: 1～999

606	Number of teeth of main axis side gear of axis 3 (second-speed gear)	1

Setting range: 1～999

607	Number of teeth of main axis side gear of axis 4 (second-speed gear)	1

Setting range: 1～999

608	Number of teeth of main axis side gear of axis 1 (third-speed gear)	1

Setting range: 1～999

609	Number of teeth of main axis side gear of axis 2 (third-speed gear)	1

Setting range: 1～999

610	Number of teeth of main axis side gear of axis 3 (third-speed gear)	1

Setting range: 1～999

611	Number of teeth of main axis side gear of axis 4 (third-speed gear)	1

Setting range: 1～999

612	Number of teeth of main axis side gear of axis 1 (fourth-speed gear)	1

Setting range: 1～999

613	Number of teeth of main axis side gear of axis 2 (fourth-speed gear)	1

Setting range: 1～999

614	Number of teeth of main axis side gear of axis 3 (fourth-speed gear)	1

Setting range: 1～999

615	Number of teeth of spindle side gear of 4th spindle (fourth-speed gear)	1

Setting range: 1～999

616	Number of teeth of position encoder side gear of axis 1 (first-speed gear)	1

Setting range: 1～999

Appendix

617	Number of teeth of position encoder side gear of axis 2 (first-speed gear)	1

Setting range: 1～999

618	Number of teeth of position encoder side gear of axis 3 (first-speed gear)	1

Setting range: 1～999

619	Number of teeth of position encoder side gear of axis 4 (first-speed gear)	1

Setting range: 1～999

620	Number of teeth of position encoder side gear of axis 1 (second-speed gear)	1

Setting range: 1～999

621	Number of teeth of position encoder side gear of axis 2 (second-speed gear)	1

Setting range: 1～999

622	Number of teeth of position encoder side gear of axis 3 (second-speed gear)	1

Setting range: 1～999

623	Number of teeth of position encoder side gear of axis 4 (second-speed gear)	1

Setting range: 1～999

624	Number of teeth of position encoder side gear of axis 1 (third-speed gear)	1

Setting range: 1～999

625	Number of teeth of position encoder side gear of axis 2 (third-speed gear)	1

Setting range: 1～999

626	Number of teeth of position encoder side gear of axis 3 (third-speed gear)	1

Setting range: 1～999

627	Number of teeth of position encoder side gear of axis 4 (third-speed gear)	1

Setting range: 1～999

628	Number of teeth of position encoder side gear of axis 1 (fourth-speed gear)	1

Setting range: 1～999

629	Number of teeth of position encoder side gear of axis 2 (fourth-speed gear)	1

Setting range: 1～999

630	Number of teeth of position encoder side gear of axis 3 (fourth-speed gear)	1

Setting range: 1～999

631	Number of teeth of position encoder side gear of axis 4 (fourth-speed gear)	1

Setting range: 1～999

632	Highest speed of main axis during rigid tapping of axis 1 (first-speed gear)	6000

Setting range: 1～9999

633	Highest speed of main axis during rigid tapping of axis 2 (first-speed gear)	6000

Setting range: 1～9999

634	Highest speed of main axis during rigid tapping of axis 3 (first-speed gear)	6000

Setting range: 1～9999

635	Highest speed of main axis during rigid tapping of axis 4 (first-speed gear)	6000

Setting range: 1～9999

636	Highest speed of main axis during rigid tapping of axis 1 (second-speed gear)	6000

Setting range: 1～9999

637	Highest speed of main axis during rigid tapping of axis 2 (second-speed gear)	6000

Setting range: 1～9999

638	Highest speed of main axis during rigid tapping of axis 3 (second-speed gear)	6000

Setting range: 1～9999

639	Highest speed of main axis during rigid tapping of axis 4 (second-speed gear)	6000

Setting range: 1～9999

640	Highest speed of main axis during rigid tapping of axis 1 (third-speed gear)	6000

Setting range: 1～9999

641	Highest speed of main axis during rigid tapping of axis 2 (third-speed gear)	6000

Setting range: 1～9999

642	Highest speed of main axis during rigid tapping of axis 3 (third-speed gear)	6000

Setting range: 1～9999

643	Highest speed of main axis during rigid tapping of axis 4 (third-speed gear)	6000

Setting range: 1～9999

644	Highest speed of main axis during rigid tapping of	6000

	axis 1 (fourth-speed gear)	

Setting range: 1～9999

645	Highest speed of main axis during rigid tapping of axis 2 (fourth-speed gear)	6000

Setting range: 1～9999

646	Highest speed of main axis during rigid tapping of axis 3 (fourth-speed gear)	6000

Setting range: 1～9999

647	4th spindle maximum rotation during rigid tapping (fourth-speed gear)	6000

Setting range: 1～9999

648	Linear acceleration/deceleration time constant of main axis and tapping axis of main axis 1 (first-speed gear)	200

Setting range: 1～9999

649	Linear acceleration/deceleration time constant of main axis and tapping axis of main axis 2 (first-speed gear)	200

Setting range: 1～9999

650	Linear acceleration/deceleration time constant of main axis and tapping axis of main axis 3 (first-speed gear)	200

Setting range: 1～9999

651	Linear acceleration/deceleration time constant of main axis and tapping axis of main axis 4 (first-speed gear)	200

Setting range: 1～9999

652	Linear acceleration/deceleration time constant of main axis and tapping axis of main axis 1 (second-speed gear)	200

Setting range: 1～9999

653	Linear acceleration/deceleration time constant of main axis and tapping axis of main axis 2 (second-speed gear)	200

Setting range: 1～9999

654	Linear acceleration/deceleration time constant of main axis and tapping axis of main axis 3 (second-speed gear)	200

Setting range: 1～9999

655	Linear acceleration/deceleration time constant of main axis and tapping axis of main axis 4 (second-speed gear)	200

Setting range: 1～9999

656	Linear acceleration/deceleration time constant of main axis and tapping axis of main axis 1 (third-speed gear)	200

Setting range: 1～9999

657	Linear acceleration/deceleration time constant of main axis and tapping axis of main axis 2 (third-speed gear)	200

Setting range: 1～9999

658	Linear acceleration/deceleration time constant of main axis and tapping axis of main axis 3 (third-speed gear)	200

Setting range: 1～9999

659	Linear acceleration/deceleration time constant of main axis and tapping axis of main axis 4 (third-speed gear)	200

Setting range: 1～9999

660	Linear acceleration/deceleration time constant of main axis and tapping axis of main axis 1 (fourth-speed gear)	200

Setting range: 1～9999

661	Linear acceleration/deceleration time constant of main axis and tapping axis of main axis 2 (fourth-speed gear)	200

Setting range: 1～9999

662	Linear acceleration/deceleration time constant of main axis and tapping axis of main axis 3 (fourth-speed gear)	200

Setting range: 1～9999

663	Linear acceleration/deceleration time constant of main axis and tapping axis of main axis 4 (fourth-speed gear)	200

Setting range: 1～9999

664	Time constant of main axis and tapping axis during tool retraction of main axis 1 (first-speed gear)	200

Setting range: 1～9999

665	Time constant of main axis and tapping axis during tool retraction of main axis 2 (first-speed gear)	200

Setting range: 1～9999

666	Time constant of main axis and tapping axis during tool retraction of main axis 3 (first-speed gear)	200

Setting range: 1～9999

667	Time constant of main axis and tapping axis during tool retraction of main axis 4 (first-speed gear)	200

Setting range: 1～9999

668	Time constant of main axis and tapping axis during tool retraction of main axis 1 (second-speed gear)	200

Setting range: 1～9999

669	Time constant of main axis and tapping axis during tool retraction of main axis 2 (second-speed gear)	200

Setting range: 1～9999

670	Time constant of main axis and tapping axis during tool retraction of main axis 3 (second-speed gear)	200

Setting range: 1～9999

677	Time constant of main axis and tapping axis during tool retraction of main axis 4 (second-speed gear)	200

Setting range: 1～9999

672	Time constant of main axis and tapping axis during tool retraction of main axis 1 (third-speed gear)	200

Setting range: 1～9999

673	Time constant of main axis and tapping axis during tool retraction of main axis 2 (third-speed gear)	200

Setting range: 1～9999

674	Time constant of main axis and tapping axis during tool retraction of main axis 3 (third-speed gear)	200

Setting range: 1～9999

675	Time constant of main axis and tapping axis during tool retraction of main axis 4 (third-speed gear)	200

Setting range: 1～9999

676	Time constant of main axis and tapping axis during tool retraction of main axis 1 (fourth-speed gear)	200

Setting range: 1～9999

677	Time constant of main axis and tapping axis during tool retraction of main axis 2 (fourth-speed gear)	200

Setting range: 1～9999

678	Time constant of main axis and tapping axis during tool retraction of main axis 3 (fourth-speed gear)	200

Setting range: 1～9999

679	Time constant of main axis and tapping axis during tool retraction of main axis 4 (fourth-speed gear)	200

Setting range: 1～9999

680	Backlash of rigid tapping main axis of axis 1 (first-speed gear)	0.000

Setting range: 1～9999

681	Backlash of rigid tapping main axis of axis 2 (first-speed gear)	0.000

Setting range: 1～9999

682	Backlash of rigid tapping main axis of axis 3 (first-speed gear)	0.000

Setting range: 1～9999

683	Backlash of rigid tapping main axis of axis 4 (first-speed gear)	0.000

Setting range: 1～9999

684	Backlash of rigid tapping main axis of axis 1 (second-speed gear)	0.000

Setting range: 1～9999

685	Backlash of rigid tapping main axis of axis 2 (second-speed gear)	0.000

Setting range: 1～9999

686	Backlash of rigid tapping main axis of axis 3 (second-speed gear)	0.000

Setting range: 1～9999

687	Backlash of rigid tapping main axis of axis 4 (second-speed gear)	0.000

Setting range: 1～9999

688	Backlash of rigid tapping main axis of axis 1 (third-speed gear)	0.000

Setting range: 1～9999

689	Backlash of rigid tapping main axis of axis 2 (third-speed gear)	0.000

Setting range: 1～9999

690	Backlash of rigid tapping main axis of axis 3 (third-speed gear)	0.000

Setting range: 1～9999

691	Backlash of rigid tapping main axis of axis 4 (third-speed gear)	0.000

Setting range: 1～9999

692	Backlash of rigid tapping main axis of axis 1 (fourth-speed gear)	0.000

Setting range: 1～9999

693	Backlash of rigid tapping main axis of axis 2 (fourth-speed gear)	0.000

Setting range: 1～9999

694	Backlash of rigid tapping main axis of axis 3 (fourth-speed gear)	0.000

Setting range: 1～9999

695	Backlash of rigid tapping main axis of axis 4 (fourth-speed gear)	0.000

Setting range: 1～9999

696	Main axis multiplier factor (CMR) of main axis 1 (first-speed gear)	512

Setting range: 1～9999

697	Main axis multiplier factor (CMR) of main axis 2 (first-speed gear)	512

Setting range: 1～9999

698	Main axis multiplier factor (CMR) of main axis 3 (first-speed gear)	512

Setting range: 1～9999

Appendix

699	Main axis multiplier factor (CMR) of main axis 4 (first-speed gear)	512

Setting range: 1～9999

700	Main axis multiplier factor (CMR) of main axis 1 (second-speed gear)	512

Setting range: 1～9999

701	Main axis multiplier factor (CMR) of main axis 2 (second-speed gear)	512

Setting range: 1～9999

702	Main axis multiplier factor (CMR) of main axis 3 (second-speed gear)	512

Setting range: 1～9999

703	Main axis multiplier factor (CMR) of main axis 4 (second-speed gear)	512

Setting range: 1～9999

704	Main axis multiplier factor (CMR) of main axis 1 (third-speed gear)	512

Setting range: 1～9999

705	Main axis multiplier factor (CMR) of main axis 2 (third-speed gear)	512

Setting range: 1～9999

706	Main axis multiplier factor (CMR) of main axis 3 (third-speed gear)	512

Setting range: 1～9999

707	Main axis multiplier factor (CMR) of main axis 4 (third-speed gear)	512

Setting range: 1～9999

708	Main axis multiplier factor (CMR) of main axis 1 (fourth-speed gear)	512

Setting range: 1～9999

709	Main axis multiplier factor (CMR) of main axis 2 (fourth-speed gear)	512

Setting range: 1～9999

710	Main axis multiplier factor (CMR) of main axis 3 (fourth-speed gear)	512

Setting range: 1～9999

711	Main axis multiplier factor (CMR) of main axis 4 (fourth-speed gear)	512

Setting range: 1～9999

712	Main axis multiplier factor (CMR) of main axis 1 (first-speed gear)	125

Appendix

Setting range: 1～9999

713	Main axis multiplier factor (CMR) of main axis 2 (first-speed gear)	125

Setting range: 1～9999

714	Main axis multiplier factor (CMR) of main axis 3 (first-speed gear)	125

Setting range: 1～9999

715	Main axis multiplier factor (CMR) of main axis 4 (first-speed gear)	125

Setting range: 1～9999

716	Main axis multiplier factor (CMR) of main axis 1 (second-speed gear)	125

Setting range: 1～9999

717	Main axis multiplier factor (CMR) of main axis 2 (second-speed gear)	125

Setting range: 1～9999

718	Main axis multiplier factor (CMR) of main axis 3 (second-speed gear)	125

Setting range: 1～9999

719	Main axis multiplier factor (CMR) of main axis 4 (second-speed gear)	125

Setting range: 1～9999

720	Main axis multiplier factor (CMR) of main axis 1 (third-speed gear)	125

Setting range: 1～9999

721	Main axis multiplier factor (CMR) of main axis 2 (third-speed gear)	125

Setting range: 1～9999

722	Main axis multiplier factor (CMR) of main axis 3 (third-speed gear)	125

Setting range: 1～9999

723	Main axis multiplier factor (CMR) of main axis 4 (third-speed gear)	125

Setting range: 1～9999

724	Main axis multiplier factor (CMR) of main axis 1 (third-speed gear)	125

Setting range: 1～9999

725	Main axis multiplier factor (CMR) of main axis 2 (third-speed gear)	125

Setting range: 1～9999

726	Main axis multiplier factor (CMR) of main axis 3 (third-speed gear)	125

Setting range: 1～9999

727	Main axis multiplier factor (CMR) of main axis 4 (third-speed gear)	125

Setting range: 1～9999

800	System axis number of PMC axis control (0: none; 1~5: axes 1~5)	0

Setting range: 0～5

801	Smallest unit of PMC axis control data (0.0001~360.0)	0.1000

Setting range: 0.0001～360.0000

802	Acceleration/deceleration time in PMC axis control speed instruction	500

Setting range: 10～99999

805	System axis number of PMC axis 2 control (0: none; 1~5: axes 1~5)	0

Setting range: 0～5

806	Smallest unit of PMC axis 2 control data (0.0001~360.0)	0.1000

Setting range: 0.0001～360.0000

807	Acceleration/deceleration time in PMC axis 2 control speed instruction	500

Setting range: 10～99999

810	System axis number of PMC axis 3 control (0: none; 1~5: axes 1~5)	0

Setting range: 0～5

811	Smallest unit of PMC axis 3 control data (0.0001~360.0)	0.1000

Setting range: 0.0001～360.0000

812	Acceleration/deceleration time in PMC axis 3 control speed instruction	500

Setting range: 10～99999

815	System axis number of PMC axis 4 control (0: none; 1~5: axes 1~5)	0

Setting range: 0～5

816	Smallest unit of PMC axis 4 control data (0.0001~360.0)	0.1000

Setting range: 0.0001～360.0000

817	Acceleration/deceleration time in PMC axis 4 control speed instruction	500

Setting range: 10～99999

850	Set axis 1 as the axis in fundamental coordinate system (0~7)	1

Setting range: 0～7

851	Set axis 2 as the axis in fundamental coordinate system (0~7)	2

Setting range: 0～7

852	Set axis 3 as the axis in fundamental coordinate system (0~7)	3

Setting range: 0～7

853	Set axis 4 as the axis in fundamental coordinate system (0~7)	4

Setting range: 0～7

854	Set axis 5 as the axis in fundamental coordinate system (0~7)	5

Setting range: 0～7

Appendix II Table of Alarms

Alarm number	Content	Remark
0000	Parameter to cut off primary power supply is modified	
0001	Failed to open file	
0002	Data entered is beyond range	
0003	Copied or renamed program number already exists	
0004	There is no address at the beginning of program segment and a number or symbol “-“ is entered. Modify program.	
0005	There is another address or EOB code rather than suitable data behind the address Modify program.	
0006	Symbol “-“ is incorrectly entered (symbol “-“ is entered behind addresses where minus sign cannot be used or two or more “-“ symbols are entered). Modify program.	
0007	Decimal point "." Input error (symbol “.” is used in addresses where such symbol is not allowed to be used or two or more “.” symbols are used). Modify program.	
0008	Program file is too large. Please use NDC to transmit	
0009	The address entered is illegal. Modify program.	
0010	Code G that cannot be used is use or code G without such function is instructed. Modify program.	
0011	Feeding speed instruction is not given or feeding speed is not appropriate during cutting feed. Modify program.	
0012	Low disk space. New file cannot be created or new file content cannot be added	
0013	Program file number has reached upper limit. No new program can be created.	
0014	No instruction can be given concerning G95 since the main axis does not support	
0015	Number of axes under simultaneous control exceeds allowed range	
0016	Current screw pitch error compensation point exceed the range	
0017	Authority is not enough to modify. Go to password page to enter password.	
0018	Null and local variables are not allowed to be modified. G10 can only modify user-level parameters.	
0019	Zoom function is not available. Please modify position parameter 60.5 to turn on the function if required.	
0020	During arc interpolation (G02 or G03), distance between starting point and arc center is different from distance between destination and arc center and the difference is larger than the value designated in data parameter 214.	
0021	During arc interpolation, axis out of the selected plane (G17, G18 and G19) is instructed. Modify program.	
0022	During arc interpolation, either R (designated arc radius), I, K or K (designated distance from starting point to center) is instructed.	
0023	During arc interpolation, I, J, K and R are instructed at the same time.	
0024	During helix interpolation, helix angle is zero; system is not processed and program modification is required.	
0025	G12 cannot be at the same segment with other G instructions.	
0026	File format is not supported by the system. File is too large or certain line has more than	

Appendix

Alarm number	Content	Remark
	1024 bytes.	
0027	Length cutter compensation instruction cannot be at the same segment with G92. Modify program.	
0028	Among plane selection instructions, two or more axes are instructed in the same direction. Modify program.	
0029	Compensation value designated by D/H code is too large. Modify program.	
0030	Cutter length compensation number or cutter radius compensation number designated by D/H code is too large. Besides, workpiece coordinate system number designated by code P is too large. Modify program.	
0031	P value instructed is too large or is not designated when setting offset value, working coordinate system, external workpiece coordinate system and additional workpiece coordinate system by G10.	
0032	Offset value is too large or is not designated when setting offset value by G10 or entering offset value via system variables. Modify program.	
0033	Points of intersection in cutter compensation C or chamfering cannot be determined. Modify program.	
0034	Cutter compensation cannot be established or cancelled when giving arc instruction. Modify program.	
0036	In cutter compensation mode, skipped cutting is instructed (G31). Modify program.	
0037	Plane selected by G17, G18 or G19 is changed in cutter compensation C. Modify program.	
0038	In cutter compensation C, overcut will occur since arc starting point or destination is consistent with arc center. Modify program.	
0039	Tool nose error in cutter compensation C	
0040	Workpiece coordinate system cannot be changed in cutter compensation C. Please cancel cutter compensation first before changing coordinate system.	
0041	There is interference in cutter compensation C and overcut will occur. Modify program.	
0042	In cutter compensation mode, over ten program segments with only tool stopping instruction and no movement instruction is given continuously. Modify program.	
0043	Authority is not enough. Authority can be altered in password screen.	
0044	In fixed cycle mode, one of G27, G28, G29 and G30 is instructed. Modify program.	
0045	In fixed cycle G73/G83, cutting depth is not designated or Q value is 0. Modify program.	
0046	In return instructions of reference points 2, 3 and 4, instructions beyond P2, P3 and P4 are given.	
0047	Execute machine tool zeroing before executing G28, G30 and G53 instructions.	
0048	In fixed cycle, plane Z should be higher than plane R	
0049	In fixed cycle, plane Z should not be higher than plane R	
0050	Move when changing fixed cycle mode	
0051	Wrong movement action or distance is designated in program segments following rounding or bevelling. Modify program.	
0052	Mirror function cannot be used in milling groove fixed cycle. Modify program	
0053	Wrong instruction format of bevelling or rounding. Modify program	
0054	DNC sending error	
0055	Chamfering movement cannot be completed	

Appendix

Alarm number	Content	Remark
0056	M99 cannot be at the same segment with macroprogram instruction (G65) Modify program.	
0057	Failed to write in file. Power should be cut to reboot.	
0058	In any bevelling or rounding program segment, designated axis is not inside selected plane. Modify program.	
0059	In retrieval of external program number, program number is not found or designated program is edited in the background. Search program number and external signal or suspend background editing	
0060	No instruction sequence number is found in the search of sequence number. Check sequence number	
0061	Axis 1 is not at reference point.	
0062	Axis 2 is not at reference point.	
0063	Axis 3 is not at reference point.	
0064	Axis 4 is not at reference point.	
0065	Axis 5 is not at reference point.	
0066	Fixed cycle mode should be cancelled before executing parameter entering (G10)	
0067	Setting format not supported by G10	
0068	Parameter switch is not turned on	
0069	Machining operation requires turning off USB flash disk operation interface	
0070	Insufficient memory. Delete unnecessary program and try again	
0071	Address searched is not found, or no program with designated number is found during program retrieval Inspection data	
0072	Number of program stored exceeds 400. Delete unwanted program.	
0073	Program number instructed has been used. Change program number or delete unwanted program.	
0074	Program number is a number beyond 1~99999. Change program number	
0075	An attempt has been made to register a protected program number	
0076	Address P is not instructed in M98 program segment (program number). Modify program.	
0077	Over five programs have been called. Modify program.	
0078	In M98 and G65 program segments, program number designated by address P is not found or macroprogram called by M06 does not exist.	
0079	System service time expires. Please contact the supplier.	
0080	Data entered is not reasonable. The highest speed is lower than the lowest speed or the lowest speed is higher than the highest speed.	
0081	Macroprogram cannot call subprogram. Modify program.	
0084	Key timeout or short circuit.	
0085	Overflow occurs when entering data into storage using serial port. Baud rate setting or I/O device is not correct	
0086	In fixed cycle mode, the system cannot be switched to another plane	
0087	Alarm No. 0087~0091 indicate that returned starting point of axis reference point is too close to reference point or the speed is too slow to execute returning to reference point. Reference point should be distant enough from the starting point or appropriate speed should be designated for returning to reference point.	

Appendix

Alarm number	Content	Remark
0092	G27 (reference point returning check) instruction cannot return to reference point.	
0093	Motor model does not match	
0098	After powering on or emergency stop, or there is G28 in the program, rebooting is executed before returning to reference point.	
0100	On parameter (setup) screen, PWE (parameter writing effective) is set as 1. Set it as 0 and reboot the system	
0101	Memory data is in disorder after power cutoff. Please ensure position is correct.	
0102	System and drive have inconsistent motor model parameter.	
0103	Bus communication error. Please check Network cable reliability	
0104	Set machine tool zero-point timeout	
0105	Drive unit data acquisition timeout	
0106	Drive unit and system servo parameter have inconsistent gear ratio	
0107	Drive unit parameter is inconsistent with system servo parameter	
0108	Please insert USB flash disk	
0110	Position data exceeds allowed range Please return to zero	
0111	Calculation result falls outside allowed range ($-10^{47} \sim -10^{-29}$, 0 and $10^{-29} \sim 10^{47}$)	
0112	Zero (including tan90°) is designated as divisor	
0113	Function instructions cannot be used are designated in user macroprogram. Modify program.	
0114	G39 format error. Modify program.	
0115	Values as variables cannot be designated or O and N as variables are designated in user macroprogram. Modify program.	
0116	There is a variable on the left of assignment statement, but assignment is not allowed. Modify program.	
0117	This parameter does not support online modification of G10. Please modify program.	
0118	Number of parentheses exceeds upper limit (five pairs). Modify program.	
0119	M00, M01, M02, M30, M98, M99 and M06 instructions cannot be at the same segment with other M instructions	
0120	Part of setup is restored	
0121	Machine tool coordinates and encoder feedback value exceed set offset values	
0122	Over five layers of macroprogram are called. Modify program.	
0123	Transfer and loop statement cannot be used in MDI and DNC mode	
0124	Program is ended illegally without M30, M02 or M99 instruction or without terminator. Modify program.	
0125	Macroprogram format error. Modify program.	
0126	Program loop is false. Modify program.	
0127	NC and use macroprogram instruction statement coexist. Modify program.	
0128	Sequence number in branch instruction is not 0-99999 or is not found. Modify program.	
0129	<independent variable assignment> has incorrect address. Modify program.	
0130	PLC axis control instruction is output to axis controlled by CNC. Or CNC axis control instruction is outputted to axis controlled by PLC. Modify program.	

Alarm number	Content	Remark
0131	Five or more external alarm messages appear. Check ladder diagram	
0132	Alarm in external alarm message does not exist. Check PLC	
0133	Axis instruction not supported by the system. Modify program	
0135	Dividing angle of dividing table is not multiple of angle unit. Modify program.	
0136	Another axis is instructed together with axis B during dividing of dividing table. Modify program.	
0137	Sequence number to be transferred by skip instruction is within loop body. Modify program.	
0138	Loop statement mismatches or skip instruction enters loop body. Modify program.	
0139	PLC axis control selects wrong axis. Modify program.	
0140	Sequence number to which macro instruction skips does not exist	
0141	MDI model and DNC mode do not support macro instruction skipping	
0142	Scaling ratio beyond 1 to 999999 is instructed. Modify scaling ratio setting	
0143	Scaling result, moving distance, coordinate values and arc radius exceed maximum instructed values. Modify program or scaling ratio	
0144	Coordinates rotation plane must be the same with arch or tool radius compensation plane C. Modify program.	
0145	G28 instruction is given when reference point is not established. Please modify program or position parameter NO.4#3 (AZR)	
0148	Automatic corner deceleration speed is beyond determined angle range set value. Modify parameters	
0160	In polar coordinates mode, only R programming can be used for arc	
0161	In polar coordinates mode, no instruction related to reference point, plane selection or direction can be executed	
0163	In rotation mode, no G instruction related to reference point or coordinate system can be executed	
0164	In zoom mode, no G instruction related to reference point or coordinate system can be executed	
0165	Please designate rotation, zoom or G10 instruction within separate program segment	
0166	No axis is designated for returning to reference point	
0167	Coordinates of middle point are too large	
0168	Shortest pause time at hole bottom should be smaller than longest pause time at hole bottom	
0170	Tool radius compensation is not cancelled when entering or quitting subprogram	
0172	In program segments calling subprogram, P is not an integer or P is smaller than 0	
0173	Times of calling subprogram cannot be more than 9999	
0175	Fixed cycle can only be executed in G17 plane	
0176	Main axis rotation speed is not designated before rigid tapping	
0177	Under G76 instruction, I/O control does not support main axis orientation function. Modify program or parameter	
0178	Main axis rotation speed is not designated before fixed cycle	
0181	Illegal M code	
0182	Main axis rotation speed is too large or small.	

Appendix

Alarm number	Content	Remark
0183	Illegal T code	
0184	Tool selected is beyond range	
0185	L is too smaller and alarm is caused by: 1) L is smaller than tool radius in rectangular groove finish milling 2) L is smaller than 0 in rectangular groove finish milling	
0186	L is too large and alarm is caused by: 1) L is larger than tool radius in inner circle groove rough milling 2) L is larger than tool radius in rectangular groove rough milling 3) L is larger than I in rectangular groove finish milling 4) L is larger than J in rectangular groove finish milling	
0187	Tool diameter is too large and alarm is caused by: 1) Tool diameter is larger than I in inner circle groove rough milling 2) Tool diameter is larger than I-J in inner circle groove finish milling 3) Tool diameter is larger than J in outer circle groove finish milling 4) Tool diameter is larger than I in rectangular groove finish/rough milling 5) Tool diameter is larger than J in rectangular groove finish/rough milling 6) Tool diameter is larger than U in rectangular groove finish/rough milling 7) Helix entry radius factor is too large or D is too large. Modify value of data parameter No.269 or radius compensation value	
0188	U is too large and alarm is caused by: 1) Two times of U is larger than I in rectangular groove compound cycle 2) Two times of U is larger than J in rectangular groove compound cycle	
0189	U is too small when it should be larger than or equal to tool radius	
0190	V is too small or is not defined when it should be larger than 0	
0191	W is too small or is not defined when it should be larger than 0	
0192	Q is too small or is not defined when it should be larger than 0	
0193	I is not defined or is 0	
0194	J is not defined or is 0	
0195	D is not defined or is 0	
0198	During constant surface cutting speed control, designated axis is wrong. (see parameter No.254). Instruction P of designated axis has illegal data. Modify program.	
0199	Macro instruction is not defined to modify program	
0200	In rigid tapping, S value is beyond range or is not designated. In rigid tapping, maximum S value is designated via parameter. Change parameter setting or modify program	
0201	There is no F value in rigid tapping. Modify program.	
0202	In rigid tapping, allocated value of main axis is too large	
0203	In rigid tapping, M code (M29) or S instruction in program has incorrect position. Modify program.	
0204	M29 should be designated in G80 model. Modify program.	
0205	When M code (M29) or G84 (or 74) is instructed, rigid tapping signal is not 1. Check ladder diagram and locate cause.	
0206	Plane switching is instructed in rigid mode. Modify program.	
0207	Designated distance is too short or too long in rigid tapping	
0208	This instruction cannot be executed in G10 model. Please quit G10 model first	

Alarm number	Content	Remark
0209	Zoom rotation pole coordinate model does not support program rebooting	
0210	Program rebooting has inconsistent file name. Please select correct file name	
0212	Chamfering or R angle chamfering is instructed or there is an additional axis in the plane. Modify program.	
0213	Tool change macroprogram does not support G31 skipping. Modify program.	
0214	Tool change macroprogram does not support segment skipping	
0215	Tool change macroprogram does not support dynamic modification of coordinate system and cutter compensation	
0216	Zoom/rotation/pole coordinates do not support G31 skipping. Modify program.	
0217	Zoom/rotation/pole coordinates do not support segment skipping	
0218	Zoom/rotation/pole coordinates do not support dynamic modification of coordinate system and cutter compensation	
0219	Tool changer is not used (parameter is not enabled). Tool change instruction M06 cannot be used	
0220	Zoom/rotation/pole coordinates do not support switching between British system and metric system	
0221	Tool change macroprogram does not support switching between British system and metric system	
0224	Reference point returning is not executed before turning on automatic operation	
0231	The following errors occur in designated format of entering program parameters: 1) Address N or R is not entered; 2) Parameter number is not designated; 3) Address P is not designated in position parameter input L50; 4) N, P and R are beyond range. Modify program.	
0232	Three or more axes have been designated as helix interpolation axes	
0233	Other operations are using device connected to RS-232-C port	
0235	End-of-record terminator is instructed (%)	
0236	Program rebooting parameter is incorrectly set	
0237	Decimal point of instructions whose decimal point must be designated has not been instructed	
0238	One address appears for too many times within a program segment, or two or more G codes belong to the same group in a program segment	
0239	An illegal G code is designated in pre-read control mode. Dividing axis is instructed in pre-machining control mode; maximum cutting feed parameter is set as 0 and acceleration/deceleration parameter before interpolation is set as 0. Set parameters correctly	
0241	Manual pulse generator exception	
0242	Bus connection error	
0250	Same axis name has been used. Please modify parameters NO.175~179	
0251	E-stop alarm goes off. Please return to zero after cancel the alarm	
0252	Program is illegally ended (CNC transmission speed is slow. Please slow down feeding)	
0261	Pulse instruction speed of DSP interpolation axis is too high. Please return to zero after resetting	
0262	DSP alarm indicates that DSP has not been turned on. Please power on again	

Alarm number	Content	Remark
0263	DSP alarm indicates that DSP parameter is wrongly set	
0264	DSP alarm indicates that data sent is too large. Please power on again	
0265	DSP alarm indicates that bus cannot be connected or bus initialization fails	
0266	DSP interpolation axis speed exceeds 200M/MIN. Please return to zero after resetting	
0267	DSP initialization symbol (5555) is abnormal. Please return to zero after resetting	
0268	DSP unit recurrent pulse value is too large. Please return to zero after resetting	
0269	DSP internal alarm goes off. Please return to zero after resetting	
0270	DSP equal-dividing interpolation point is too short	
0271	DSP received interpolation data is too small. Please press E-stop and return to zero	
0272	DPS receives G codes that cannot be identified	
0273	DSP hardware data interaction is abnormal (instruction type)	
0274	DSP hardware data interaction is abnormal (data type)	
0275	In high-speed mode, interpolation speed multiple is 0	
0280	All axes should be returned to zero before using tool setting function	
0281	[Setup][tool setting & edge finding] interface must be switched to when using tool setting function	
0282	Please check whether tool setting device has been installed or set position parameter 1.6 as 1	
0283	Axis Z exceeds safe position. Please check tool setting device or tool length setting	
0286	Automatic tool length measurement has error. Please execute again	
0401	Drive unit alarm 01: servo motor speed exceeds set value	
0402	Drive unit alarm 02: main circuit power supply voltage is too high	
0403	Drive unit alarm 03: main circuit power supply voltage is too low	
0404	Drive unit alarm 04: positional deviation counter value exceeds set value	
0405	Drive unit alarm 05: motor temperature is too high	
0406	Drive unit alarm 06: speed regulator is saturated for a long time	
0407	Drive unit alarm 07: CCW and CW drive inhibiting input is off	
0408	Drive unit alarm 08: positional deviation counter value has absolute value over 230	
0409	Drive unit alarm 09: encoder signal error	
0410	Drive unit alarm 10: control power ±15V is low	
0411	Drive unit alarm 11: IPM intelligent module fault	
0412	Drive unit alarm 12: motor current is too large	
0413	Drive unit alarm 13: servo drive unit and motor overload (instantaneous overheating)	
0414	Drive unit alarm 14: brake circuit fault	
0415	Drive unit alarm 15: encoder counter exception	
0420	Drive unit alarm 20: EEPROM error	
0430	Drive unit alarm 30: encoder Z pulse error	
0431	Drive unit alarm 31: encoder UVW signal is wrong or mismatches with encoder	

Appendix

Alarm number	Content	Remark
0432	Drive unit alarm 32: UVW signal has full-high or full-low level	
0433	Drive unit alarm 33: communication outage	
0434	Drive unit alarm 34: encoder speed exception	
0435	Drive unit alarm 35: encoder status exception	
0436	Drive unit alarm 36: encoder counter exception	
0437	Drive unit alarm 37: encoder single-ring count overflow	
0438	Drive unit alarm 38: encoder multiple-ring count overflow	
0439	Drive unit alarm 39: encoder battery alarm	
0440	Drive unit alarm 40: encoder battery low	
0441	Drive unit alarm 41: motor model mismatch	
0442	Drive unit alarm 42: absolute position data exception alarm	
0443	Drive unit alarm 43: encoder EPPROM calibration alarm	
0449	Ethernet initialization fails Please check hardware	
0450	Drive unit disconnects. Please check whether hardware connection is correct	
0451	Axis 1 drive unit alarm	
0452	Axis 2 drive unit alarm	
0453	Axis 3 drive unit alarm	
0454	Axis 4 drive unit alarm	
0455	Axis 5 drive unit alarm	
0456	Main axis drive unit alarm	
0500	Axis 1 soft limit – direction overrun: (manually or manual pulse generator moves in + direction to lift)	
0501	Axis 1 soft limit – direction overrun: (manually or manual pulse generator moves in - direction to lift)	
0502	Axis 2 soft limit – direction overrun: (manually or manual pulse generator moves in + direction to lift)	
0503	Axis 2 soft limit – direction overrun: (manually or manual pulse generator moves in - direction to lift)	
0504	Axis 3 soft limit – direction overrun: (manually or manual pulse generator moves in + direction to lift)	
0505	Axis 3 soft limit – direction overrun: (manually or manual pulse generator moves in - direction to lift)	
0506	Axis 4 soft limit – direction overrun: (manually or manual pulse generator moves in + direction to lift)	
0507	Axis 4 soft limit – direction overrun: (manually or manual pulse generator moves in - direction to lift)	
0508	Axis 5 soft limit – direction overrun: (manually or manual pulse generator moves in + direction to lift)	
0509	Axis 5 soft limit – direction overrun: (manually or manual pulse generator moves in - direction to lift)	
0510	Axis 1 hard limit – direction overrun: (overrun lifting or manual pulse generator moves in + direction to lift)	
0511	Axis 1 hard limit – direction overrun: (overrun lifting or manual pulse generator moves in -	

Appendix

Alarm number	Content	Remark
	direction to lift)	
0512	Axis 2 hard limit – direction overrun: (overrun lifting or manual pulse generator moves in + direction to lift)	
0513	Axis 2 hard limit – direction overrun: (overrun lifting or manual pulse generator moves in - direction to lift)	
0514	Axis 3 hard limit – direction overrun: (overrun lifting or manual pulse generator moves in + direction to lift)	
0515	Axis 3 hard limit – direction overrun: (overrun lifting or manual pulse generator moves in - direction to lift)	
0516	Axis 4 hard limit – direction overrun: (overrun lifting or manual pulse generator moves in + direction to lift)	
0517	Axis 4 hard limit – direction overrun: (overrun lifting or manual pulse generator moves in - direction to lift)	
0518	Axis 5 hard limit – direction overrun: (overrun lifting or manual pulse generator moves in + direction to lift)	
0519	Axis 5 hard limit – direction overrun: (overrun lifting or manual pulse generator moves in - direction to lift)	
0600	Operation keyboard disconnects. Please check operation keyboard connecting cable	
1001	Relay or coil address is not set	
1002	Function code of the code entered does not exist	
1003	Function instruction COM is not used correctly. Relation between COM and COME is wrong or function instruction is used between COM and COME	
1004	User ladder diagram exceeds maximum allowed lines or steps. (Solution) Reduce edited NET number.	
1005	Function instructions END1 and END 2 do not exist; or END1 and END2 have error; or END1 and END2 have wrong order	
1006	There is illegal output in the Network. Please check output format	
1007	Hardware failure or system interruption results in inability of PLC to communicate. Please contact system equipment manufacturer	
1008	Function code is not correctly connected	
1009	Network horizon is not connected	
1010	Power fails when editing ladder diagram, resulting in loss of Network for editing	
1011	Address or data is not consistent with format of this function instruction. Input again	
1012	Address or data is not correctly entered. Input again	
1013	Illegal character is designated or data exceeds range	
1014	CTR address is repeated Reselect other CTR addresses that are not in use	
1015	Function instruction JMP is not correctly used and relation between JMP and LBL is wrong. Or JMP function instruction is used again between JMP and LBL	
1016	Network structure incomplete Change ladder diagram	
1017	Network structure currently not supported appears. Change ladder diagram	
1019	TMR address is repeated. Reselect other TMR addresses that are not in use	
1020	Parameter is absent from function instruction. Enter legal parameter	
1021	PLC execution timeout. The system automatically stops PLC. Please check ladder diagram logic and exclude endless loop or excessively repeated calling	

Alarm number	Content	Remark
1022	Function instruction name is absent. Please correctly enter function instruction name	
1023	Function instruction parameter address or constant exceeds range	
1024	Unnecessary relay or coil exists. Delete unnecessary connection	
1025	Function instruction is not correctly output	
1026	Network connection line number exceeds supported range. Change ladder diagram	
1027	Same output address is used elsewhere. Reselect output addresses that are not in use	
1028	Ladder diagram format error	
1029	Ladder diagram file in use is lost	
1030	There is incorrect vertical in the Network. Delete vertical line	
1031	User data area is full. Please reduce COD code data sheet capacity	
1032	First level of ladder diagram is too large and cannot be executed timely. Reduce first level of ladder diagram	
1033	Function instruction SFT exceeds maximum allowed number of use. Please reduce use	
1034	Function instruction DIFU/DIFD address is repeated. Please reselect address	
1039	Instruction or Network is not within executable range. Please clear	
1040	Function instruction CALL or SP is not correctly use. Relation between CALL and SP or between SP and SPE is wrong, or SP function instruction is used again between SP and SPE, or SP is set before use of END2	
1041	Horizontal breakover line is in serial connection with node Network	
1042	PLC system parameter file is not loaded	