This user manual describes all items concerning the operation of this CNC system in detail. However, it is impossible to give particular descriptions for all unnecessary or unallowable operations due to the length limitation and the product application conditions; Therefore, the items not especially described herein should be considered as "impractical" or "unallowable".

Preface

Your Excellency,

We are honored by your purchase of GSK 25i Milling CNC System made by GSK CNC Equipment Co., Ltd.

This book is GSK25i Machining Center CNC System User Manual Volume I: Programming and Operation”.

The incorrect operation may cause the accident, so only the professional can operate the system. Please read the manual carefully before operation!

Caution:

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

It can't be applied to other purposes, or else it may cause serious danger.

Warning and Precaution

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

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

Warning

The operator may be injured or the equipment damaged if the operation instructions and the procedures are not observed.

Caution

The equipment may be damaged if the operation instructions or the procedures are not observed.

Note

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

Announcement

- This manual describes all the various matters as much as possible. However, we can not describe all the matter which must not be done, or which can not be done, because there are so many possibilities. Therefore, matters which are not especially stated in this manual shall be considered as unallowable.

Notes

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

Precautions

■ Delivery and storage

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

■ Open-package inspection

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

■ Connection

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

■ Troubleshooting

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

Safety Responsibility

Manufacturer's Responsibility

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

User's Responsibility

— Be trained with the safety operation of CNC system and familiar with the safety operation procedures.
— Be responsible for the dangers caused by adding, changing or altering the original CNC systems and the accessories.
— Be responsible for the danger without observing the regulations in the manual for operation, adjustment, maintenance, installation and storage.

All specifications and designs are subject to change without notice.

This manual is kept by the end user.

Thank you for your cooperation during using GSK product!

CONTENT

GENERAL

About this manual

This manual consists of the following parts:

GENERAL

Describe the composition of chapters, related manuals, and notes for reading this manual.

Volume Ⅰ PROGRAMMING

Describe each function: Command format, characteristics and restrictions used for programming in the CNC language.

Volume Ⅱ OPERATION

Describe the manual, MPG and automatic operation of a machine, procedures for MDI mode and editing a program.

APPENDIX

It lists the alarm parameters of the CNC system.

1. General

Based on the market and the user requirement, GSK25i milling machine CNC system (herein abbreviated as the system) is the CNC control device of the new generation with the high precision, high performance, five-axes linkage, closed loop (semi-closed or closed loop) control, which is researched and developed by GSK CNC equipment Co., Ltd. It is the high-end and universal CNC system, which is widely applied in the machining center digit/numerical value control.

This manual describes GSK25i system programming, operation method and the parameter, input/output interface introduction in detail.

The manual also introduces the system selection function, while not all the selection functions are included in the actual assembly, so please refer to the manual from the machine tool builder.

2. Notes for Reading this Manual

The control function of CNC machine is not only set by CNC controller, but also the machine strong current return circuit, the servo device, the CNC controller and the machine operation control. About the combination, programming and operation of these control functions, the manual cannot describe all the matters. The user manual introduces the CNC system function; about the detailed description of the various machining machine control function, please refer to the user manual from the machine tool builder.

All the matters described in this manual are prior to those of the manual from the machine tool builder.

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

This manual makes explanations for some special items in notes.

Part Ⅰ Programming

PART Ⅰ PROGRAMMING

CHAPTER Ⅰ GENERAL

1.1 Definition

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

1.2 Program Configuration

A program consists of a group of blocks and a block is composed of several words. Each block is separated by end of block code ";"(ISO with LF code and EIA with CR code). In this manual, word "**;**" is the EOB code.

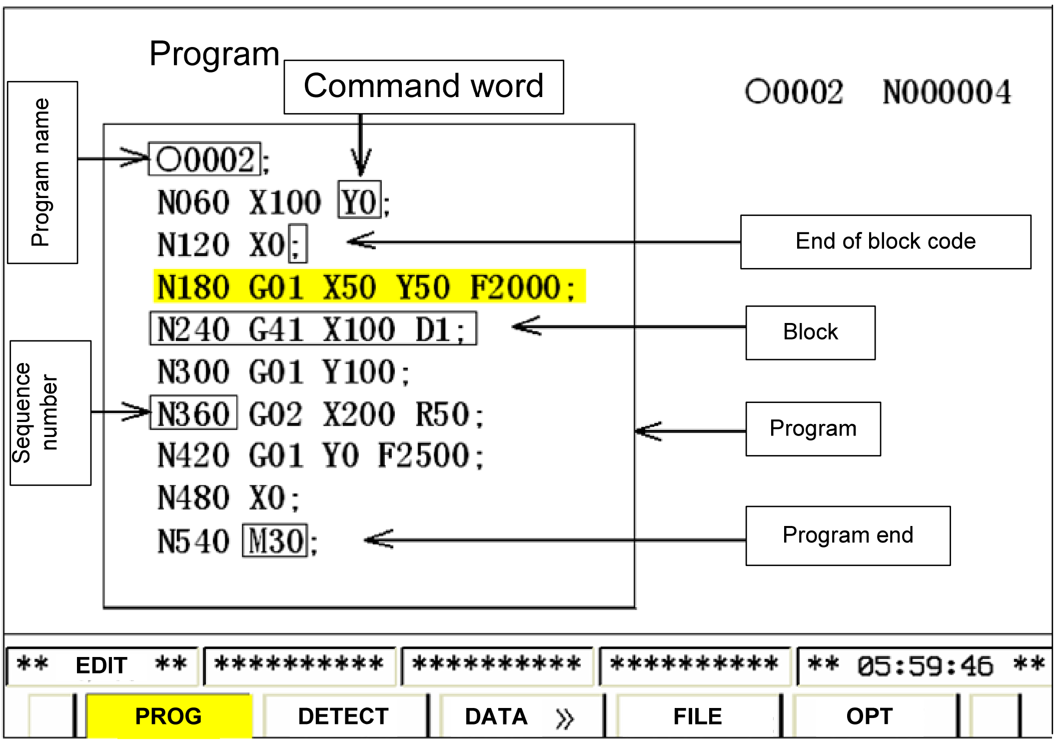


Fig. 1-2-1 Program configuration

A group of commands to control CNC machine tool for completing the part machining is called the program. After a program is input to CNC system, the system executes the operation such as the tool linear/circular movement, the spindle rotation/stop. The program should be edit in accordance with the actual move sequence of a machine tool. The program configuration is shown as Fig.1-2-1.

1.2.1 Program Name

This system is able to store several different programs. To identify them, a program name is composed of the letters and the ordered digits with the postfix .NC.

Example: YePian1.NC

Ball123.NC

If a program should be called (such as: M98), it should consist of the address O followed by four-digit number, which is shown as Fig. 1-2-2.

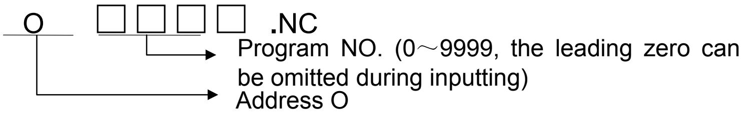


Fig. 1-2-2 Composition of the program name

1.2.2 Sequence Number and Block

A program consists of several commands. One command unit is called a block (refer to fig. 1-1). One block is separated from the other with the end of block code (refer to fig. 1-1) and in this manual, word ";" is taken as the EOB code.

At the head of a block, a sequence number is composed of address N followed by six-digit numbers (refer to fig. 1-1). The leading zero can be omitted. Sequence number can be specified in a random order, and its intervals can be unequal. Sequence number may be specified for all blocks or only for important blocks of a program. In general, based on the machining steps, it is convenient to assign sequence numbers in ascending order in the important part of a program. (For example, a tool is changed or machining proceeds to a new surface with table indexing.)

1.2.3 Word

Words are essential for a block (refer to Fig.1-2-3). A word consists of an address followed by some digits. (The plus sign (+) or minus sign (-) may be prefixed to a number.)

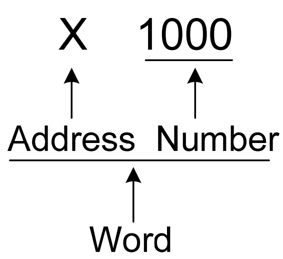


Fig. 1-2-3 Composition of a word

For an address, one of the letters (A to Z) is used. And the letter defines the meaning of digits those follow it. Table1-1 indicates the usable addresses, their meanings and its range.

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

List 1-1

Address	Range	Meaning
O	0～9999	Program name
N	0～999999	Sequence number
G	000～999	Preparatory function
X	-999999.9999～999999.9999（mm）	X-coordinate address
	0～9999.999（s）	Dwell time
Y	-999999.9999～999999.9999（mm）	Y-coordinate address

Address	Range	Meaning
Z	-999999.9999～999999.9999（mm）	Z-coordinate address
R	-999999.9999～999999.9999（deg）	Angle displacement
	-999999.9999～999999.9999（deg ）	Arc center/Plane R in the canned cycle
I	-999999.9999～999999.9999（mm）	X vector of arc center relative to the start position
J	-999999.9999～999999.9999（mm）	Y vector of arc center relative to the start position
K	-999999.9999～999999.9999（mm）	Z vector of arc center relative to the start position
	0～9999	Repeated times of canned cycle
F	0.01～200000（mm/min）	Feedrate per minute
	0.001～1000(mm/r)	Feedrate per revolution
S	0～50000（r/min）	Specifying spindle speed
	00～06	Multi-gear spindle output
T	0～999	Tool function
M	00～999	Miscellaneous function output, program executed flow, subprogram call
P	0～9999999（ms）	Dwell time
	0～9999	Call subprogram number
Q	-999999.9999～999999.9999（mm）	Cutting depth or offset amount of hole bottom in canned cycle
H	00～400	Length offset number
D	00～400	Radius offset number

Please note the List 1-1 shows the restrictions only for CNC device, the restrictions for machine tool are not included. Therefore, read this manual as well as the one provided by machine tool builder before programming.

1. 3 General Program Structure

A program contains the main program and subprogram. Usually, the CNC system performs according to the main program, unless there is a subprogram call in the main program, CNC operates based on the subprogram; while the command of returning to the main program occurs in the subprogram, CNC returns to the main program for executing. The sequence is shown in Fig. 1-3-1.

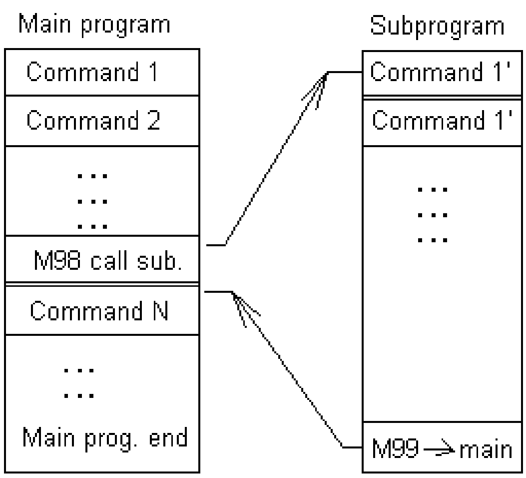


Fig. 1-3-1 Program running sequence

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

If a program contains a fixed sequence and frequently repeated pattern, the sequence or pattern can be stored as a subprogram in memory to simplify the program. A subprogram can be called in Auto mode by commanding M98 in main program. A called subprogram can also call the other subprogram. The subprogram called from the main program is named as the 1st nest one; subprogram call can be nested up to four levels (shown in Fig.1-3-2). The last block of the subprogram should be the return command M99. After executing M99 command, the program is returned to the main program to call the next program of the called one for executing. The program can be repeated when M99 is executed at the end of main program.

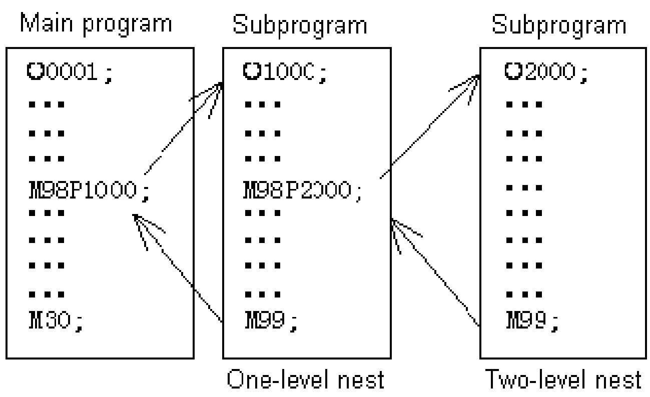


Fig. 1-3-2 Two-level nesting subprogram

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

1.3.1 Subprogram Writing and Call

1.3.1.1 Subprogram Writing

Write a subprogram as following format:

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

Fig. 1-3-3

At the head of a subprogram, the address O followed with the subprogram number is placed.

The end of the subprogram is command M99 (Editing format is shown as above).

The subprogram file name must be composed of O and the following digits, the digit range is 0001-9999.

(1) A subprogram is called by a calling command of the main program or that of the subprogram; its format is shown as follows:

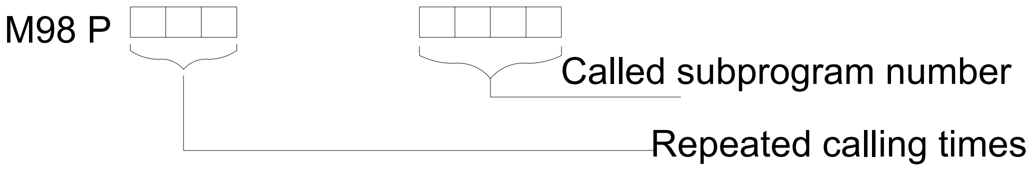


Fig. 1-3-4

● If the number of repetition times is omitted, it is assumed to be 1.

Example:

M98 P51002;（It indicates that subprogram number 1002 is called continually for 5 times）

The sequence of subprogram call in a subprogram is the same with that in the main program.

(2) The subprogram is called with the calling command of the main program or that of the subprogram; the command format of calling subprogram is shown as below:

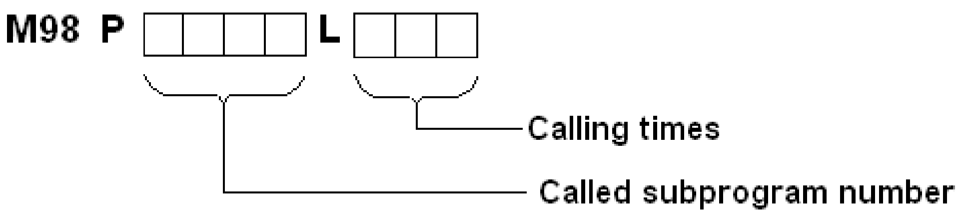


Fig. 1-3-5

● If the number of repetition times is omitted, it is assumed to be 1.

Example:

M98 P1002L5 ；（It indicates that subprogram number 1002 is called continually for 5 times.）

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

1.3.2 Program Inputting Format

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

Example:

With the following command, the tool can be positioned to 50.123mm along X axis:

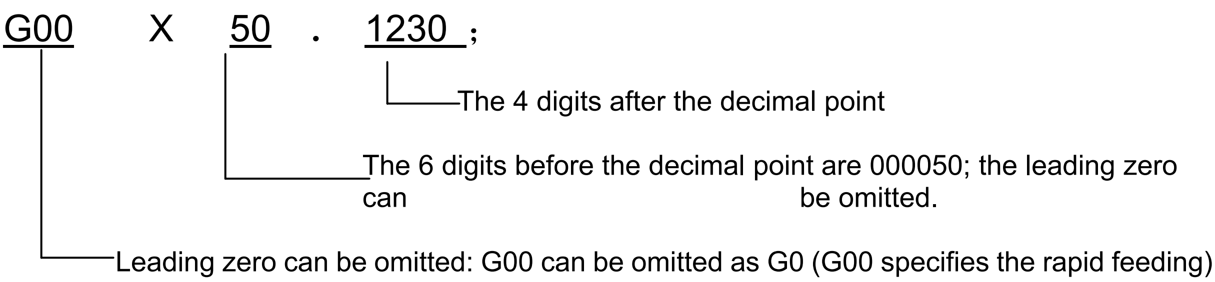


Fig. 1-3-6

Note: 1. If two commands are assigned by one address in the same block, the later command is valid in principle. No alarm will occur. Like the following example:

G00 G01 X100. Y200.;

G01 is valid, G00 is invalid.

1）The last specified G code of each group is valid.

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

2. In one block, if there only exists the semicolon, the system will automatically clear it after saving the program.

1.3.3 Program End

A Program starts from the program name and ends with command M02, M30 or M99.

1. M02 and M30 enable the system enter into resetting state at the end of a program; the difference between M02 and M30 is that the system returns to the beginning after executing M30; while it doesn't occur after M02.

2. If M99 is executed at the end of a program, the system returns to the program head and the program is executed in cycle; if M99 is executed at the end of the subprogram, the system returns to the program which has called the subprogram.

Warning!
1. If the optional block skip switch on the machine operation panel is ON, including the block with any optional skip block code, such as /M02, /M30 or /M99, they do not indicate the program end.
2. M02 is executed in MDI mode, the system returns to the program head.

1.3.4 Optional Block Skip

When “/” is at the head of a block, and optional block skip switch on the machine operator panel is set ON, the information contained in the block of the specified switch is invalid. When the optional block skip switch is set OFF, the information contained in the block specified by / is valid. This means the operator can decide whether skip blocks with / or not.

Example: (Not correct) (Correct)

// G00X10.0; /G00X10.0;

When the programs are registered in the memory, the blocks with / are also saved in it.

Notes:

1. The position of the slash

The slash (/) should be at the head of a block; otherwise, the information after the slash is ignored.

2. Disabling of optional block skip switch

When a block is read into buffer from memory, the optional block skip operation is ignored. Please turn on the switch before five blocks during the program running.

CHAPTER Ⅱ PROGRAMMING FUNDAMENTALS

2.1 Controlled Axes

List 2-1

ITEM	GSK25i
Number of basic controlled axes	5 axes（X,Y,Z,4th,5th）
Extended controlled axes（in total）	6 axes at most

2.2 Axis Name

The names of 5 basic axes are always X, Y, Z, 4th and 5th. The control is set by data parameter N0.0800 and the name of each axis is set by NO.1020.

2.3 Coordinate System

2.3.1 Machine Coordinate System

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

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

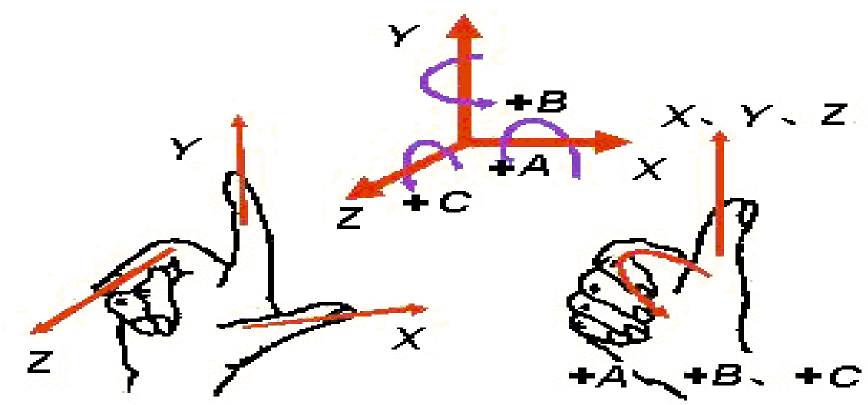


Fig. 2-3-1

2.3.2 Reference Position

A CNC machine tool has a special position, where, generally, the tool is exchanged or the coordinate system is set. This position is referred to as a reference position. It is a fixed position in machine coordinate system set by the machine builder. By the reference point return function, the tool can be easily moved to this position. Generally this position in CNC milling system coincides with the machine zero, while the reference position of machining center is usually the tool change point.

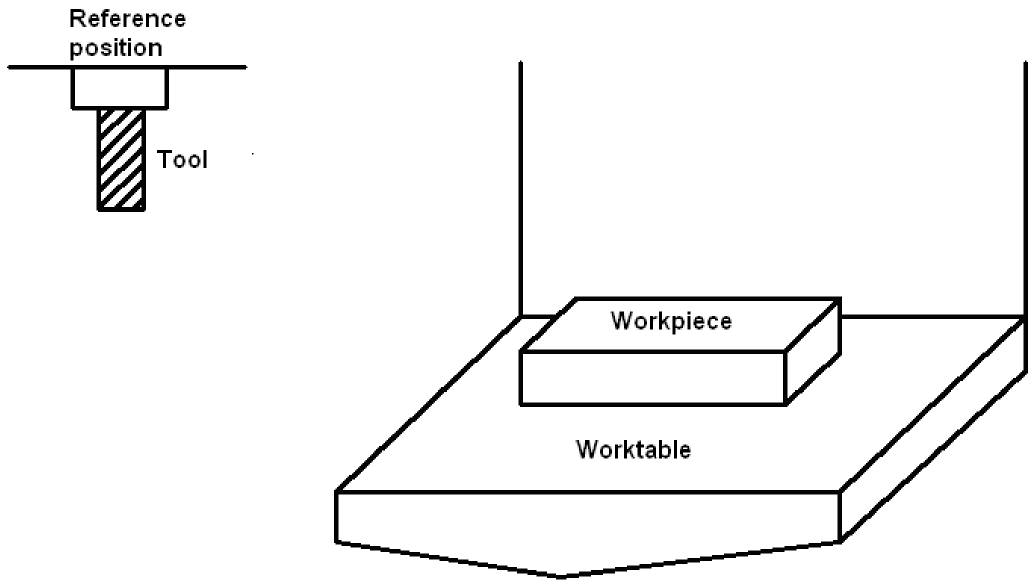


Fig. 2-3-2

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

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

2. Automatic reference point return

2.3.3 Workpiece Coordinate System

The coordinate system used for machining a workpiece is referred to as a workpiece coordinate system (also called as a part coordinate system), which is preset with CNC system（setting a workpiece coordinate system）.

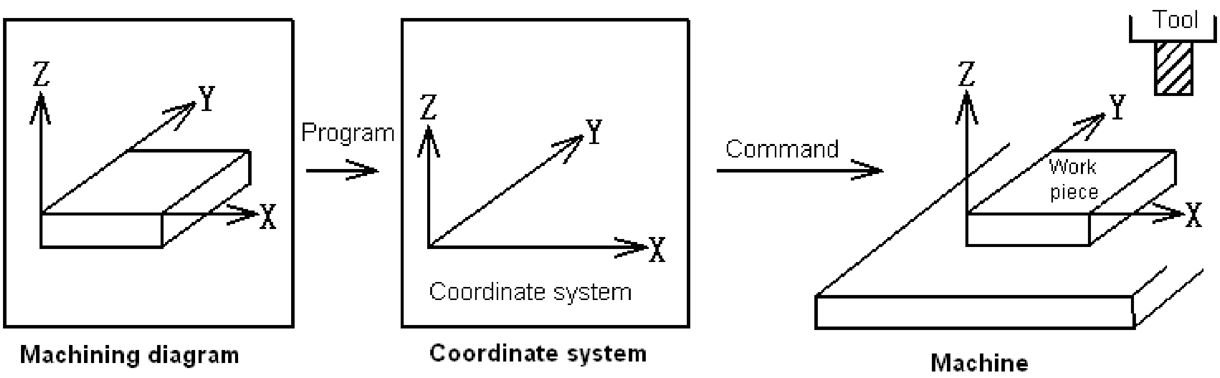


Fig. 2-3-3

The tool is on the workpiece coordinate system commanded by CNC and is machined the workpiece into the desired shape based on the drawing on which the programs are edit, so it is necessary to set the relative relationship between the machine coordinate system and the workpiece one.

The method of determining the relative relationship between these two coordinate systems is called the alignment. The different methods can be adopted according to the part shape or the workpiece quantity.

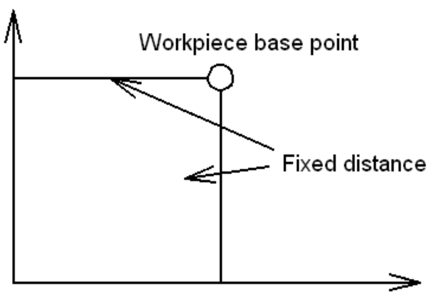
<table>
<tr><td>Ⅰ) With the workpiece base point</td><td>Ⅱ) The workpiece fixed on the jig</td></tr>
<tr><td>

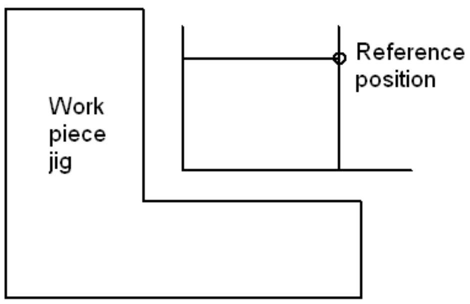
Fig. 2-3-4</td><td>

Fig. 2-3-5</td></tr>
<tr><td>To position the tool center into the workpiece base point, specify the workpiece coordinate system by CNC commands at this position, and then the workpiece coordinate system coincides with the programming one.</td><td>Because the tool center can't be directly positioned at the workpiece standard point, the tool should be positioned on which the certain known distance from the standard point or on the reference point, the workpiece coordinate system instructed by CNC commands is set by the known distance (e.g. G92).</td></tr>
</table>

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

A workpiece coordinate system can be set using one of two methods:

1. About method G92, see 3.2.13 for details.
2. About methods G54 to G59, see 3.2.10 for details.

2.3.4 Maximum Stroke

List 2-2 Maximum stroke

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

Note:

1. A command exceeding the maximum stroke cannot be specified.
2. The actual stroke depends on the machine tool.

2.3.5 Absolute and Incremental Programming

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

As the incremental command, the axis relative move amount is directly used for programming. Regardless of the coordinate, it just needs the move direction and distance of the end position relative to the start position.

G90 and G91 are used to instruct absolute and incremental commands, respectively.

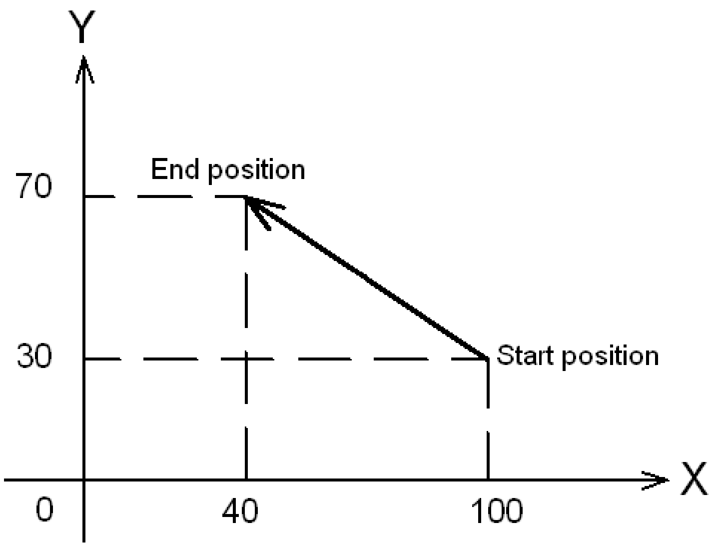


Fig. 2-3-6

In Fig. 2-3-6, moving from the start position to the end one involves the following situations using G90 programming command and G91 incremental command:

G90 G0 X40 Y70;

or G91 G0 X－60 Y40 ;

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

Explanation:

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

System parameters

At power-on, whether the defaulted position parameter is G90 (when parameter is 0) or G91 (when parameter is 1) is set by parameter N0:1801#3.

2.4 Modal and non-modal

Modal means that once the value of one address is set, it remains valid until it is reset.

Another meaning of modal is that after a function word is set, it is not necessary to re-input the word when the same function is used.

- **Example:**

G0 X100 Y100; （rapid positioning to X100 Y100）

X120 Y30; （rapid positioning to X120 Y30, G0 is modal and can be omitted.）

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

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

G0 X0 Y0; （rapid positioning to X0 Y0）

Initial mode is the default programming mode after power-on; please refer to table 3-1 for details.

Example:

- O0001
- X100 Y100; （rapid positioning to X100 Y100, G0 is the initial mode）
- G1 X0 Y0 F100;（linear interpolation to X0 Y0, at a feedrate of 100mm/min, G94 is the system initial mode）

Non-modal means that the values of corresponding address are valid only in the block which is specified, and they should be re-specified in the next block, like G commands of group 00 shown in List 3-1.

List 2-3 describes the modal and non-modal commands.

Table 2-3 Modal and non-modal commands

Modal	Modal G function	G commands can be cancelled by the others in the same group. Once the function is executed, it remains valid until it is cancelled by the other G commands.
	Modal M function	M commands can be cancelled by the others in the same group. Once the function is executed, it remains valid until it is cancelled by the other M commands.
Non-modal	Non-modal G function	Only valid in the specified block and cancelled at the block end
	Non-modal M function	Only valid in the current block

2.5 Decimal Point Programming

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

Explanation:

1. Whether use the decimal point for programming is set by parameter NO: 1800#0. When NO: 1800#0=1, the unit of programming value is mm, inch or degree; when NO: 1800#0=0, the unit is the least movement unit, which is set by parameter N0:1000#1.
2. Fractions less than the least input increment unit are truncated.

Example:

X9.87654; Truncated to X 9.877 when the least input increment is 0.001mm.
Processed as X 9.8765 when the least input increment is 0.0001mm

2.6 Basic Functions

2.6.1 Tool Movement along Workpiece Parts Figure—Interpolation

1） Tool movement along a straight line

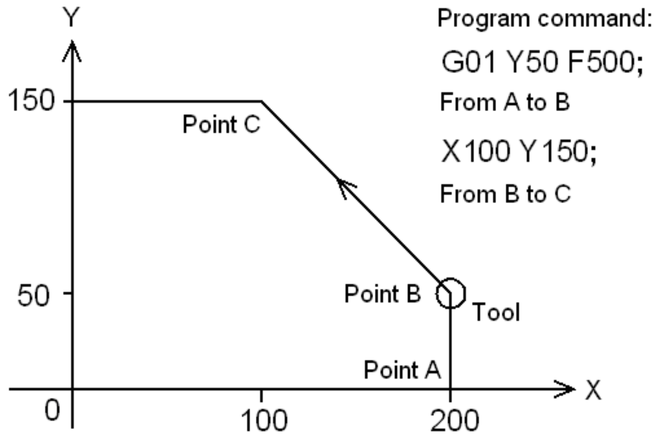


Fig. 2-6-1

2） Tool movement along an arc

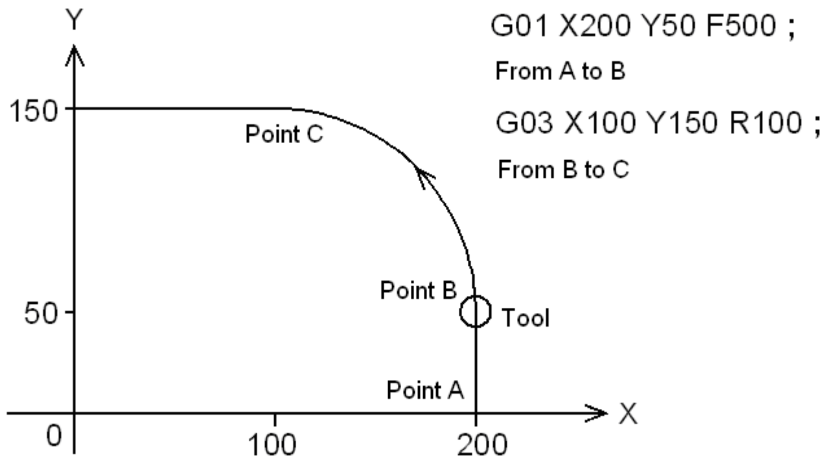


Fig. 2-6-2

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

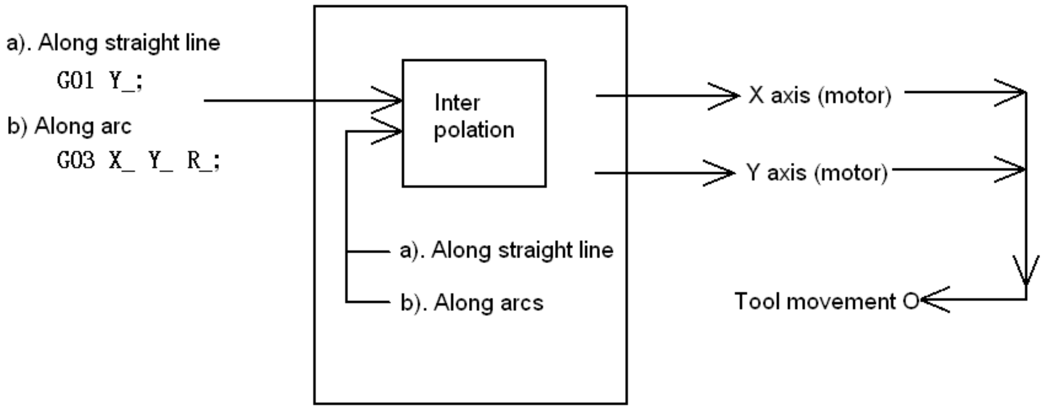


Fig. 2-6-3

Note: Some machines move tables instead of tools but this manual assumes that tools are moved against the workpiece. About the actual movement, please refer to the machine actual operation direction to avoid the personal injury and machine damage.

2.6.2 Feed—Feed Function

The function of deciding the feed rate is called the feed function.

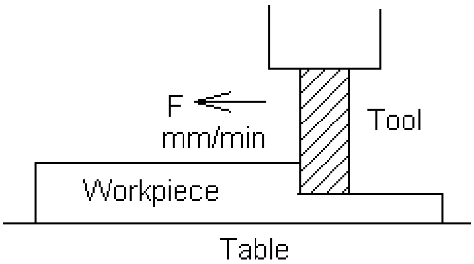


Fig. 2-6-4

Movement of the tool at a specified speed for cutting a workpiece is called the feed. Feedrate can be specified by using numeric. For example, to feed the tool at a rate of 200mm/min, specify the following in the program: F200.

2.6.3 Cutting Speed, Spindle Speed Function

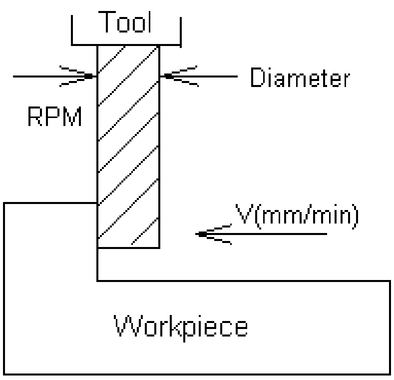


Fig. 2-6-5

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

For example, when a workpiece is machined with a tool 10mm in diameter at a cutting speed of 8m/min, the spindle speed is about 250r/min, which is obtained from N=1000V/πD. The command is S250.

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

2.6.4 Commands for Machine Operation—Miscellaneous Function

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

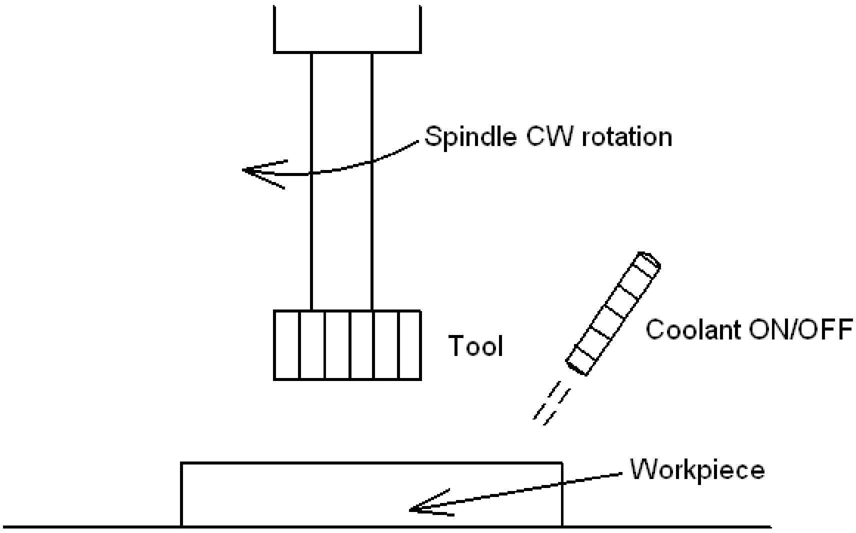


Fig. 2-6-6

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

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

2.6.5 Selection of Tool Used for Various Machining—Tool Function

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

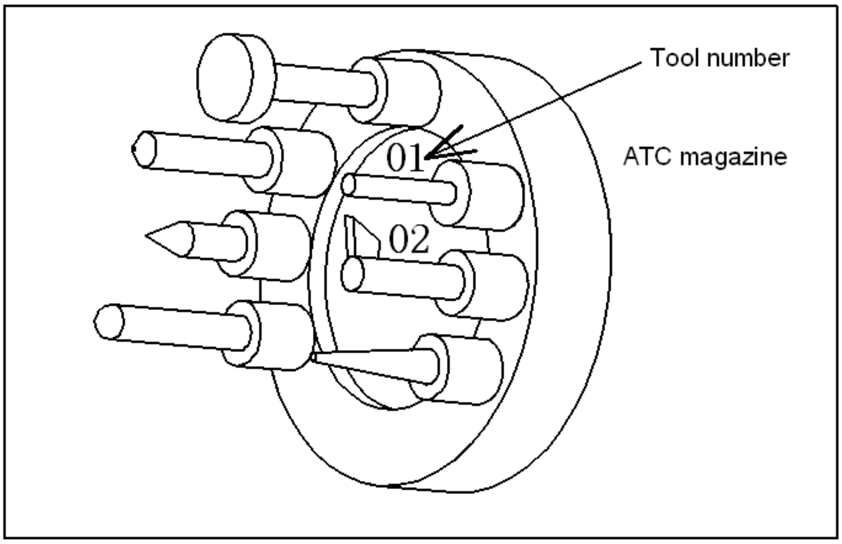


Fig. 2-6-7

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

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

2.6.6 Tool Figure and Tool Motion by Program

2.6.6.1 Tool Length Compensation

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

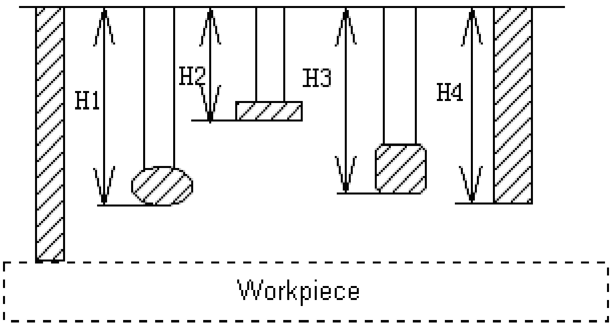


Fig. 2-6-8

Therefore, the length of each tool used should be measured in advance. By setting the difference between the length of the standard tool (normally the first tool) and the length of each tool in the CNC, machining can be performed without altering the program even when the tool is changed.

The distance from the tool end to the workpiece remains unchanged after Z axis positioning (such as G0Z0) is executed. This function is called tool length compensation.

2.6.6.2 Cutter Compensation Function

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

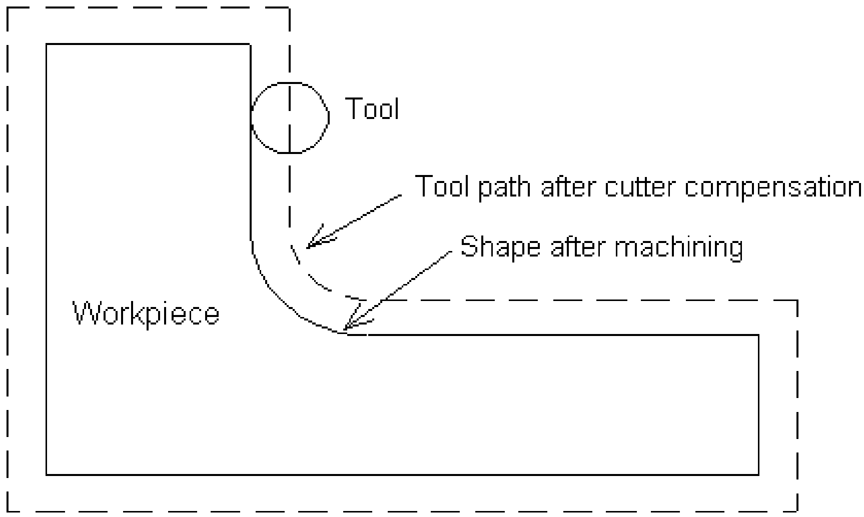


Fig. 2-6-9

If radius of cutters is stored in the CNC cutter compensation list in advance, the tool can be moved by cutter radius apart from the machining part figure. The function is called cutter compensation.

2.6.6.3 Tool movement range--stroke

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

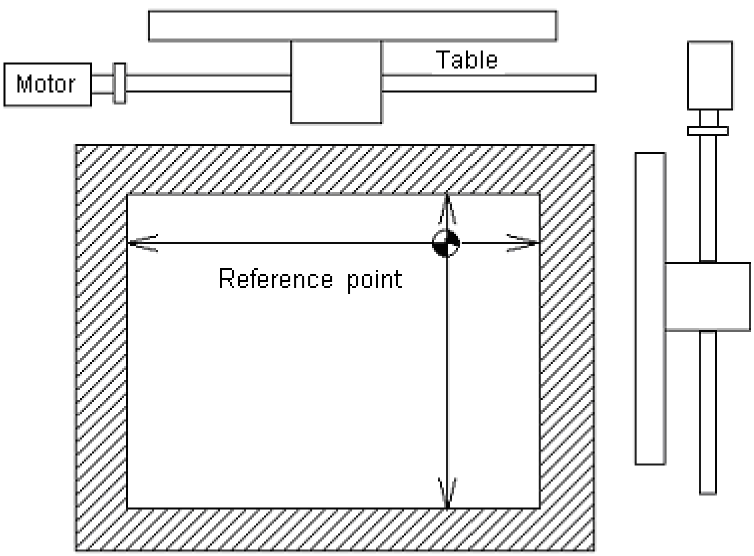


Fig. 2-6-10

CHAPTER III PREPARATORY FUNCTION G CODES

3.1 Types of G Codes

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

Table 3-1 Types of G codes

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

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

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

Table 3-2 List of G codes

G code	Group	Commands format	Function
*G00	01	G00 X_Y_Z_	Positioning (rapid traverse)
G01	01	G01 X_Y_Z_F_	Linear interpolation (cutting feed)
G02	01	G02 / G03 X_Y_ R_ / I_J_ F_;	Circular interpolation (CW)
G03	01		Circular interpolation (CCW)
G04	00	G04 P_ or G04 X_	Dwell, Exact stop
G05(G05.1)	00	G05 P_	High-speed contouring control in high precision
G06.2	01	G06.2 | P_| K_X_Y_Z_ |R_|	NURBS interpolation
G07.1 （G107）	00	G07.1 X_Y_Z_r	Column interpolation
G09	00		Exact stop
G10	00	G10L_；N_P_R_	Programmable data input
*G11	00	G11	Programmable data input mode cancel
*G15	14	G15	Polar coordinate command cancel
G16	14	G16	Polar coordinate command
*G17 G18 G19	02	It is written with other programs in the block and used in circular interpolation and cutter compensation	XY plane selection ZX plane selection YZ plane selection
G20	06		Input in inch
G21	06		Input in metric

G22	04	G22 X_Y_Z_I_J_K_			Stored travel detection function valid
G23		G23			Stored travel detection function invalid
G27	00	G27	X_Y_Z_		Reference point return check
G28		G28			Return to reference point
G29		G29			Return from the reference point
G30		G30Pn			2nd, 3rd,and 4th reference point return
G31		G31			Skip function
G37		G37			Automatic tool length measure
*G40	07	G17	G40 G41 G42	X_Y_	Cutter compensation cancel
G41		G18		X_Z_	Left cutter compensation
G42		G19		Y_Z_	Right cutter compensation
G43	08	G43		Z_	Tool length compensation + direction
G44		G44			Tool length compensation - direction
*G49		G49			Tool length compensation cancel
G43.4		G43.4 X_Y_Z_α_β_ H ；			Tool center point control
G45	00	G45		X_Y_Z__D_	Tool offset value increase
G46		G46			Tool offset value reduction
G47		G47			Two times of tool offset value
G48		G48			1/2 time of tool offset value
*G50	11	G50			Scaling cancel
G51		G51 X_ Y_ Z_ P_			Scaling
G50.1	21	G50.1 X_Y_Z_			Programmable mirror image cancel
G51.1		G51.1 X_Y_Z_			Programmable mirror image valid
G52	00	G52 X_Y_Z_			Local coordinate system set
G53		Written in the program			Machine coordinate system selection
G53.1					Tool axial direction control
*G54	17	It is written with other programs in the block and put at the beginning of the program.			Workpiece coordinate system 1
G55					Workpiece coordinate system 2
G56					Workpiece coordinate system 3
G57					Workpiece coordinate system 4
G58					Workpiece coordinate system 5
G59					Workpiece coordinate system 6
G54.1					Additional workpiece coordinate system
G60	00/01	G60 X_Y_Z_ F_			Single direction positioning
G61	15	G61			Exact stop mode
G62		G62			Automatic corner override
G63		G63			Tapping mode
*G64		G64			Cutting mode
G65	00	G65 P_L_＜Argument designation ＞			Macro program calling
G66	12	G66 P_L_＜Argument designation＞			Macro program mode calling
G67					Cancel macro program mode

			calling
G68	16	G68 X_ Y_ R_	Coordinate rotation
*G69		G69	Coordinate rotation cancel
G73	09	G73 X_Y_Z_R_Q_F_;	Peck drilling cycle
G74		G74 X_Y_Z_R_P_F_;	Left-hand tapping cycle
G76		G76 X_Y_Z_R_P_F_K_;	Fine boring cycle
*G80		It is written with other programs in the block.	Canned cycle cancel
G81		G81 X_Y_Z_R_F_;	Drilling cycle (spot drilling cycle)
G82		G82 X_Y_Z_R_P_F_;	Drilling cycle (stepped boring cycle)
G83		G83 X_Y_Z_R_Q_F;	Peck drilling cycle
G84		G84 X_Y_Z_R_P_F_;	Tapping cycle
G85		G85 X_Y_Z_R_F_;	Boring cycle
G86		G86 X_Y_Z_R_F_;	Boring cycle
G87		G87 X_Y_Z_R_Q_P_F_;	Back boring cycle
G88		G88 X_Y_Z_R_P_F_;	Boring cycle
G89		G89 X_Y_Z_R_P_F_;	Boring cycle
*G90	03	It is written with other programs in the block.	Absolute programming
G91			Incremental programming
G92	00	G92 X_Y_Z_	Coordinate system set
*G94	05	G94	Feed per minute
G95		G95	Feed per revolution
*G98	10	It is written with other programs in the block.	Return to the initial plane in canned cycle
G99			Return to point R (canned cycle)
G110	09	X_ Y_ R_ Z_ I_ L_ W_ Q_ V_ D_ F_ K_	Circular groove inner rough milling (CCW)
G111		X_ Y_ R_ Z_ I_ L_ W_ Q_ V_ D_ F_ K_	Circular groove inner rough milling (CW)
G112		X_Y_ R_ Z_ I_ L_ D_ F_ K_	Full circle finish milling cycle (CCW)
G113		X_Y_ R_ Z_ I_ L_ D_ F_ K_	Full circle finish milling cycle (CW)
G116		X_Y_ R_ Z_ I_ L_D_F_ K_	Circular outer finish milling cycle (CCW)
G117		X_Y_ R_ Z_ I_ L_D_F_ K_	Circular outer finish milling cycle (CW)
G130		X_Y_Z_R_I_J_L_W_Q_V_U_D_F_K_	Rectangular groove rough milling (CCW)
G131		X_Y_Z_R_I_J_L_W_Q_V_U_D_F_K_	Rectangular groove rough milling (CW)
G132		X_ Y_ R_ Z_ I_ J_ D_ L_ U_ F_ K_	Rectangular groove inner finish milling cycle (CCW)
G133		X_ Y_ R_ Z_ I_ J_ D_ L_ U_ F_ K_	Rectangular groove inner finish milling cycle (CW)
G136		X_Y_R_Z_I_J_D_L_U_F_K_	Rectangular outer finish milling cycle (CCW)
G137		X_Y_R_Z_I_J_D_L_U_F_K_	Rectangular outer finish milling cycle (CW)

G120	00	G120 G73-G89 X_Y_**R_Z_(Q_P_)** I_ J_ K_**F_**; or G73-G89 X_ Y_**R_Z_(Q_P_)F_;** G120 X_ Y_ I_ J_ K_ _	Circle hole cycle
G121		G121 G73-G89 X_Y **R_Z_(Q_P_)** I_ J_ K_**F_**; or G73-G89 X_ Y_**R_Z_(Q_P_)F_** G121 X_ Y_ I_ J_ K_ _	Angle linear hole cycle
G122		G122 G73-G89 X_ Y_**R_Z_(Q_P_) I-** _ J_ **E**_K_**F_**; or G73-G89 X_ Y_**R_Z_(Q_P_)F_;** G122 X_ Y_ I_ J_**E**_ K_ ;	Arc hole cycle
G123		G123 G73-G89 X_ Y_**R_Z_(Q_P_) I-** _ J_ **E**_K_**F_**; or G73-G89 X_ Y_**R_Z_(Q_P_)F_;** G123 X_ Y_ I_ J_**E**_ K_ ;	Grid hole cycle
G124		**G124 G73-G89 X_Y_R_Z_(Q_P) J_ E_ K_ F_** ; or G73-G89 X_ Y_**R_Z_(Q_ P_)F_** G124 X_ Y_ J_**E**_ K_ _	Rectangular drilling (CW)
G125		**G125 G73-G89 X_Y_R_Z_(Q_P) J_ E_ K_ F_** ; or G73-G89 X_ Y_**R_Z_(Q_ P_)F_** G125 X_ Y_ J_**E**_ K_ __	Rectangular drilling (CCW)
G126		X_Y_Z_I_J_L_F_	Round trip milling
G127		X_Y_Z_I_J_L_F_	Single trip milling

Note:

1. The G codes with mark * are the default ones at power-on state.
2. The system alarms when G codes which doesn't exist are commanded or can't be selected.
3. Multiple G codes of different groups can be specified in the same block. If multiple G codes that belong to one group are specified in the same block, only the last specified one is valid.
4. The canned cycle G codes and G codes of group 01 are specified in the same block, those of 01 group are executed; in canned cycle (not in the same block), if G codes of 01 group are specified, the canned cycle is cancelled and G80 is set.
5. G codes are indicated by group numbers according to their types.
6. About the programming details, refer to G code programming rules in Appendix list 1.

3.2 Simple G Codes

3.2.1 Positioning (G00)

Format: G00 IP_
Function:

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

Explanation: IP_:

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

Path:

The tool path of G00 is shown as fig. 3-1. The tool path and the linear interpolation (G01) are same; the tool is positioned in the shortest time at the rapid traverse rate.

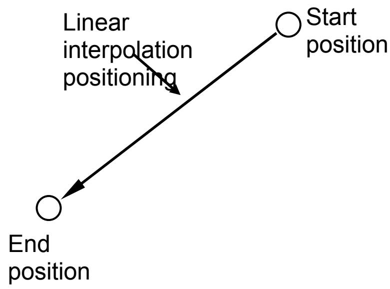


Fig. 3-2-1

Explanations:

1. G00 rapid traverse rate is set by parameter NO: 1126, and the current tool move mode is changed into G00 mode. By changing the value of parameter NO: 1801#0, the default mode after power-on can be set as G00 (parameter value is 0) or G01 (parameter value is 1)

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

3. G00 is identical with G0.

Restrictions:

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

G0 X0 Y10 F800;   Feeding at a rate set by system parameters
G1 X20 Y50;   at the rate set by F800

2. The following keys on the operation panel are used to adjust rapid feedrate, see fig. 3-2, involving such overrides as F0, 25%, 50% and100%; The feedrate corresponding to F0 is set by parameter NO: 1231, and it applies to all axes.

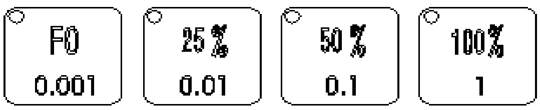


Fig.3-2-2 Rapid feedrate override keys

3.2.2 Linear Interpolation G01

Format: G01 IP_ F_

Function: A tool moves along a line to a specified position at the feedrate set by F.

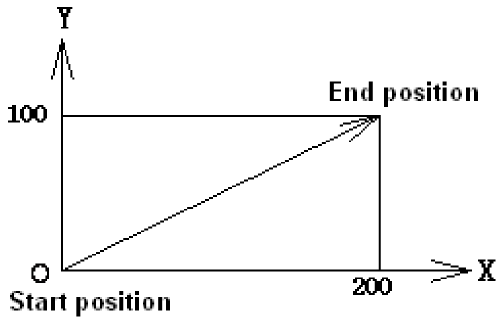
IP_: For absolute command, the coordinates of an end point, and for an incremental command, the distance the current tool moves.

F_: Speed of tool feed (feedrate)

Explanation:

The feedrate should be specified in F and it is effective until a new value is specified. The feedrate commanded by the F code is measured along the linear interpolation path. If the F code is not commanded, the feedrate is regarded as the speed set by parameter NO: 1211.

Example (Fig.3-2-3)



G01 X200 Y100 F200

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

G01 Xα Yβ ZγFf ；

In this block:

Feedrate of X axis direction:

$$F_X = \frac{\alpha}{L} \times f$$

Feedrate of Y axis direction:

$$F_Y = \frac{\beta}{L} \times f$$

Feedrate of Z axis direction:

$$F_Z = \frac{\gamma}{L} \times f$$

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

Fig. 3-2-3

Explanations:

1. The ceiling limits of cutting feedrate F for each axis can be set by parameter NO:1125. If the actual cutting feedrate (feedrate after override is used) exceeds the ceiling limit, the feedrate is limited in the ceiling one (Unit mm/min). The ceiling limit of multi-axes resultant cutting feedrate can be set by parameter No.: 1124. If the actual cutting feedrate (feedrate after override is used) exceeds the ceiling limit, the feedrate is limited in the ceiling one (Unit: mm/min).

2. The tool does not move when a position parameter followed by G01 is not specified, and the current tool move mode is changed into G01 mode. By changing the bit parameter No: 1801#0, the default mode after power-on can be set as G00 (parameter value is 0) or G01 (parameter value is 1).

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

Example: G91 G01 B120 F500;

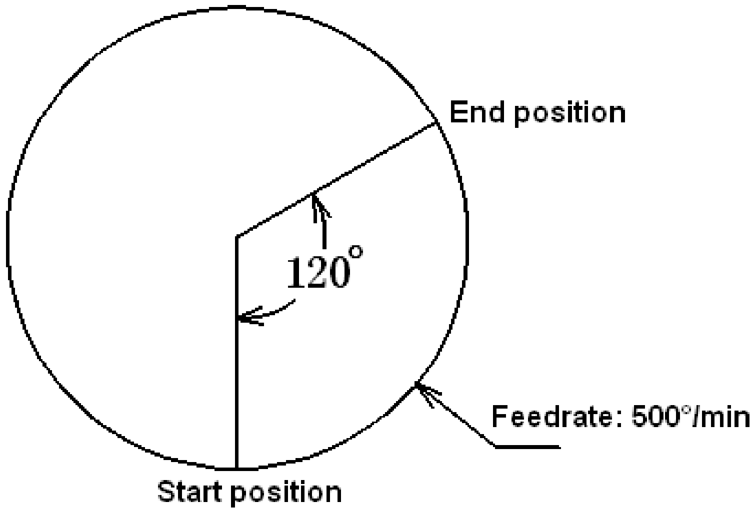


Fig. 3-2-4

Example: G91 G01 X20 B40 F300;

When the unit (degree) of B axis movement command is changed from degree to mm or inch, the calculation formula of machining time is as follows:

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

The feedrate of B axis is:

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

3.2.3 Circular Interpolation (Helical Interpolation) G02/G03

3.2.3.1 Circular Interpolation G02/G03

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

Arc in the X—Y plane

$$G17\begin{Bmatrix}G02\\ G03\end{Bmatrix} X\text{——}Y\text{——}\begin{Bmatrix}R\text{——}\\ I\text{——}J\text{——}\end{Bmatrix} F\text{——};$$

Arc in the Z——X plane

$$G18\begin{Bmatrix}G02\\ G03\end{Bmatrix} X\text{——}Z\text{——}\begin{Bmatrix}R\text{——}\\ I\text{——}K\text{——}\end{Bmatrix} F\text{——};$$

Arc in the Z——X plane

$$G19\begin{Bmatrix}G02\\ G03\end{Bmatrix} Y\text{——}Z\text{——}\begin{Bmatrix}R\text{——}\\ J\text{——}K\text{——}\end{Bmatrix} F\text{——};$$

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

After power-on, G17 is effective as an initial code for selecting the plane.

Explanations:

The command is used in G51 mode, if the scaling is set by I, J and K parameters, the radius scaling is set by the current plane. For example, on G17 plane, the radius scaling is the maximum value of I and J absolute values.

"Clockwise"（G02） and "counterclockwise"（G03）on the XeYe plane（ZeYe plane or YeZe plane） are defined when the XeYe plane is viewed in the positive-to-negative direction of Ze axis（Ye axis or Xe axis respectively) in the Cartesian coordinate system. See the figure below.

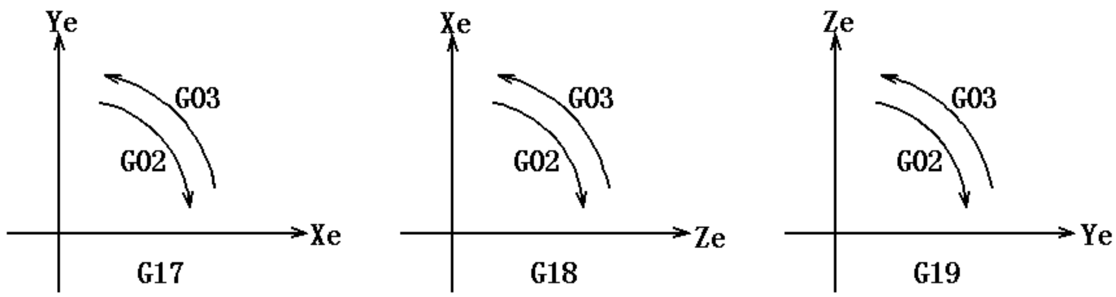


Fig. 3-2-5

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

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

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

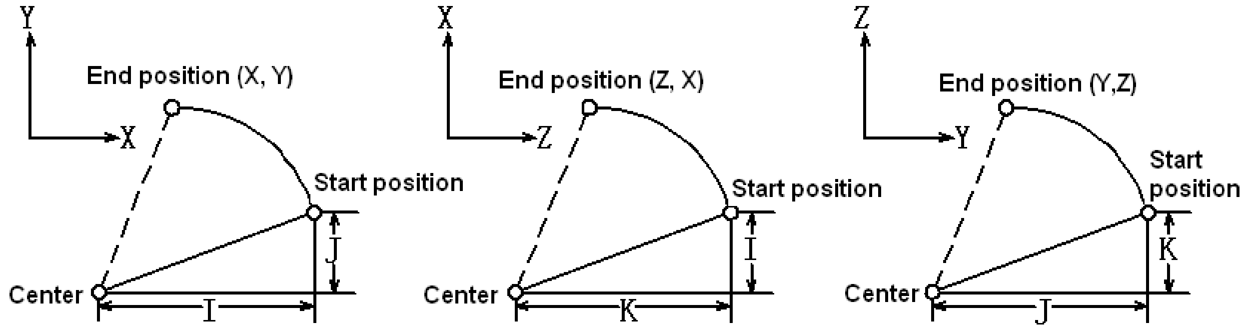


Fig. 3-2-6

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

G02 I_; command a full circle

If the difference between the radius at the start point and that at the end point exceeds the permitted value in a parameter (No: 1810), an alarm occurs.

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

G02 R; （the cutter does not move）

Example 1:

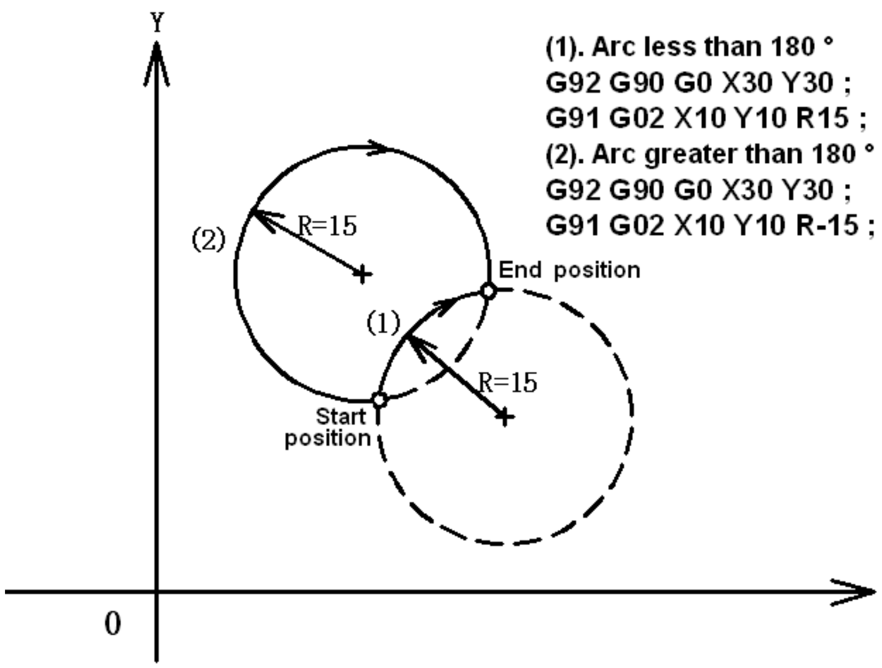


Fig. 3-2-7

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

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

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

Example 2:

a）Absolute programming

(1) Using I and J for programming:
G92　G90 X180 Y20 ；
G90　G03 X100 Y100 I-80 ；
G02　X35 Y35 I-65 ；
(2) Using R for programming:
G92　G90 X180 Y20 ；
G90　G03 X100 Y100 R80 ；
G02　X35 Y35 R65 ；

b）Incremental programming

(1) Using I and J for programming:
G91　G03 X-80 Y80 I-80 ；
G02　X-65 Y-65 I-65 ；
(2) Using R for programming:
G91　G03 X-80 Y80 R80 ；
G02　X-65 Y-65 R65 ；

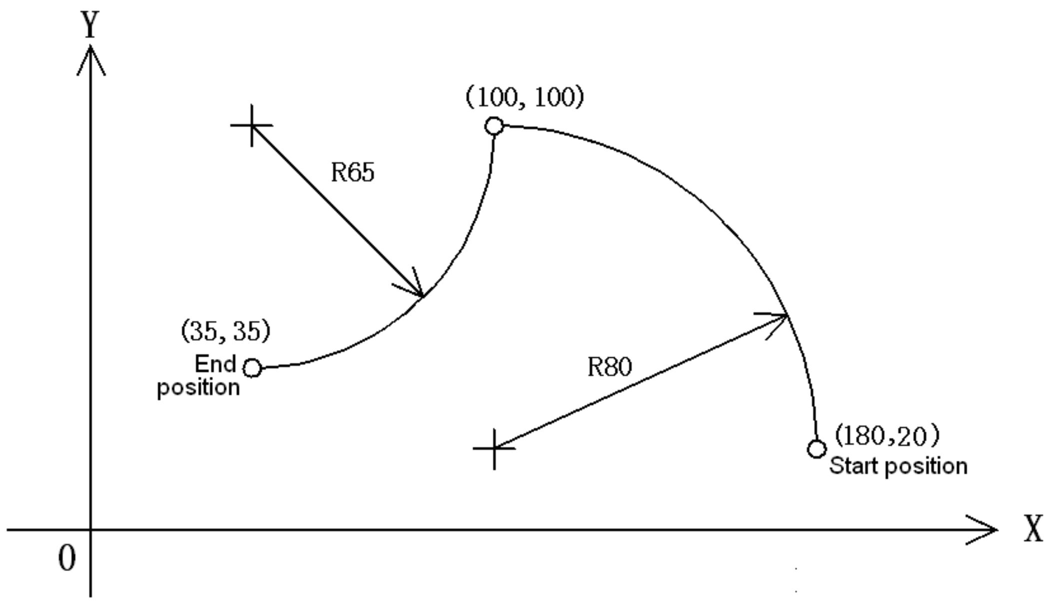


Fig. 3-2-8

3.2.3.2 Helical Interpolation G02/G03

Format:G02/G03

Arc of XY plane

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

Arc of ZX plane

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

Arc of YZ plane

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

Fig. 3-2-9

Function:

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

Explanations:

When the command is used in G51 mode, if the scaling is set by I, J and K parameters, the radius scaling is determined by the current plane. For example, on G17 plane, the radius scaling is I or J maximum absolute value.

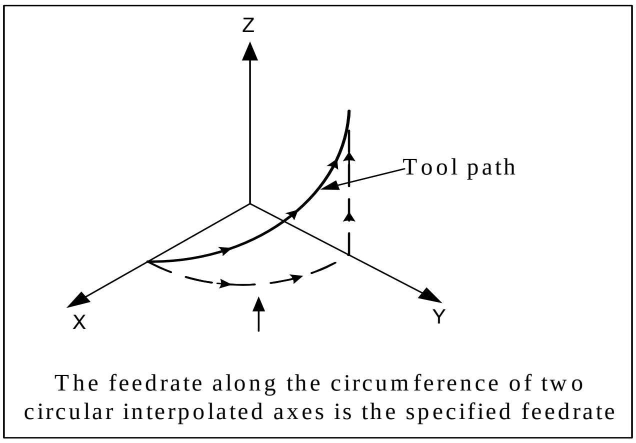


Fig. 3-2-10

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

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

Restrictions:

1. Cutter compensation is applied only for a circular arc

2. F is just the arc linear speed, and the synchronous move axis speed is the follow-up speed. The system only limits the arc linear speed set by No.1224 and the synchronous move axis speed set by No.1225, but the final combined speed may be greater than the value set by No.1224.

3.2.4 Cylindrical Interpolation (G07.1)

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

Format:

```
G07.1 IP r: The cylindrical interpolation mode is started
          :
          :
          :
G07.1 IP 0: The cylindrical interpolation mode is cancelled.
        IP: An address for the rotation axis
        r: The radius of the cylinder
     Specify G07.1 IP r and G07.1 IP 0 in different blocks,
     G107 can be used instead of G07.1.
```


Explanations:

(1) Plane selection (G17, G18, G19)

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

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

(2) Feedrate

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

(3) Circular interpolation (G02, G03)

In the cylindrical interpolation mode, circular interpolation can be performed between the rotary axis and the other linear axis. Radius R is used in commands in the same way as circular interpolation.

The unit for a radius is not degrees but mm (for metric input) or inch (for inch input)

Example:

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

G18 Z_C_;
G02(G03) Z_C_R_;

It doesn't matter to set the parameter (No.1024) of C axis as 7 (axis parallel to Y axis).

The command is:

G19 C_Z_;
G02(G03) Z_C_R_;

(4) Cutter compensation

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

5) Cylindrical interpolation accuracy

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

$$\text{Actual distance}=\left[\frac{\text{Distance/REV}}{2\times 2\pi R}\times\left[\mathit{Specified\ value}\times\frac{2\times 2\pi R}{\text{Distance/REV}}\right]\right]$$

REV distance: The distance per revolution (normally 360°) can be set by the parameter (No.1068) in the system.
R: Workpiece radius.
[ ] : The minimum setting unit can be rounded off.

Restrictions:

1) Format of cylindrical interpolation mode

The cylindrical interpolation must be set and cancelled based on the command format; otherwise, the system alarms.

2) Specify arc radius in cylindrical interpolation mode

In cylindrical interpolation mode, the radius of cylindrical interpolation should be greater than 0.5 mm or 0.5 inch for the interpolation precision; otherwise, the system processes as the illegal G107 command. Without setting the cylindrical interpolation, the cylindrical interpolation also causes the mistake.

3) Circular interpolation and cutter compensation

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

4) Positioning

In cylindrical interpolation mode, positioning cannot be specified (including cycles that generate rapid traverse, such as G28, G53, G73, G74, G76, G80~G89). Cylindrical interpolation mode should be cancelled before positioning.

5) Coordinate system setting

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

6) Cylindrical interpolation and resetting

In the cylindrical interpolation mode, the cylindrical interpolation mode cannot be reset. The cylindrical interpolation mode should be cancelled through pressing the emergency stop button after feed hold or directly pressing it.

7) Index table indexing function

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

(8) Cutting codes

In the cylindrical interpolation, the cutting codes only support arcs G0, G1, G2 and G3, if the cutting codes of the other types (such as the helical interpolation or the thread cutting, etc), the system processes as the incorrect G codes.

The cutting codes can only occur in the cylindrical rotation axis; otherwise, the cylindrical interpolation can not be performed correctly.

(9) The display of the revolving axis and the movement principles

Because the function of the rotary axis in the cylindrical interpolation is different with that in the common mode, the rotary axis coordinate type only adopts linear axis one in the cylindrical interpolation mode, which is not affected by the parameter #1023.2, but its display is still affected the parameter #1023.2; therefore, it's suggested to use the command value display mode.

Since the movement principle is changed, when the cylindrical interpolation is entered at the first time, the absolute coordinate of the cylindrical interpolation rotary axis is switched into the equal phase, then, (0,360) is obtained, and the numerical value is taken as basic one of the cylindrical interpolation rotary axis to calculate the rotary amount of the cylindrical interpolation. Therefore, the difference value is 360*N degree, which is between the programming command value of the cylindrical interpolation rotary axis and the display command value, and N is the integer, the big or small is determined by the equal phase switching process. After cancelling the cylindrical interpolation, the rotary amount caused by the rotary axis is kept in the cylindrical interpolation, it is suggested that the user respectively execute the rotary axis coordinate positioning each time before and after the cylindrical interpolation.


```
N61 G91 G28 Z0;
N62 G17 G90 A0 Z150;
N63 G1 Z57.299 X110 F300;
N64 G07.1 A57.299;
N65 G41 D1;
N66 A30;
N67 G02 X80 A60 R30;
N68 G01 X60;
N69 G03 X50 A70 R10;
N70 G01 A150;
N71 G03 X60 A190 R75;
N72 G01 X100 A230;
N73 G02 X120 A270 R75;
N74 G1 A360;
N75 G0 Z150;
N76 G40 A365;
N77 A375;
N78 G07.1 A0;
N79 M30;
```

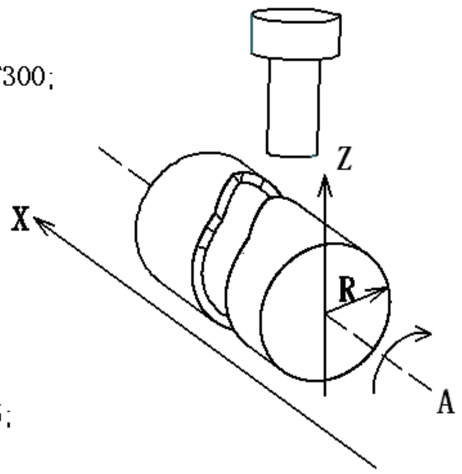


Fig. 3-2-11

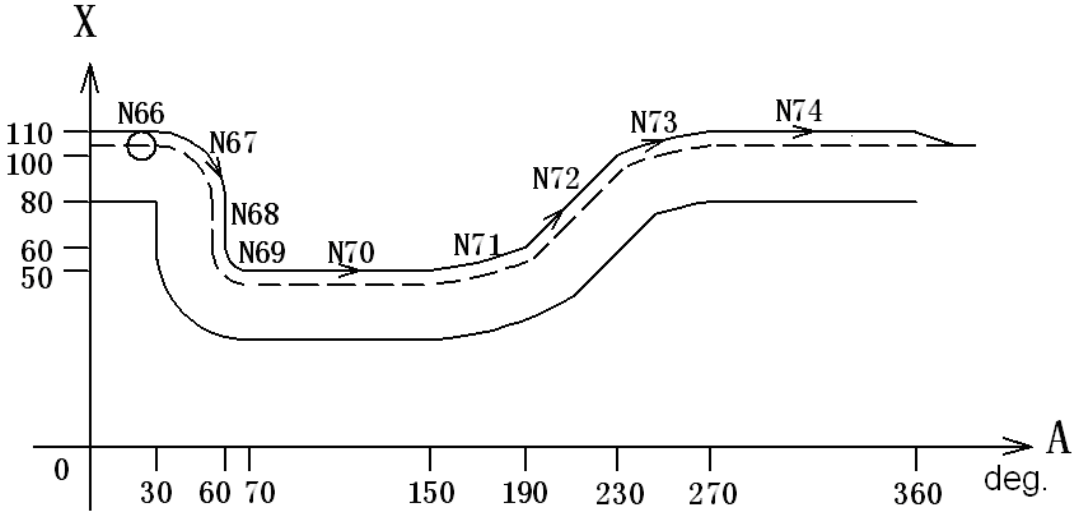


Fig. 3-2-12

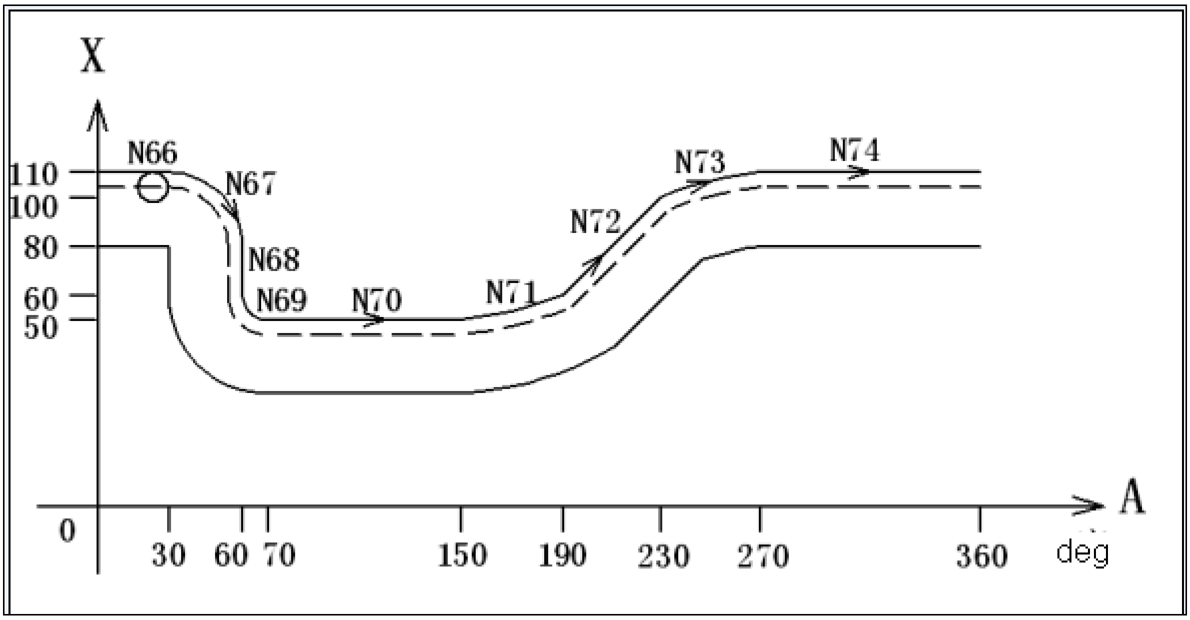


Fig. 3-2-13

3.2.5 NURBS interpolation

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

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

For the NURBS expression form output by CAD, the NURBS is created with CAM based on the length of the tool post and the cutter compensation, etc. The NURBS curve is programmed by using 3 parameters of NC sentence format: control point, weight and knot.

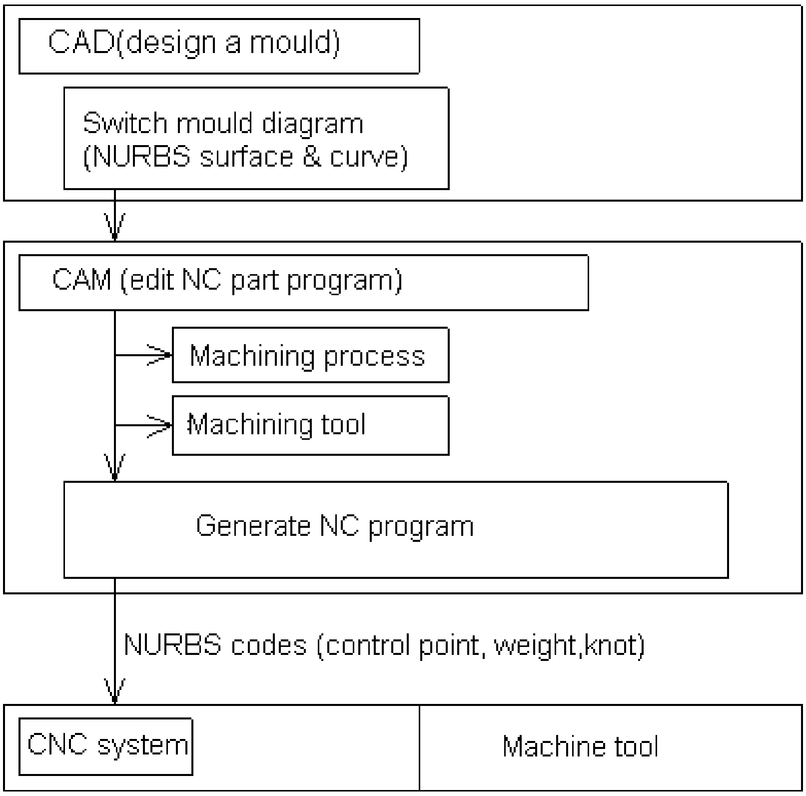


Fig. 3-2-14 NC part program for machining mould based on NURBS curve commands

CNC executes the NURBS interpolation while the acceleration/deceleration of all axes is controlled within the permitted range to prevent collision. The command format is as below:

```
G06.2 ［P_］K_X_Y_Z_［R］［F］;
      K_X_Y_Z_［R］;
      K_X_Y_Z_［R］;
      K_X_Y_Z_［R］;
      ...
      K_X_Y_Z_［R］;
      K_;
      ...
      K_;
G01...
...
```

G06.2: NURBS interpolation mode ON
P_: The rank of NURBS curve
X_Y_Z_: Control point
R_: Weight
K_: Knot
F_: Feedrate

Notes:

1. NURBS interpolation mode

NURBS interpolation mode is selected when G06.2 is modal G code of 01 group. Therefore, specifying the G codes in 01 group other than G06.2 (such as G00, G01, G02 and G03, etc) can end the NURBS interpolation.

2. NURBS rank

A rank of NURBS can be specified by address P. the rank setting, if any, must be specified in the first block. If the rank setting is omitted, rank 4 (degree 3) is assumed for NURBS. The valid data range for P is 2-4. The P values have the following meanings:

P2: NURBS in rank 2 (degree 1)
P3: NURBS in rank 3 (degree 2)
P4: NURBS in rank 4 (degree 3) (default value)

The rank referred here is the “k” in the definition expression of NURBS curve described latter. For example, degree 3 is for NURBS curve when the rank is 4.

3. Weight

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

4. Knot

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

5. NUBRS curve

Describe every variable in following formats:

k: rank
Pi: control point
Wi: weight
Xi: knot ($X_i \le X_{i+1}$)
Knot vector [X0, X1,..., Xm] (m = n+ k)
t: spline parameter

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

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

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

Then, NURBS curve P(t) for interpolation is shown as below.

$$P(t)=\frac{\sum_{i=0}^{n} N_{i,k}(t)w_iP_i}{\sum_{i=0}^{n} N_{i,k}(t)w_i} \qquad (x_0 \leq t \leq x_m)$$

Fig. 3-2-15

6. Reset

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

Restrictions:

1. Controlled axes

Up to 3 axes can perform NURBS interpolation. All the axes that perform NURBS interpolation should be specified in the first block (G06.2 block) for dedicating the control points. When there is no axis commanded in the first block, the axes are specified in the second block or the following one, the program error occurs and an alarm is issued.

2. Commands in NURBS interpolation mode

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

3. Manual intervention

When manual intervention is performed when the manual absolute mode is ON, P/S alarm is generated.

4. Cutter compensation

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

5. Control point

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

Example

< Program example of NURBS interpolation>

G54G40G17G49G90G21;

G91G28Z0.0;

```
G0G90X0.0Y0.0;
X-33.22Y-15.695S0M03;
G43Z10.H00;
Z-21.F5000;
G0X54.493Y52.139Z0.000;
G6.2P4K0.000000X54.493Y52.139Z0.000R1.000;
K0.000000X55.507Y52.139Z0.000R1.000;
K0.000000X56.082Y49.615Z0.000R1.000;
K0.000000X56.780Y44.971Z0.000R1.200;
K0.008286X69.575Y51.358Z0.000R1.000;
K0.014978X77.786Y58.573Z0.000R1.000;
K0.036118X90.526Y67.081Z0.000R1.000;
K0.085467X105.973Y63.801Z0.000R1.000;
K0.129349X100.400Y47.326Z0.000R1.000;
K0.150871X94.567Y39.913Z0.000R1.000;
K0.193075X92.369Y30.485Z0.000R1.000;
K0.227259X83.440Y33.757Z0.000R2.000;
K0.243467X91.892Y28.509Z0.000R1.000;
K0.256080X89.444Y20.393Z0.000R1.000;
K0.269242X83.218Y15.446Z0.000R5.000;
K0.288858X87.621Y4.830Z0.000R3.000;
K0.316987X80.945Y9.267Z0.000R1.000;
K0.331643X79.834Y14.535Z0.000R1.100;
K0.348163X76.074Y8.522Z0.000R1.000;
K0.355261X70.183Y12.550Z0.000R1.000;
K0.364853X64.171Y16.865Z0.000R1.000;
K0.383666X59.993Y22.122Z0.000R1.000;
K0.400499X55.680Y36.359Z0.000R1.000;
K0.426851X56.925Y24.995Z0.000R1.000;
K0.451038X59.765Y19.828Z0.000R1.000;
K0.465994X54.493Y14.940Z0.000R1.000;
K0.489084X49.220Y19.828Z0.000R1.000;
K0.499973X52.060Y24.994Z0.000R1.000;
K0.510862X53.305Y36.359Z0.000R1.000;
K0.533954X48.992Y22.122Z0.000R1.000;
K0.548910X44.814Y16.865Z0.000R1.000;
K0.573096X38.802Y12.551Z0.000R1.000;
K0.599447X32.911Y8.521Z0.000R1.000;
K0.616280X29.152Y14.535Z0.000R1.100;
K0.635094X28.040Y9.267Z0.000R1.000;
K0.644687X21.364Y4.830Z0.000R3.000;
K0.651784X25.768Y15.447Z0.000R5.000;
K0.668304X19.539Y20.391Z0.000R1.000;
K0.682958X17.097Y28.512Z0.000R1.000;
K0.711087X25.537Y33.750Z0.000R2.000;
K0.730703X16.602Y30.496Z0.000R1.000;
K0.743865X14.199Y39.803Z0.000R1.000;
K0.756479X8.668Y47.408Z0.000R1.000;
K0.772923X3.000Y63.794Z0.000R1.000;
K0.806926X18.465Y67.084Z0.000R1.000;
K0.849130X31.197Y58.572Z0.000R1.000;
K0.870652X39.411Y51.358Z0.000R1.000;
K0.914534X52.204Y44.971Z0.000R1.200;
K0.963883X52.904Y49.614Z0.000R1.000;
K0.985023X53.478Y52.139Z0.000R1.000;
K0.991714X54.492Y52.139Z0.000R1.000;
K1.000000;
K1.000000;
K1.000000;
K1.000000;
```

G0Z10.;
G0Z50.;
G91G28Z0.0;
G91G28Y0.0 ;

The operation result is shown as the figure below:

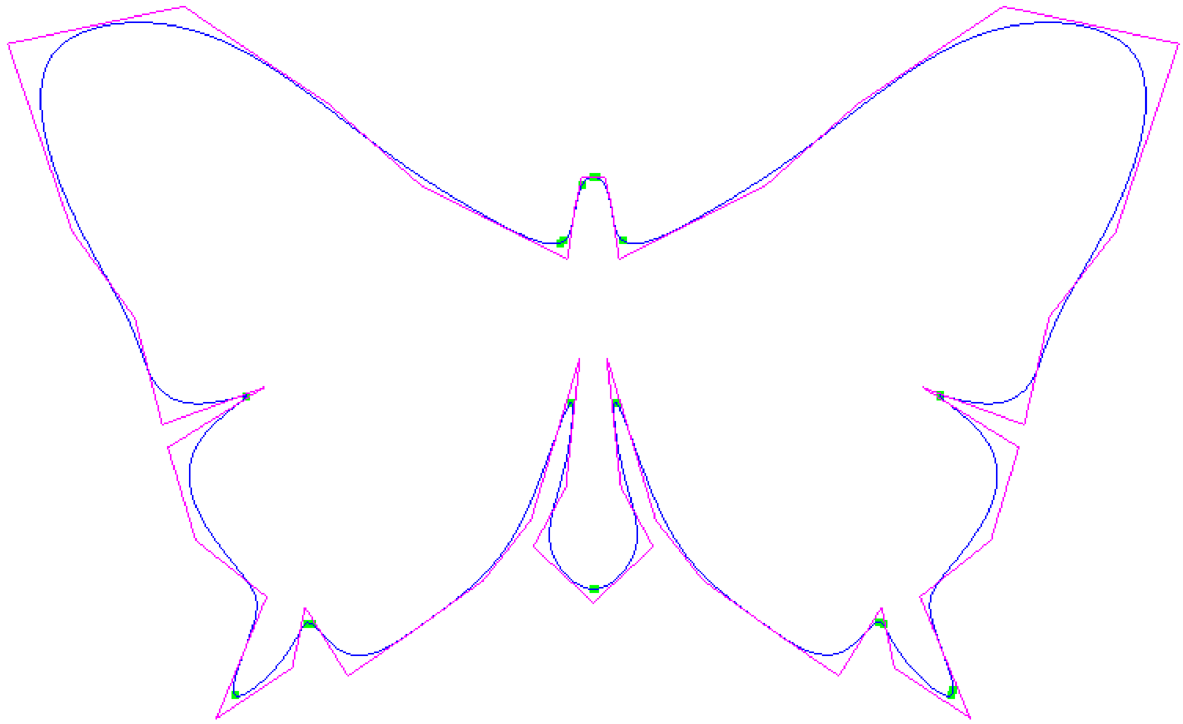


Fig. 3-2-16

3.2.6 High-speed machining function in high precision

3.2.6.1 High-speed machining mode (G05)

Format:

G05 P1 High-speed machining mode ON

G05 P0 High-speed machining mode OFF

Function:

The function is to pre-read many blocks and to process, so the minute line segment program can run smoothly at high speed.

Explanations:

1.In the high-speed machining mode, the program actual running path is different with the programmed path, and the difference maximum value is limited by parameter #1473.

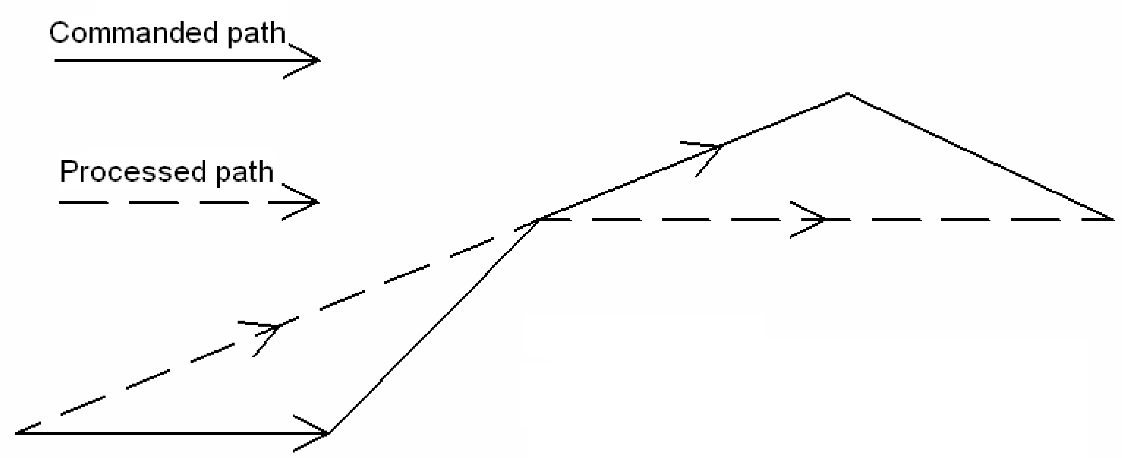


Fig. 3-2-17

2. It is not allowed to run the program more than three axes in the high-speed machining mode, and the mode invalid during five-axis machining.
3. In the high-speed machining mode, it is suggested to switch on G05 P1 and switch off G05 P0 blocks in the single block.

Example:
G91 G28 Z0.0
G00 G90 G54 X0 Y0 S3000 M03
Z20.
G05 P1（High-speed machining mode is ON）
G01 X0.1 Y0.1 Z0.1 F1000
:
:
X0.23 Y0.23 Z0.23
X0.45 Y0.45 Z0.45
:
G05 P0（High-speed machining mode is OFF）
M30

3.2.7 Dwell (G04)

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

Function:
By specifying a dwell with G04, the execution of the next block is delayed by the specified time.

Explanations:
1. As G04 is non-modal command, it is only effective in the current line.
2. When the tool stop is specified with G04, the execution of the next block is delayed by the specified time. In addition, a dwell can be specified to make an exact stop check in the cutting mode (G64 mode).
3. When P and X are specified simultaneously, P is effective.
4. Alarm No.006 will occur if the value specified by P and X is negative.
5. When X/P is specified as zero, the system executes the exact stop.

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

Command value range	Dwell time unit
0～9999.999	s

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

Command value range	Dwell time unit
0～9999999	0.001 s

Example: Dwell 3.8s

G04　X3.8 or G04　P3800；

3.2.8　Single Direction Positioning（G60）

Format: G60 X_ Y_ Z_

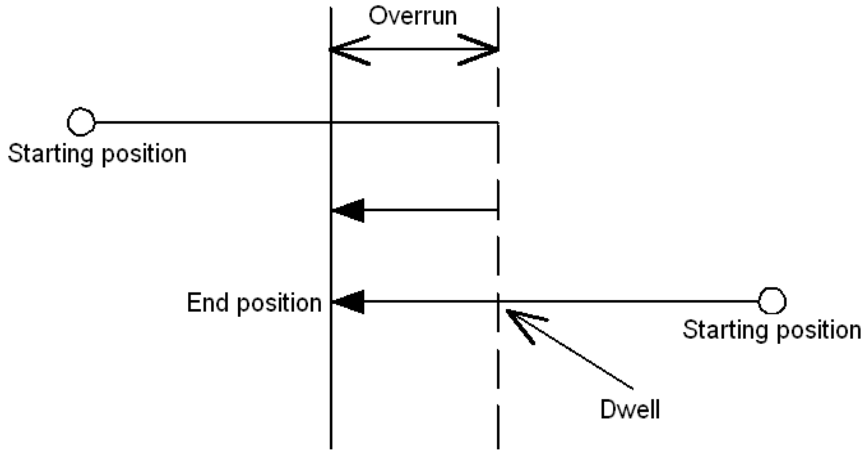


Fig. 3-2-18

Function:

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

Explanations:

Parameters X, Y and Z: For an absolute command, the coordinates of an end position, and for an incremental command, the distance the tool moves.

In the figure above, the marked overrun can be set by parameter No: 1880, and the defaulted dwell time is 1s. The positioning direction can be set by the positive or negative value of overrun. See system parameters for details.

System parameters:

No:1880	Overrun on X axis
No:1880	Overrun on Y axis
No:1880	Overrun on Z axis
No:1880	Overrun on 4th axis
No:1880	Overrun on 5th axis

Example:

G90 G00 X-10 Y10;
G60 X20 Y25;   (1)

In the case that the parameter No:1880 is set to（-8, 5, 0, 0, 0）, for statement (1), the tool path is AB →1s dwell→BC.

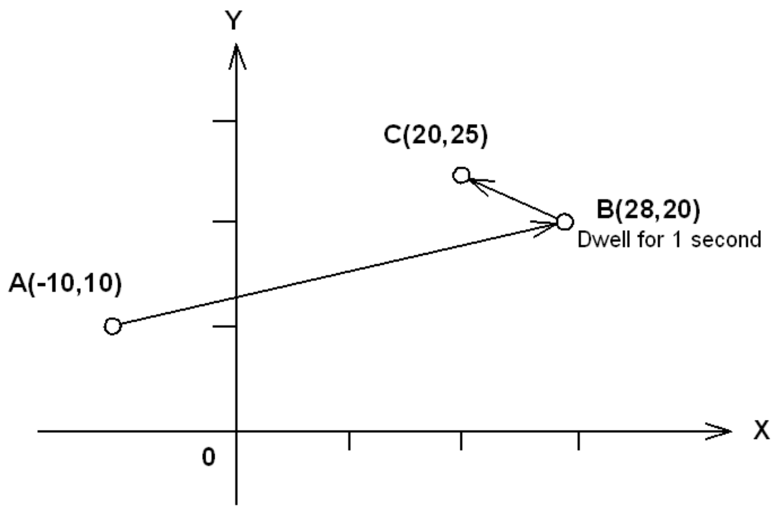


Fig. 3-2-19

G60, which is a one-shot G-code, can be used as a modal G-code in group 01 by setting1 to the parameter (No:1870#0 MDL）. While modal G60 can't cancel the canned cycle, this setting can eliminate specifying a G60 command for every block and it can be cancelled with other G codes in group 01. Other specifications are the same as those for a non-modal G60 command.

Example:

<table>
<tr><td colspan="2">When one-shot G60 commands are used(No:1870#0=0)</td><td colspan="2">When modal G60 command is used (No:1870#0=1)</td></tr>
<tr><td colspan="2">G90</td><td>G90 G60</td><td>Single direction positioning mode</td></tr>
<tr><td>G60 X10 Y10</td><td rowspan="3">Single direction positioning valid</td><td>X10 Y10</td><td rowspan="4">Single direction positioning valid</td></tr>
<tr><td>G60 X30 Y30</td><td>X30 Y30</td></tr>
<tr><td>G60 X80 Y80</td><td>X80 Y80</td></tr>
<tr><td>X50 Y50</td><td>Single direction positioning mode cancel</td><td>X50 Y50</td></tr>
</table>

Notes:

1. During canned cycle for drilling, no single direction positioning is effected in Z axis.
2. No single direction positioning is effected in an axis for which no overrun has been set by the parameter.
3. When the move distance 0 is commanded, the single direction positioning is not performed.

3.2.9 Skip Function G31

3.2.9.1 Ordinary Skip

Format: G31 X_Y_Z_

Function:

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

Explanations:

1. As a non-modal code, G31 is effective only in specified blocks.

2. If G31 command is issued while cutter compensation is applied, an alarm is displayed. Cancel the cutter compensation before the command is specified.

3. Disable feedrate override, dry run, and automatic acceleration/deceleration (however, these functions are valid by setting the parameter No.1940#7(SKF) to 1) when the feedrate per minute is specified; and allowing for an error in the position of the tool when a skip signal is input. These functions are enabled when the feedrate per revolution is specified.

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

1) The next block is an incremental command.

Incremental movement is performed from the break point

Example: G31 G91 X100.0;

Y50.0;

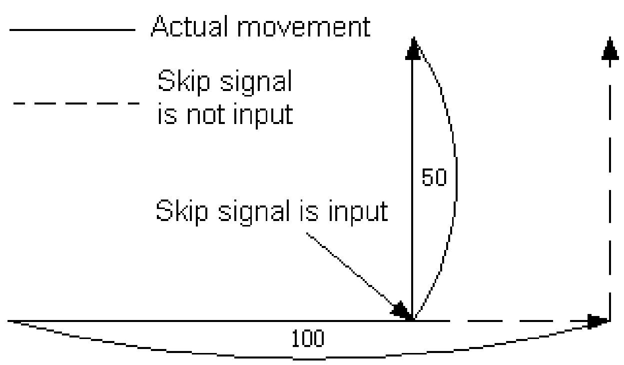


Fig. 3-2-20

2) The next block is an absolute command for one axis

In the next block, the commanded axis moves to the specified position, the unspecified one stays at the position where the skip signal is input.

Example: G31 G90 X200.0;

Y100.0;

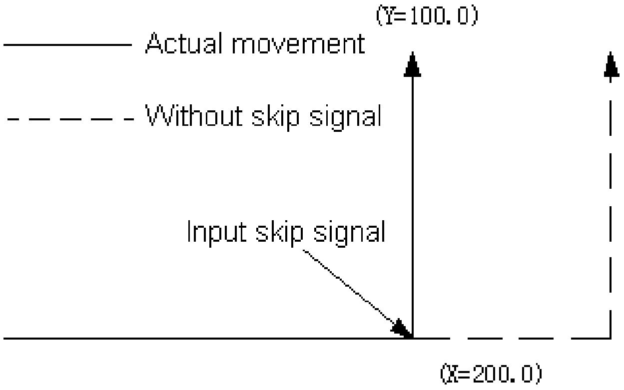


Fig. 3-2-21

3) The next block is an absolute command for two axes

The next block moves to the specified position wherever a skip signal is input.

Example: G31 G90 X200.0;

X300.0 Y100.0;

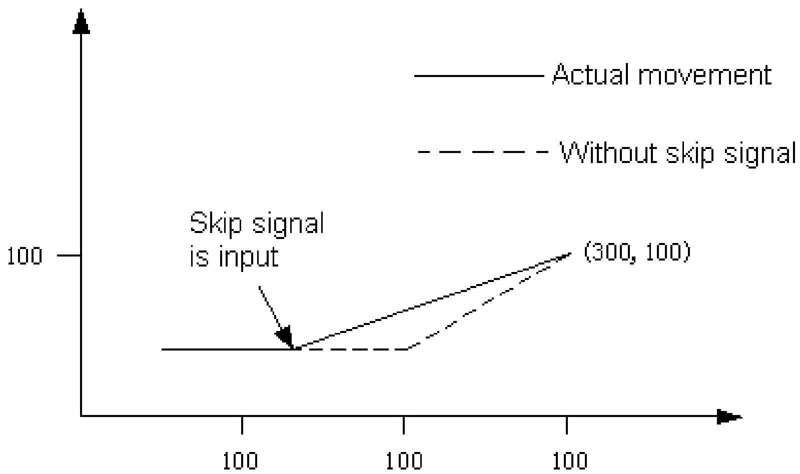


Fig. 3-2-22

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

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

b) To set by parameter

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

#5061…………………………Coordinate values of the 1st axis
#5062…………………………Coordinate values of the 2nd axis
#5063…………………………Coordinate values of the 3rd axis
#5064…………………………Coordinate values of the 4th axis
#5065………………………...Coordinate values of the 5th axis

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

a）Measure with the tool touching the sensor.

Notes:

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

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

3.2.9.2 High Speed Skip Signal

Format:

G31 X_Y_Z_;

G31: One-shot G code（It is effective only in the block in which it is specified.）

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

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

3.2.10 System Parameter On-line Modification (G10)

Format:

G10L50: Parameter entry mode setting
N_R_: For parameters other than the axis type
N_P_R_: For axis type parameters
⋮
G11: Parameter entry mode cancel

G10L51: Screw pitch compensation entry mode setting

Function:

It can modify parameters and screw-pitch error compensation data through program on-line rewriting.

Explanations:

N_ ; Parameter No. (4 digits) or compensation position No. for pitch errors compensation

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

P_; Axis No:1~5（Axis type parameters）

Bit No: 0~7（Bit type parameter）(If it is the axis bit type parameter: 0~39)

Screw pitch positive and negative: 0 positive; 1 negative

Additional explanations:

1．Parameter setting value（R_）: When it is a bit parameter, R can only be 1 or 0.

2．Axis number (P): Specify an axis number （P_）from 1 to 5 (at most 5 axes) for an axis type parameter. If P is not specified, the parameter of the 1st axis is rewritten. The control axes are numbered in the order in which they are displayed on the CNC screen. Bit parameters are from 0 to 7, from the low to the high bit; the axis type bit parameter is from 0 to 39, from the low to the high based on the control axis (0~7, 8~15, 16~23, 24~31 and 32~39).

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

Notes:

1. G10 block should be commanded independently. Do remember to cancel the parameter input mode with G11 after executing G10 (G10 is modal code only in L50 mode) to avoid affecting the program normal running.

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

3. The parameters which can be rewritten by G10 include: 1.1020~1021, 1031~1046, 1605~1642, 1801~1930, 1933~2034, 2112, 2113 and 2600~2653.

4. The parameters unable to be rewritten by G10 or without authority to rewrite for G10, the system alarms.

5. When G10 is executed in L50 mode, if the other non-modal command is performed, L50 mode is also cancelled.

6. The canned cycle mode should be cancelled before parameter input; otherwise, an alarm will occur.

Explanations about G10 in modal state and non-modal state:

1. G10 is a G modal code only in L50, while one-shot in L2, L10, L11, L12, L13, L20 and L51.

2. G10 and L2, L10, L11, L12, L13, L20 and L51 should be executed in the complete specified command format; G10 and L should be specified in each line.

3. When G10 is modal in L50 mode, G11, L2, L10, L11, L12, L13, L20, L51 and other one-shot G codes of the same group can cancel the modal.

Example:

1. Set bit 5 (SEQ) of the bit type parameter No:1

G10 L50 N1 P5 R1; SEQ is set as 1
G11；Cancel parameter inputting mode

2.G10 L50 N1810 R0.03； Rewriting the arc radius allowance error

N1862 P3 R10；Rewriting Z axis scaling
G11；Cancel parameter entry mode

3.2.11 Workpiece coordinate system G54～G59

Format:G54～G59

Function:

The workpiece coordinate system is relative to the machine coordinate system, CNC normally is with six standard workpiece coordinate systems（G54～G59）for the user setting. Once the workpiece coordinate systems are set, the following programs are executed in the selected coordinate system, each coordinate system is determined by the set distance from each axis reference point (the machine fixed point) to their own coordinate origin (workpiece origin offset amount). Please refer to fig. 3-4 for details:

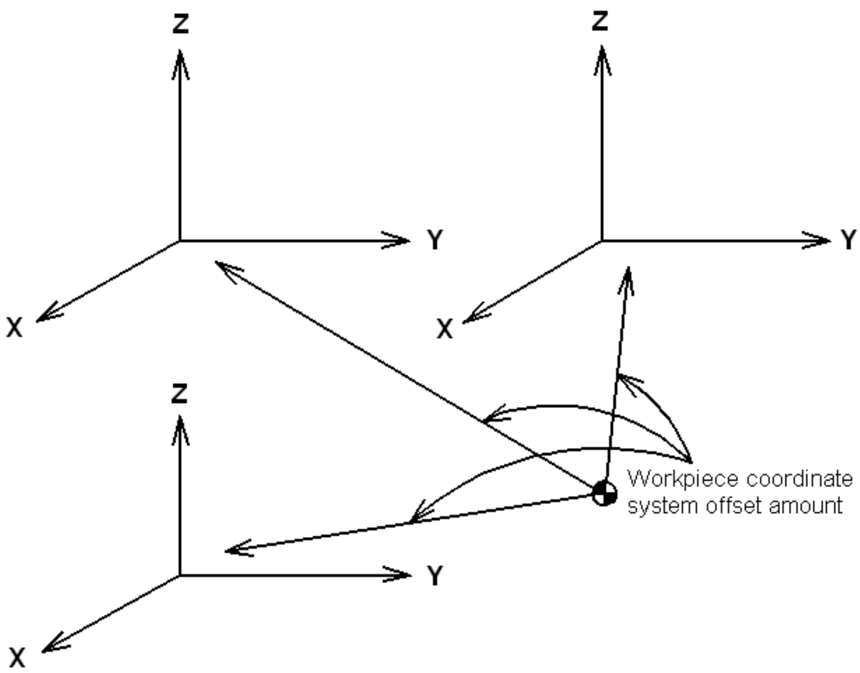


Fig. 3-2-23

Do not set a coordinate system with G92 command when the above coordinate systems are in use. G92 will replace the coordinate system set by G54～G59 if G54~G59 and G92 are used meanwhile; Therefore, they are not used simultaneously.

Explanations:

1．No command parameters;

2．Up to 6 workpiece coordinate systems can be set in system, each one can be selected by its corresponding commands (G54~G59).

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

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

Example:

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

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

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

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

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

With command G10 L2 Pp X_Y_Z_

P=0: External workpiece zero offset value.

P=1～6: Workpiece zero offset of the coordinate systems 1 to 6.

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

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

An external workpiece zero offset or workpiece zero offset can be changed by G10; if the currently specified workpiece coordinate system is changed, it will become valid in the following commands.

About changing the workpiece coordinate system in JOG mode, please refer to the points for attention in Appendix lists1 and 12.

The workpiece coordinate system is change in JOG mode or G10, the absolute coordinate and the relative coordinate won't be refreshed immediately while refreshed during running.

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

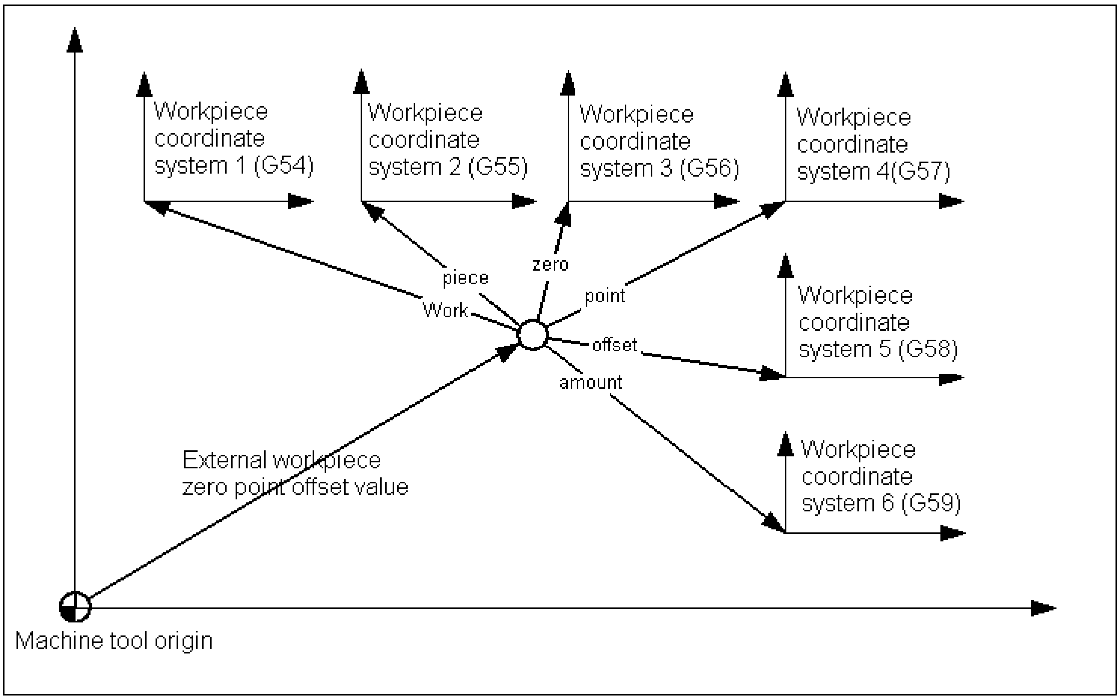


Fig. 3-2-24

As shown in Fig. 3-2-24, the machine returns to machine zero by manual zero return function after power-on. The machine coordinate system is set up based on this machine zero, thus machine reference point is generated and the workpiece coordinate system is set. The corresponding values of offset data parameter No.1040～1046 in workpiece coordinate system indicate the whole offset amount of the 6 workpiece coordinate systems. The 6 workpiece coordinate system origins can be specified by entering coordinate offset in MDI mode or setting by data parameters No.1040～1046. These 6 workpiece coordinate systems are set up based on the distances from machine zero to their coordinate system origins.

Example :

N10 G55 G90 G00 X100 Y20;
N20 G56 X80.5 Z25.5;

In the example above, when N10 block is being executed, rapid-traverse positioning to the workpiece coordinate system G55 (X=100, Y=20) is performed. When N20 block is being executed, the absolute coordinate value is changed automatically to the coordinate value （X=80.5, Z=25.5）in workpiece coordinate system G56 at rapid traverse rate.

3.2.12 Additional workpiece coordinate systems (G54.1Pn）

Besides six workpiece coordinate systems (standard workpiece coordinate systems) selected with G54 to G59, 48 additional workpiece coordinate systems (additional workpiece coordinate systems) can be used.

Format: G54.1 Pn；or G54 Pn；

Pn： Codes specifying the additional workpiece coordinate systems
n ： 1～48
G54.1 P1…… Additional workpiece coordinate system 1
G54.1 P2 ...... Additional workpiece coordinate system 2
⋮
G54.1 P48 …… Additional workpiece coordinate system 48

Change the workpiece coordinate system zero point offset value by G10:

G10 L20 Pn IP_;
Pn: Codes specifying the workpiece coordinate system for setting the workpiece zero point offset value;
n: 1～48
IP_: Axis addresses and a value set as the workpiece zero point offset

Explanations:

1. Selecting the additional workpiece coordinate systems

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

2) A workpiece coordinate system, once selected, is valid until the other one is chosen.

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

4) The following commands become valid when the additional workpiece coordinate system zero point offset value in the current workpiece coordinate is changed with G10,.

2. Setting the workpiece zero point offset value in the additional workpiece coordinate system

As with the standard workpiece coordinate system, the following operations can be performed for a workpiece zero offset in an additional workpiece coordinate system:

1) The 

key can be used to display and to set a workpiece zero point offset value.

2) The G10 function enables the workpiece zero point offset value to be set by programming.

3) A custom macro allows a workpiece zero point offset value to be handled as a system variable.

4) Workpiece zero point offset data can be entered or output as external data. When an absolute workpiece zero point offset value is specified, the specified value becomes a new offset value. When an incremental workpiece zero point offset value is specified, the specified value is added to the current offset value to produce a new offset value.

Notes:

1. A P code must be specified after G54.1. If G54.1 is not followed by a P code in the same block, additional workpiece coordinate system 1（G54.1P1）is assumed. If a value not within the limited range is specified in a P code, an alarm (No.030) is issued: illegal compensation number. P codes other than workpiece offset numbers can't be specified in G54.1 block.

E.g: G54.1 (G54) P1000 G04

2. G54.1 (G54) P1～G54.1 (G54) P48 shares one block with canned cycle, the alarm #12 occurs.

3.2.13 Selecting a Machine Coordinate System (G53)

Format: G53 X_ Y_ Z_

Function:

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

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

When manual reference position return is performed after CNC system power-on, a machine coordinate system is set so that the reference position is at the coordinate values of (＠， ＆).

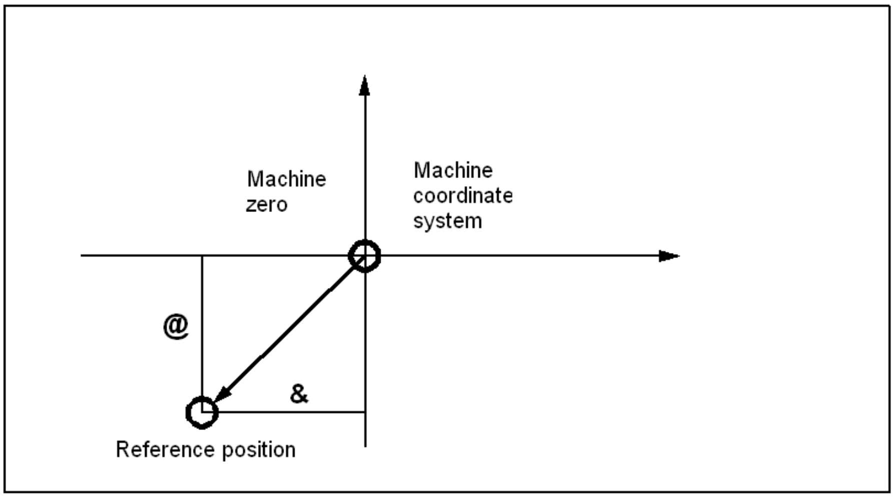


Fig. 3-2-25

Explanations:

1. When the G53 command is specified and the tool moves to the position at the rapid traverse rate, the tool compensation (the tool length offset and the cutter compensation) are cancelled temporarily.

2. Since the machine coordinate system must be set before the G53 command is specified, at least one manual reference position return (zero return should be operated in the JOG mode) or automatic reference position return by the G28 command must be performed after the power is turned on. This is not necessary when an absolute-position detector is attached.

3.2.14 Floating Coordinate System (G92)

Format: G92 X_ Y_ Z_
Function:

The workpiece coordinate system is to set some point of the tool (such as the tool nose) in the specified tool nose position. During the tool length offset, if the coordinate system is set by G92, the coordinate system is set with the coordinate value without offset by G92; during the tool radius compensation, the coordinate system is set with the coordinate value without compensation by G92.

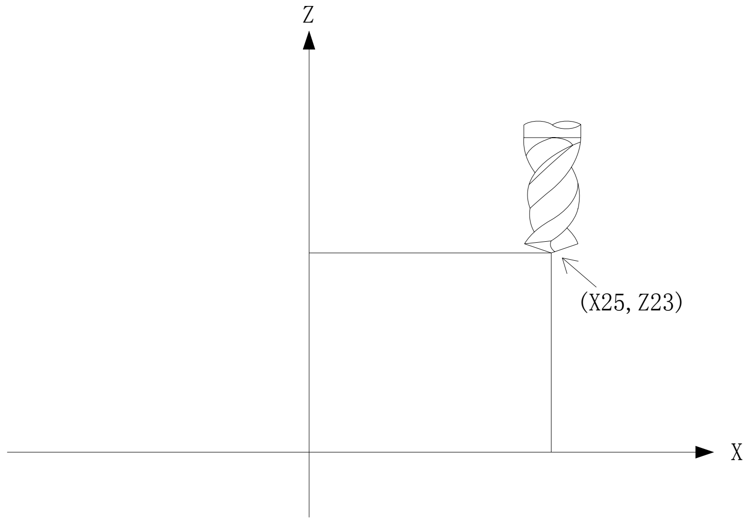


Fig. 3-2-26

G92　X25.0　Z23.0；

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

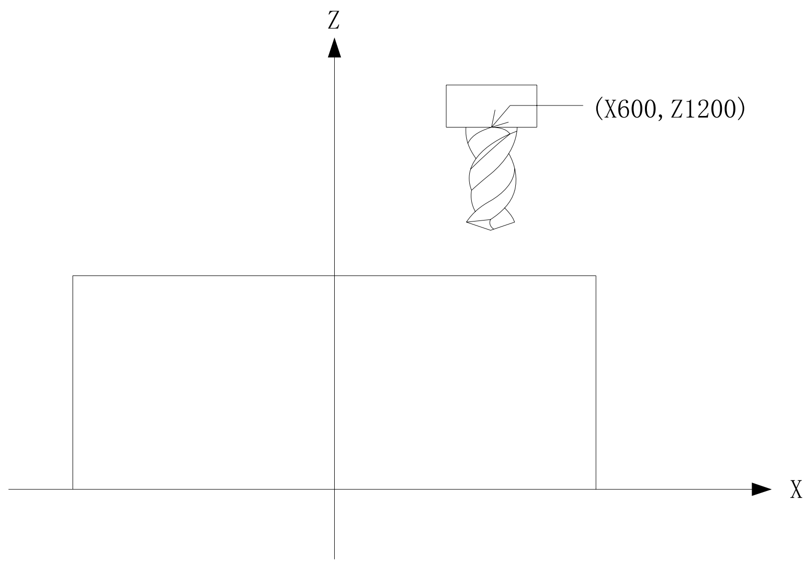


Fig. 3-2-27

G92 X600.0 Z1200.0;

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

Notes:

1. During offset, in the coordinate system set by G92, the tool coordinate value in the specified position doesn't include the offset value.

2. When the workpiece coordinate system is set, the tool position (such as the tool nose) is in the specified coordinate position. If G92 sets the coordinate system during the tool length offset, G92 sets the coordinate system through the coordinate value without the offset. About G92 in the cutter compensation mode, it will be introduced in "process G92 in the offset mode" in details.

3. After the external workpiece zero position offset value is set, the coordinate system isn't affected by the offset value when G92 sets the coordinate system. For example, when G92X100.0Z80.0 is commanded, the tool current position is specified by the coordinate system X=100.0, Z=80.0.

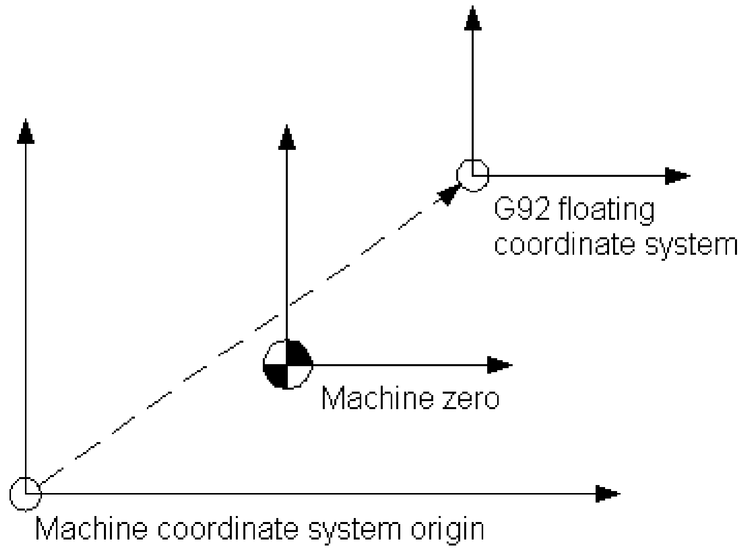


Fig. 3-2-28

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

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

1) Before system powers off

2) Before calling the workpiece coordinate system and the additional coordinate system

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

Restriction:

When G92 is cancelled through calling the workpiece coordinate system and the additional coordinate system, the machine should be performed zero return once or reference point return by

G28.

3.2.15 Local Coordinate System (G52)

When a program is created in a workpiece coordinate system, a child workpiece coordinate system can be set for easier programming. Such a child coordinate system is referred to as a local coordinate system.

Format:

G52 IP_; Setting the local coordinate system

¦

G52 IP0; Canceling the local coordinate system

IP_: Origin of the local coordinate system

Explanation:

By specifying G52 IP_;, a local coordinate system can by set in all the workpiece coordinate systems (G54 to G59). The origin of each local coordinate system is set at the position specified by IP_ in the workpiece coordinate system. When a local coordinate system is set, the move commands in absolute mode (G90), which is subsequently commanded, are the coordinate values in the local coordinate system.

The local coordinate system can be changed by specifying the G52 command with the zero point of a new local coordinate system in the workpiece coordinate system. To cancel the local coordinate system and specify the coordinate value in the workpiece coordinate system, match the zero point of the local coordinate system with that of the workpiece coordinate system.

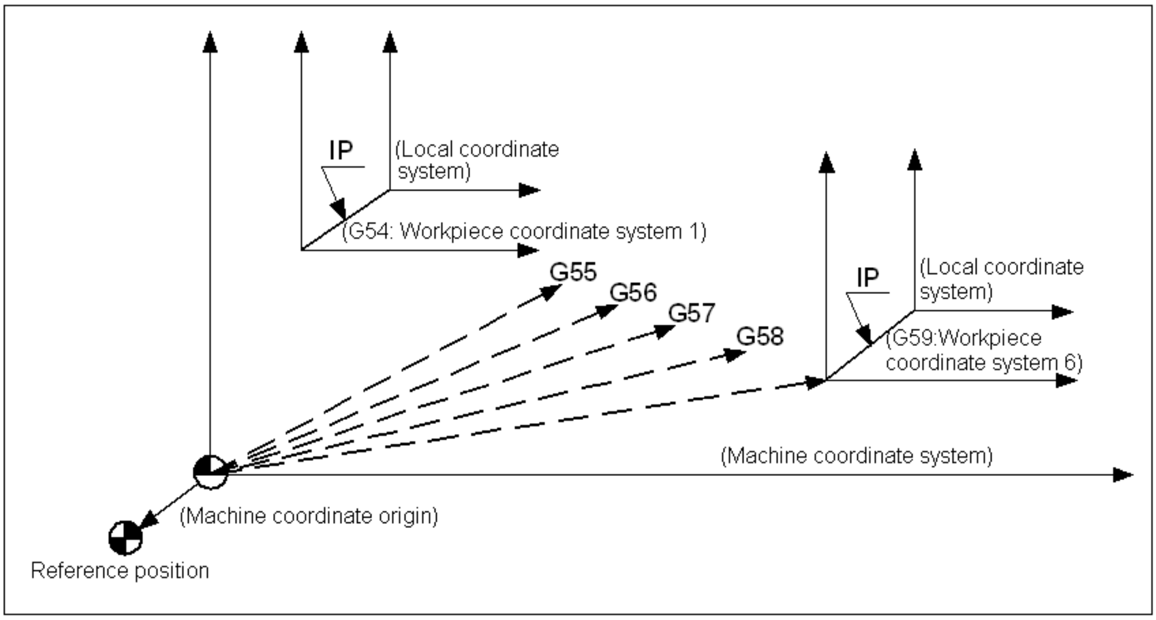


Fig. 3-2-29 Setting the local coordinate system

Warning:

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

2. The local coordinate system is cleared during resetting, emergency stop, M30 or mode switching.

3. In the radius compensation, G52 processing is same as G92.

Part Ⅰ Programming

3.2.16 Plane Selection G17/G18/G19

Format: G17/G18/G19

Function:

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

Explanations:

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

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

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

The plane is unchanged in the block in which G17, G18 or G19 is not commanded, and the addresses of basic three axes are omitted.

Example:

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

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

3.2.17 Polar Coordinates G16/G15

The end point coordinate value can be input in polar coordinates (radius and angle). The plus direction of the angle is counterclockwise of the selected plane first axis + direction, and the minus direction is clockwise. Both radius and angle can be commanded in either absolute or incremental command (G90, G91）.

Format:

G□□r**G○○ G16**	Starting the polar coordinate command (polar coordinate mode)
G○○ IP_	Polar coordinate command
……	
G15	Cancel the polar coordinate command (cancel polar coordinate mode)
G□□	Plane selection of the polar coordinate command （G17, G18 or G19）
G○○	Selecting the origin of the polar coordinate command
G90	Specifying the zero point of the workpiece coordinate system as the origin of the polar coordinate system
G91	Specifying the current position as the origin of the polar coordinate system
IP_	Specifying the addresses of axes constituting the plane selected for the polar coordinate system, and their values.

The 1st axis: polar coordinate radius

The 2nd axis: polar coordinate angle

Regulations of the polar coordinate origin:

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

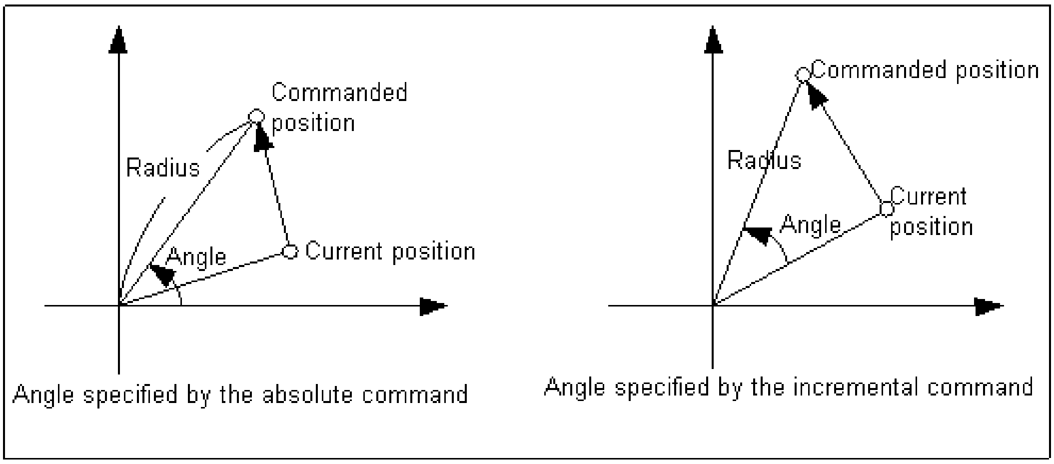


Fig. 3-2-30

2. In G91 incremental mode, the current position is set as the origin of the polar coordinates with G16 command.

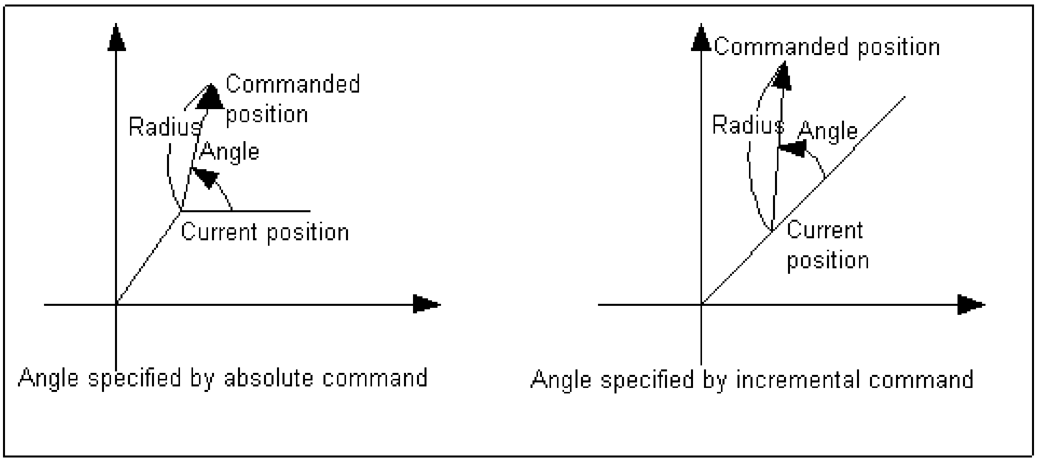


Fig. 3-2-31

Example: Bolt hole cycle

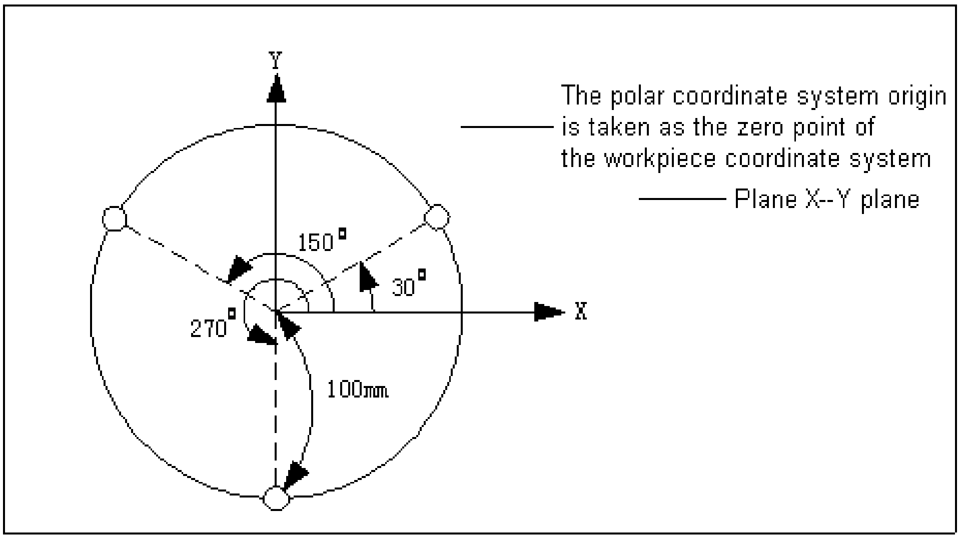


Fig. 3-2-32

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

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

G81 X100.0 Y30.0 Z-20.0 R-5.0 F200.0；Specifying a distance of 100mm and an angle of 30°

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

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

G15 G80； Canceling the polar coordinate command

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

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

G81 X100.0 Y30.0 Z-20.0 R-5.0 F200.0； Specifying a distance of 100mm and an angle of 30°

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

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

G15 G80； Canceling the polar coordinate command

Restrictions:

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

—Dwell （G04）

—Programmable data input（G10）

—Setting the local coordinate system（G52）

—Converting the workpiece coordinate system（G92）

—Selecting the machine coordinate system（G53）

—Stored stoke check （G22）

—Coordinate system rotation（G68）

—Scaling（G51）

3.2.18 Scaling in the Plane G51/G50

Format:

1）Equivalent scaling of each axis

G51 X_ Y_ Z_ P_ （X, Y, Z: Absolute commands for center coordinate value of scaling;
P: Equivalent scaling for each axis）

… Scaled machining block

G50 Cancelling the scaling

2）Different scaling of each axis:

G51 X_ Y_Z_ I_ J_ K_（X, Y, Z: Absolute commands for center coordinate value of scaling,
I, J, K: Scaling magnification for X, Y and Z axes respectively）

… Scaled machining block

G50 Canceling the scaling

Function:

The programmed figure, of which the specified point is set as the center by G51, can be scaled up or down with the same or different ratios of magnification. Moreover, specify G51 in a separate block and the scaling is cancelled by G50.

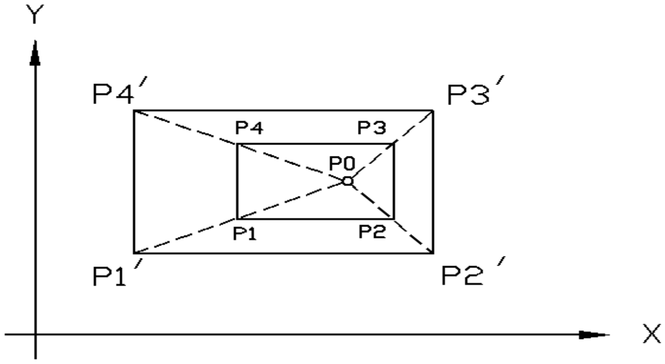


Fig. 3-2-33 Scaling (P1 P2 P3 P4→ $P'1\,P'2\,P'3\,P'4$)

P1～P4：	The figure of the machining program
P1′～P4′:	The figure after scaling
P0：	The scaling center

If P isn't specified, the scaling ratio can be set by the parameter; if X, Y and Z are omitted, the current tool position is taken as the scaling center.

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

Range:

P: 0～999999.9999

I, J, K: -999999.9999～+999999.9999

Explanations:

1. Scaling center: G51 is with 3 position parameters X_Y_Z_ and they can be selected. The positioning parameter specifies the scaling center of G51. If the positioning parameter doesn't specify, the system sets the current tool position as the scaling center. The current positioning mode is the absolute mode, and the scaling center is specified by the absolute positioning mode; the current positioning mode is the incremental mode, the scaling center is the tool current position. Moreover, in polar coordinate mode G16, the parameter commanded by G51 is indicated by the Cartesian coordinate system.

A/B: X axis scaling
C/D: Y axis scaling
O: Scaling center

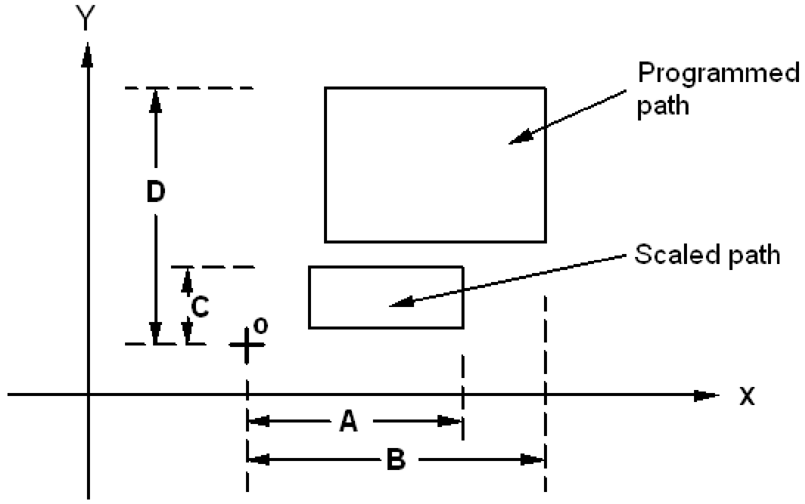


Fig. 3-2-34 Each axis scaling

Example:

G17 G91 G54 G0 X10 Y10;
G51 X40 Y40 P2;
G1 Y90; Parameter Y still uses the increment mode. In the mode, the scaling center is the current tool position.

1. Scaling: No matter it is G90 or G91 mode, the scaling is still indicated by the absolute mode. The scaling can be specified in the program, and also be set in the parameter. The data parameter No:1862 respectively corresponds to the magnifications of X, Y, Z, 4th and 5th axes; if there isn't the scaling magnification command, the scaling can be performed by data parameter No:1861. If the parameter values of I, J and K are negative, the mirror images are used for corresponding axes.

2. Setting scaling: Whether the scaling is valid is set by bit parameter N0:1850#3; whether the scaled ratio of each axis is valid is set by bit parameter N0:1850#6.

3. Cancelling scaling: After G50 cancels the scaling and follows with the movement command, it defaults the current tool position is the start position of the movement command when the scaling is canceled.

4. When different magnifications are applied to each axis and a circular interpolation is specified with radius R, it becomes Fig. 3-2-35 (in the example shown below, a magnification of 2 is applied to the X axis and a magnification of 1 is applied to the Y axis.).

```
G90G00X0.Y100.
G51X0.Y0.Z0.I2.J1.
G02X100.Y0.R100.F500
```

Above commands equivalent to the following ones:

```
G90G00X0.Y100.Z0.
G90G02X200.Y0.R200.F500
```

Fig. 3-2-35

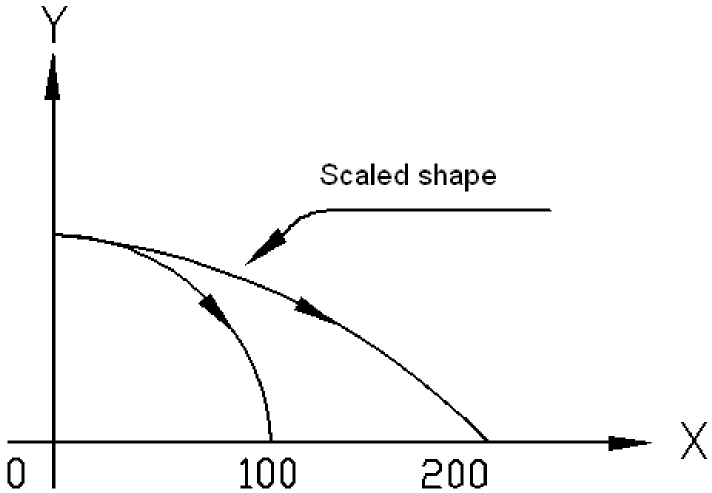


Fig. 3-2-36 Scaling of arc interpolation

5. The scaling is invalid for the cutter compensation value, the tool length compensation value and the tool offset value, which is shown as Fig. 3-2-37.

Program example:

```
G90 G50 G0 G92
G51 X25 Y25 P0.5
X-50 Y100
G41 D1 X0 Y50
X50 Y50
Y0 X50
X0 Y0
Y50 X0
X-50 Y100 G40
```

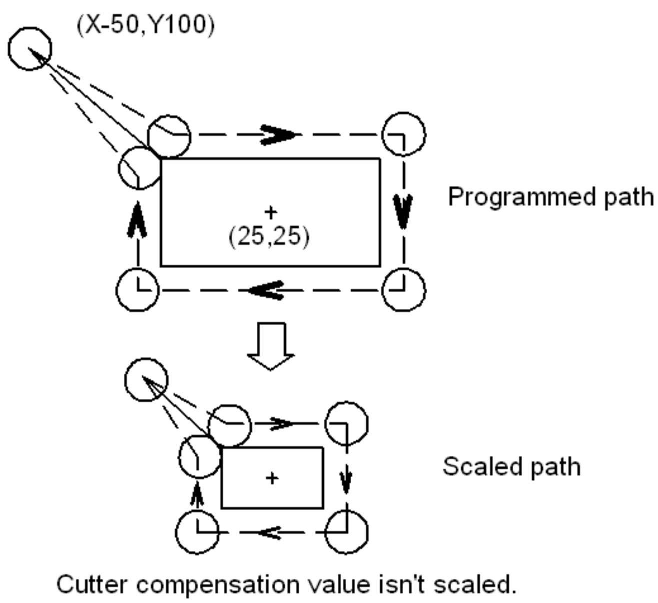


Fig. 3-2-37 Scaling during cutter compensation

6. Two of G51，G51.1 and G68 can't be in the same block; Otherwise, the system alarms.

7. In G51, G68 and G51.1 modes, the commands G27~G30, G52~G59 and G92 are not allowed; otherwise, #156 alarm is issued.

Notes:

1. The position displays as the coordinate value after scaling.
2. A parameter setting value is employed as a scaling magnification without specifying P.
3. Whether each axis scaling function is valid is set by the parameter.
4. Scaling function is invalid for the manual operation, but it is valid in DNC, Auto or MDI mode.
5. Scaling is not applicable to the following movements in case of the canned cycle.
   * Z and R position point in canned cycle.
   * Cut-in value Q and retraction value of peck drilling cycle (G83, G73).
   * X and Y displacement amount during the finish boring (G76) and the back boring (G87).

Example of mirror image program:

Main program

```
G00 G90 G50;
G92 X40 Y100 ;
M98 P9000;
G51 X50.0 Y50.0 I1 J-1;
M98 P9000;
G51 X50.0 Y50.0 I-1 J-1;
M98 P9000;
G51 X50.0 Y50.0 I-1 J1;
M98 P9000;
G50;
```

Subprogram

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

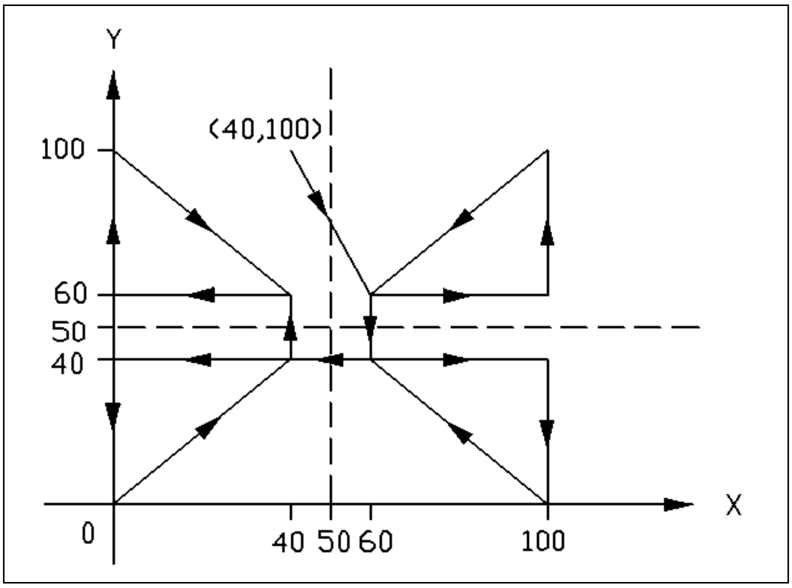


Fig. 3-2-38

3.2.19 Coordinate System Rotation G68/G69

A programmed shape can be rotated. When there is a pattern comprising some identical shapes for the machined workpiece, the coordinate rotation function can be used to program the subprograms of the shape and the subprogram is called after rotation.

Format ：

G17 **G68** X_ Y_ R_

Or G18 **G68** X_ Z_ R_

Or G19 **G68** Y_ Z_R_

G69

Function:

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

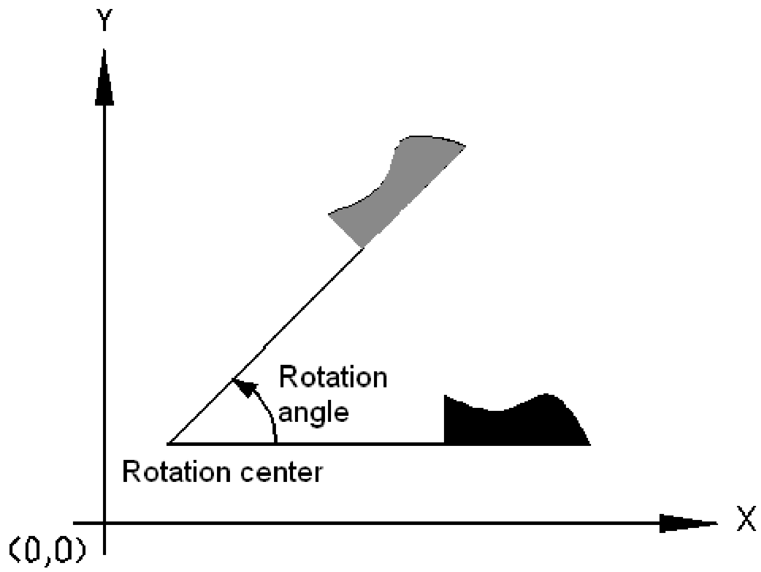


Fig. 3-2-39

Explanations:

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

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

R_: Angular displacement with a positive value indicates CCW rotation. Parameter bit No.1850#0 selects whether the specified angular displacement is always considered as an absolute value or is considered as an absolute or incremental value depending on the specified G code (G90 or G91).

Minimum input increment unit: 0.0001deg.

Valid data range: -360.0000~360.0000.

Additional Explanations:

1. G17, G18 and G19 for selecting the plane should be specified before G68 block.
2. When the rotation angle R is omitted, the value set by No.1860 is an angle default by the system.
3. After the coordinate system rotation, the cutter compensation, the tool length compensation and other compensation are operated.

4. In the scaling mode (G51), if the coordinate rotation command is executed, the coordinate value of the rotation center is also scaled. However, the rotation angle is not scaled. When the movement command is sent, the scaling is executed firstly, the coordinate rotation is performed.
5. Two of G51, G51.1 and G68 can't share the same block; otherwise, the system alarms.
6. In G51, G68 and G51.1 modes, G27~G30, G52~G59 and G92 are not allowed to command; otherwise, #156 alarm is issued.
7. If the rotation center is not specified, the system takes the current tool position as the rotation center. The current position mode is the absolute, the rotation center is specified by G68; the incremental, the rotation center is the current tool position.

In the coordinate rotation mode, the coordinate system before rotating calculates the commanded position and the coordinate is rotated.

Example:

```
N01 G92 G90 G0 X0 Y0 ;
N02 G01 X10 Y10 F6000 ;
N03 G68 X0 Y0 R45. ;
N04 Y14.142 ;
N05 G69 ;
```

The coordinate system （X, Y） before rotating calculates the commanded position and the coordinate is rotated. Therefore, X position which isn't commanded in N04 becomes （X10，Y10）, the commanded position is（X10, Y14.142）. Then, the system rotates toward the 45°position（X-2.9288，Y17.0710）.

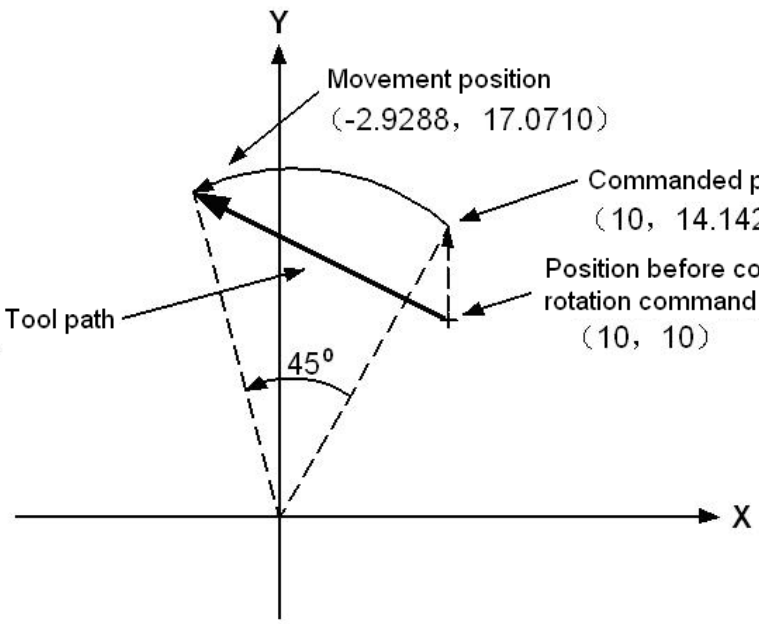


Fig. 3-2-40

Example:

Example 1: Rotation
G92 X-120 Y-120 G69 G17;
G68 X-120Y-120 R60;

G90 G01 X0 Y0 F200;
G91 X100;
G02 Y100 R100;
G3 X-100 I-50 J-86.603;

G01 Y-100;
G69 ;

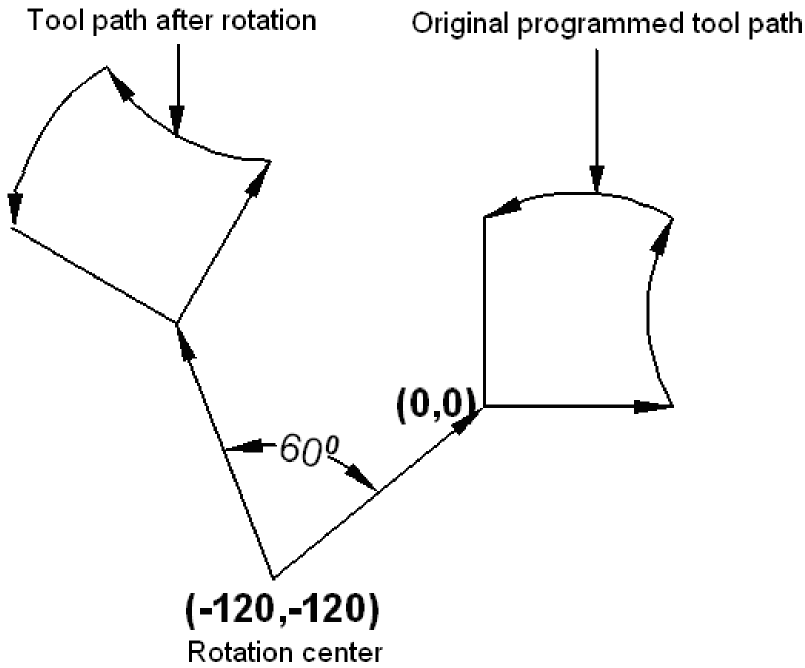


Fig. 3-2-41

Example 2 Cutter compensation and coordinate rotation

In cutter compensation mode, G68 and G69 command the rotation plane complying with the cutter compensation one.

G92 X0 Y0 G69 G01;
G42 G90 X20 Y20 F1000 D1;
G68 R-30 ;
G91 X50 ;
G03 Y30 R30 ;
G01 X-50 ;
Y-30 ;
G69 G40 G90 X0 Y0 ;
M30 ;

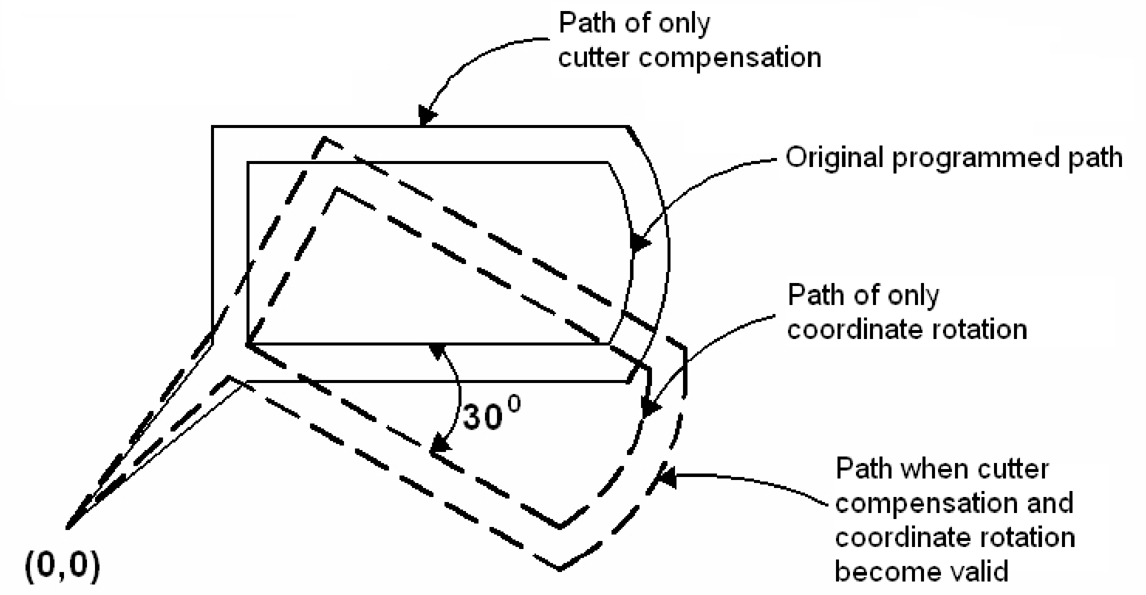


Fig. 3-2-41

Example 3: Scaling and rotating

```
G92 G90 X0 Y0
G51 X450 Y200P0.5;
G68 X300Y150R45;
G01 G90 X600 Y150
G91 Y150
X-300;
Y-150
X300;
G69 G50;
```

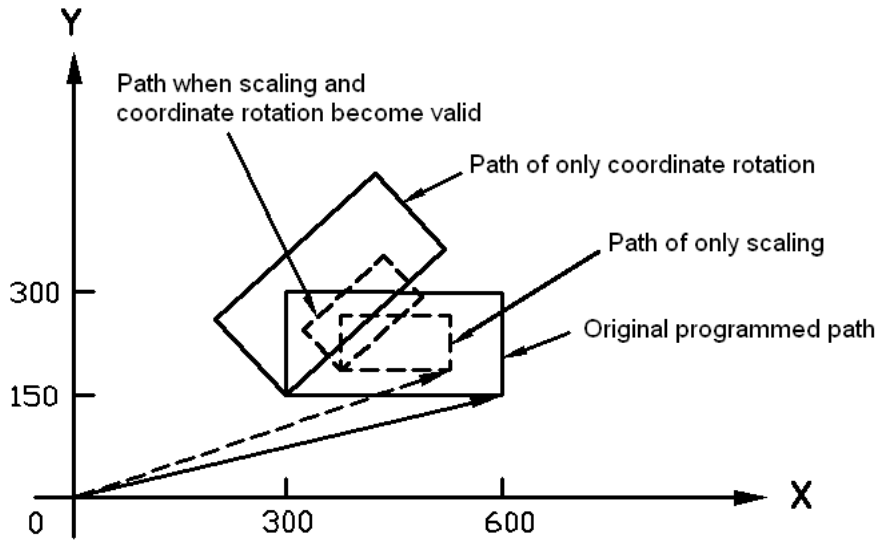


Fig. 3-2-42

Example 4: G68 is repeatedly used

One program is stored as a subprogram and the subprogram can be called for many times by changing the angle.

Based on the program (main program)

```
G92 X0 Y0 Z0 G69 G17;
G01 F200;
M98 P3155;                    (Calling the subprogram)
```

M98 P073166 ； （Calling for 7 times）
G00 G90 X0 Y0 Z0 ;
M30 ;

Subprogram3166
O3166;
G68 X0 Y0 G91 R45.0 ; (Relative to the rotation angle)
G90;
M98 P3155; (Subprogram O3166 calling subprogram O3155)
M99 ；

Subprogram3155
O3155;
G90 G01 G42 X0 Y-10 D1 ； (Setting right tool compensation)
X4.142 ；
X7.071 Y-7.071 ；
M99 ;

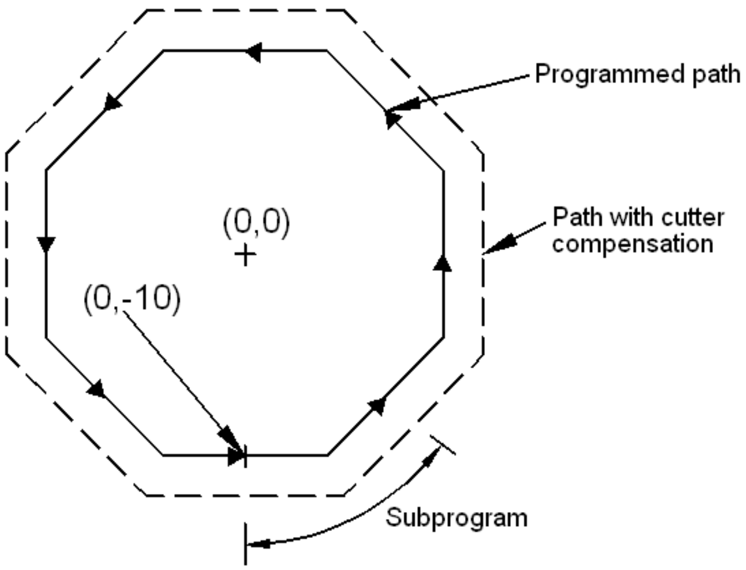
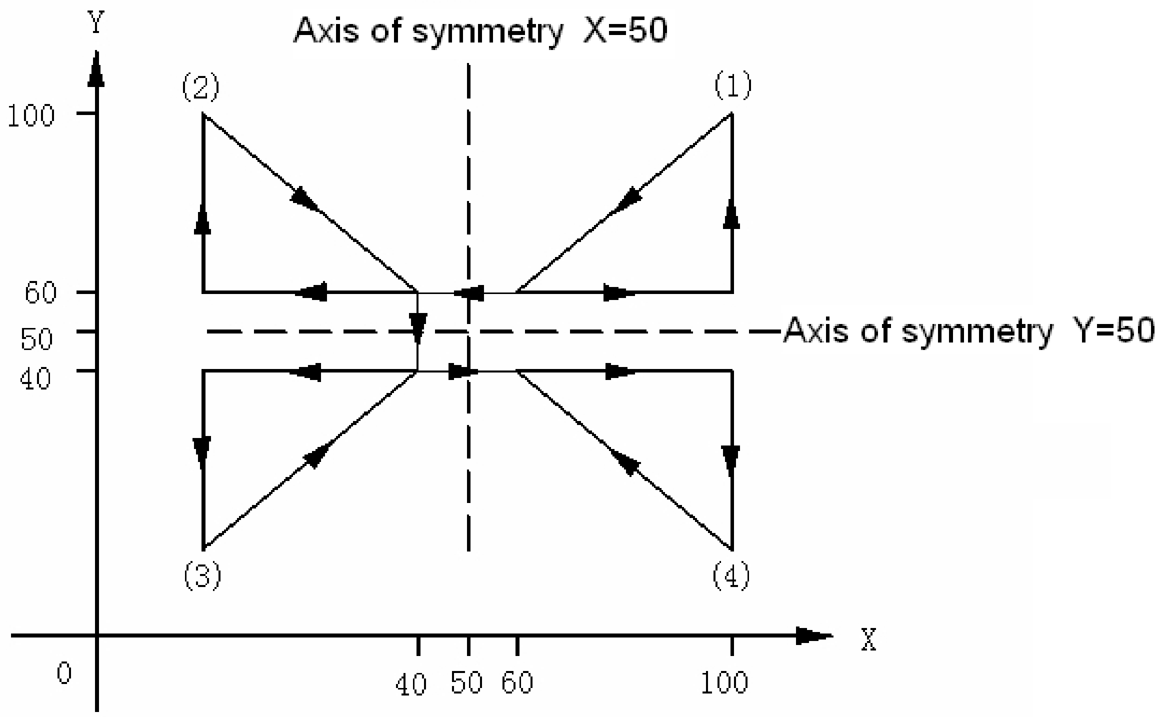


Fig. 3-2-43

3.2.20 Programmable mirror image (G50.1/G51.1)

The axis of symmetry specified in the program can produce a mirror image in the specified position of the program, which is shown as the following figure:



(1) The original program commands;
(2) The program commands for programmable mirror image applied in X50 position;
(3) The program commands for programmable mirror image applied in X50 and Y50 positions;
(4) The program commands for programmable mirror image applied in Y50.

Fig. 3-2-44

Format:

G51.1 IP_: Setting a programmable image

.
. } A mirror image of a command specified in these blocks is produced with respect to the
. axis of symmetry specified by G51.1 IP_;

G50.1 IP_: Cancelling a programmable mirror image;

IP_: Command producing a mirror image of the symmetry axis when specified with G51.1.

Axis of symmetry for producing a mirror image when specified with G50.1 can not move in any position.

Explanations:

1. Two of G51, G51.1 and G68 can not be in the same block; otherwise, the system alarms.

2. G27~G30, G52~G59 and G92 are not allowed to command in G51.1, G51 and G68 modes; Otherwise, alarm #156 is issued.

3. The current position mode is absolute and the mirror image center is specified by G51.1. When the position mode is incremental, the mirror image center is the current tool position.

4. The following commands will be changed if only one axis uses the mirror image on the specified plane:

Command	Description
Arc command	G02 and G03 is interchanged.
The cutter compensation or the tool nose radius compensation	G41 and G42 is interchanged.

3.2.21 Inch/Metric Conversion G20/G21

Format: G20: inch input
G21: mm input

Function: The program inch or metric system can be input.
Explanation:

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

Unit	G code	Least input unit
Inch (inch system)	G20	0.0001inch
mm (metric system)	G21	0.0001 ㎜

N10 G20；

N20 G92 X—Y—；

Pay special attention to the following contents are changed after metric/inch system conversion:

（1）Absolute coordinate display

（2）Relative coordinate display

（3）Unit of scale for manual pulse generator

Pay special attention to that the following contents are NOT changed after metric/inch system conversion, which is different with other milling machines.

1) Workpiece zero offset amount
2) Tool offset value
3) The units of all parameters are not changed
4) Pitch error compensation value
5) Machine coordinate value

Whether the metric and the inch system are automatically switched is set by parameter #2602 OIM during inputting these data. Whether the machine is metric or inch system is set by #1000 INM and the fixed data units are set by the parameters.

Notes:

1．Inch and metric input must not be switched during a program. G20/G21 must be specified before the program starts executing the machining command; otherwise, the system alarms.

2．When switching inch input to metric input and vise versa, the tool compensation value must be preset based on the least input increment.

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

4．When the least input increment and the least command increment systems are different, the maximum error is half of the least command increment. This error is not accumulated.

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

3.3 Reference Position G Codes

The reference position is a fixed position on a machine tool on which the tool can be positioned through the reference position return command. About the reference position, there are three command modes. For example, by G28, the tool is automatically moved to the reference position via an intermediate point along a specified axis, which is shown as fig. 3-14; By G29, the tool is automatically moved from the reference position to a specified point via an intermediate point along a specified axis.

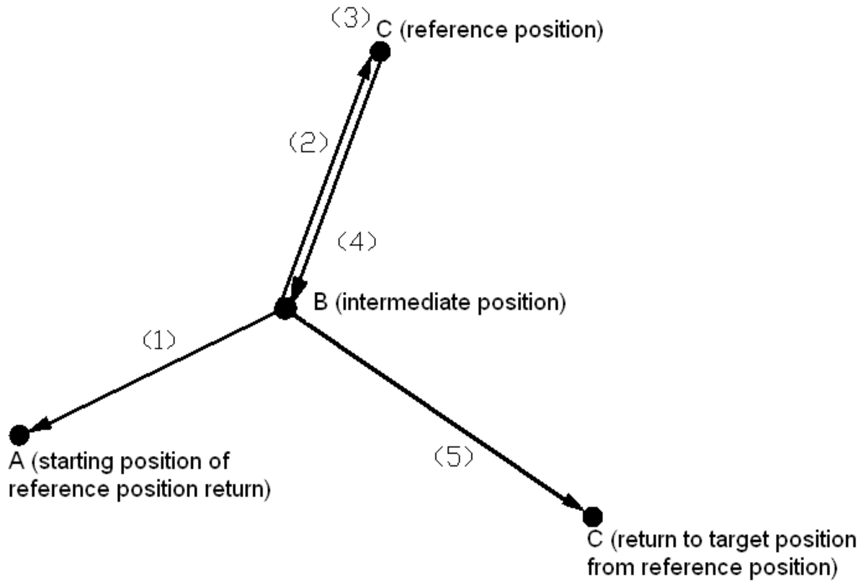


Fig. 3-3-1

3.3.1 Reference Position Return Check G27

Format: G27 X_ Y_ Z_

Function:

The reference position return check is executed by a G27 command; X_ Y_ Z_ specifies the command (absolute/incremental command) of the reference position.

Explanations:

1. If the tool is on the reference position, the reference position return indicator corresponding to controllable axis is ON. If the tool doesn't reach the reference position, an alarm occurs.

2. When the machine system is an inch system with metric input, the reference position return indicator is also ON even the tool programmed position is just shifted for 1μ from the reference position. This is because the least input increment is less than the least movement increment of the machine system.

3. In the offset mode, the position to be reached by the tool with G27 command is the position obtained by adding the offset value. Therefore, if the position with the offset value added is not the reference position, the indicator is OFF, but an alarm is displayed instead. Usually, offsets are cancelled before G27 is commanded.

3.3.2 Reference Position Return G28

Format: G28 X_ Y_ Z_
Function:

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

Explanations:

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

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

The movement of G28 block is introduced as below:

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

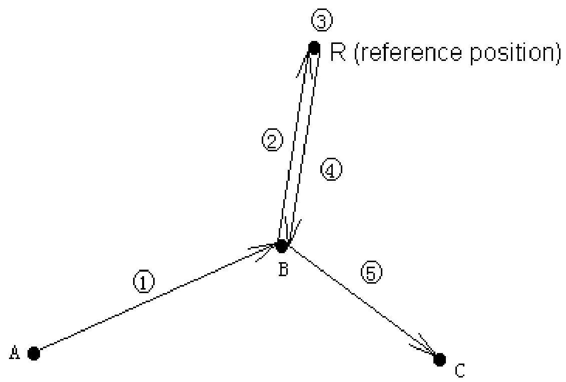


Fig. 3-3-2

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

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

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

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

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

Example 1:

N1 G90 G54 X0 Y15.；	
N2 G28 X45. ；	Set the intermediate point in X axis as X45.0 in G54 workpiece coordinate system via the intermediate point (45.0，15.0) and return to the reference position, and the machine coordinate of X45.0 is saved.
N3 G29 X35. ；	From the reference position via the point（45.0，15.0） and returning to point (35.0，15.0), namely, X axis returns to the target position independently.
N4 G01 X25.；	
N5 G28 Y65. ；	Y axis moves to the reference position, the machine coordinate of the intermediate point Y65.0 is saved. Because X axis hasn't been commanded, replace with the saved coordinate X45.0 commanded by G28.

Please pay attention to that the intermediate point is NOT（25，65）.

N6 G55； When the workpiece coordinate system is changed, the machine tool coordinate of the intermediate point is still the machine coordinate point (45.0，65.0) of G54 workpiece coordinate system.

N7 G29 X65. Y25.； From the reference position via the machine coordinate system intermediate point (45.0，65.0) of G54 workpiece coordinate system and return to the point (65.0，25.0).

G28 cancels the tool compensation temporarily. However, the command is normally used during automatic tool change; namely, after the reference position return, the tool is changed in the reference position. Therefore, for safety, the cutter compensation and the tool length compensation should be cancelled before executing this command.

Notes:

1. The coordinate value of the move command and the machine coordinate value of the intermediate point are stored in G28 block. For the other axes which are not commanded in G28, the previously specified coordinate value in G28 is taken as the machine coordinate value of the intermediate point for the axis.
2. When G28 command is specified and the reference position has been set after power-on, the movement from the intermediate point is same as the manual reference position return. In this case, the direction shifted from the intermediate point becomes the reference position return one specified in the parameter.
3. About the rotation axis, when G28 is specified, the move direction from the intermediate point to the reference position becomes the reference position return one set by the parameter. And the move amount should be in the range of 360°.

3.3.3 Automatic return from the reference position G29

Format: G29 X_ Y_ Z_

Function:

When G29 is commanded, the tool traverses from the reference position via the intermediate point specified by G28 command and returns to the specified point.

Explanations:

The tool can position on the specified point via the intermediate point with the function and the command always follows G28 command.

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

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

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

1. The movement of G29 block can be divided into the following steps (refer to figure 3-15):

(1) Position from the reference position to the intermediate point specified by G28 at the rapid traverse rate (point R→point B).

(2) Position from the intermediate point to the specified point at the rapid traverse rate (point B→point C).

2. G29 is one-shot command and only valid for the current block. Normally, after G28 command, return from the reference position command should be executed.

3. In G29 command format, the parameters X, Y and Z can be selected to specify the destination position for return from the reference position, such as point C in Fig. 3-3-3, and it can be indicated by the absolute or the incremental command. About the incremental programming, the commanded value specifies the incremental value away from the intermediate point. When some axes aren't specified, it means the relative intermediate point in the axis doesn't have any move amount. After G29, only the command with one axis returns independently, and other axes remains still.

4. In the cutter compensation state, G28 intermediate point from which G29 passes isn't with the offset.

Application examples of G28 and G29

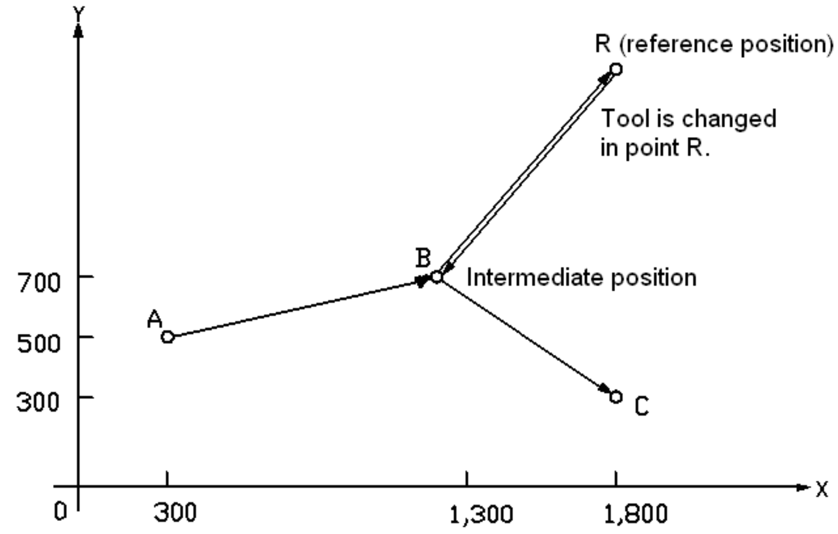


Fig. 3-3-3

Example 1:

See Fig. 3-3-3:

G91；
G28　X1000.0　Y200.0；（From A to B）
M06；
G29　X500.0　Y-400.0；（From B to C）

Example 2 :

G90 G0 X10 Y10；
G91 G28 X20 Y20；　Return the reference position via the intermediate point（30.0，30.0）of the machine coordinate

G29 X30 Y0；　Return from the reference position (60.0, 30.0) via the intermediate point（30.0, 30.0）specified by G28 previously. Pay attention to that in the incremental programming mode, the vector of X-axis direction is 60.

3.3.4　2nd, 3rd or 4th Reference Position Return G30

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

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

G30 { P2 / P3 / P4 } X_ Y_ Z_; （P2 can be omitted）

P2 : The 2nd reference position
P3 : The 3rd reference position
P4 : The 4th reference position

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

Explanations:

1．X_ Y_ Z_: Specifying the intermediate position commands (absolute /incremental command).

2．G30 command setting and restrictions are same as those of G28; about setting the 2nd , 3rd or 4th reference position, refer to data parameter No:1051～1053.

3.4 Canned cycle G codes

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

List 3-5 Canned cycles

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

The normal process of the canned cycle:

A canned cycle consists of a sequence of six movements (Fig. 3-4-1).

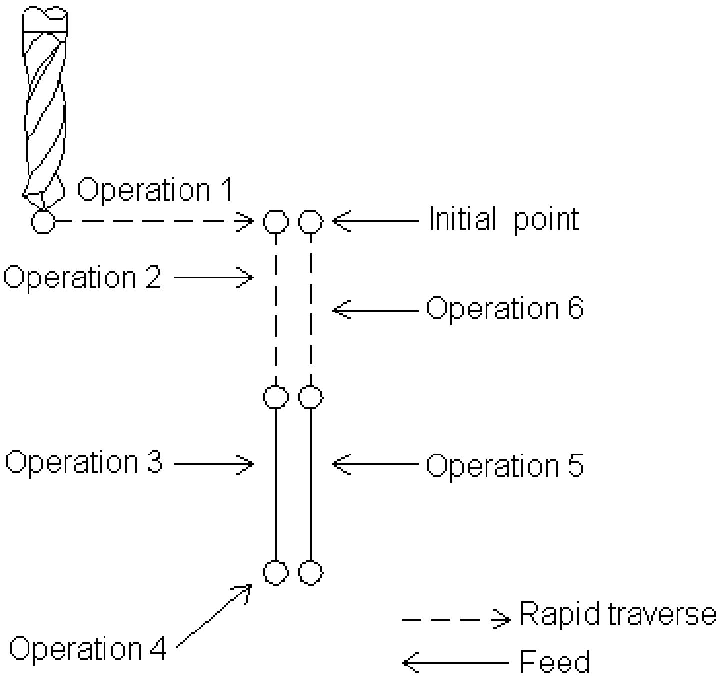


Fig. 3-4-1

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

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

① Data type
- G90 Absolute
- G91 Incremental

② Return to point level
- G98 Initial level
- G99 Point R level

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

Note: The initial level is the absolute value position in Z direction when the canned cycle cancel mode is switched into the canned cycle mode.

The initial level and point R level

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

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

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

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

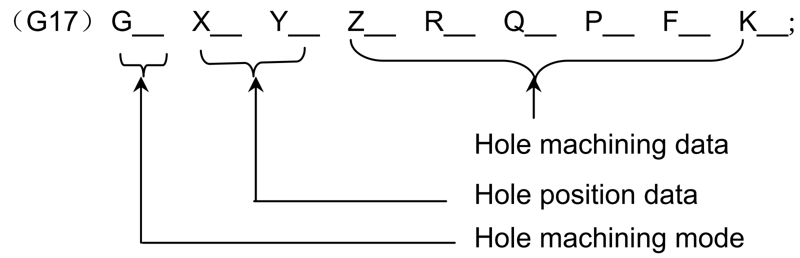


Fig. 3-4-2

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

List 3-6

Specified content	Parameter word	Remark
Hole machining mode	G	Refer to list 3-5, and pay attention to the above restrictions.
Hole position data	X，Y	Specify the position of a hole through the absolute or incremental value, and the control is same as that of G00 positioning.
Hole machining data	Z	The incremental value specifies the distance from point R to the bottom of a hole or the absolute value specifies the coordinate value of the hole bottom, which is shown as 3-17(A). The feedrate is the speed specified by F in movement 3; while in movement 5, it can be the rapid feed or the speed specified by F code based on the hole machining mode.
	R	The incremental value specifies the distance from the initial level to point R in figure 3-17(B), or the absolute value specifies the coordinate value of point R. The feedrate is the rapid feed during the movements 2 and 6. The initial level is taken as point R one if value R is lack.
	Q	Specify the cutting amount of each time in G73 and G83, or the translation amount (incremental value) in G76 and G87.

	P	Specify the dwell time at the bottom of a hole. The canned cycle command can be with one parameter P_ based on the different hole machining modes. And when P is less than 0, the system alarms. The parameter value in P_ specifies the dwell time after the tool reaches Z drilling depth. The unit is 0.001s.
	F	Specify the cutting feedrate.
	K	Specify the repeated times in K_ parameter value, and K is only valid in the specified block. And it can be omitted and one time is assigned by default. The maximum drilling times are 9999. When the times specified by K is greater than the max. repeated times, an alarm is issued. When it is specified as the negative value, and executed as its absolute value; if it is zero, the drilling isn't operated while the position is executed and the mode is changed.

Restrictions:

- The canned cycle G codes are the mode commands, G codes remain valid till the canned cycle cancel commands G80, G00, G01, G02 or G03 is specified.
- G codes of cancelling the canned cycle include G80, G00, G01, G02 and G03.
- Once the machining date are specified in the canned cycle, and they remain valid till the canned cycle is canceled. Therefore, at the beginning of the canned cycle, all the required hole machining data are specified. The following canned cycle just specifies the rewritten data.
- After the canned cycle is cancelled, the drilling hole axis can be switched; otherwise, the system alarms.
- The #167 alarm is issued when the canned cycle share the same block with the following G codes: G05, G07.1, G10, G22, G27, G28, G29, G30, G51, G50.1, G51.1, G52, G92, G65, G66, G68 and G53.
- When the canned cycle is with G20 or G21 in one block, the #48 alarm is issued during metric system/inch system switch; otherwise, the #167 alarm is issued.

Notes:

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

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

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

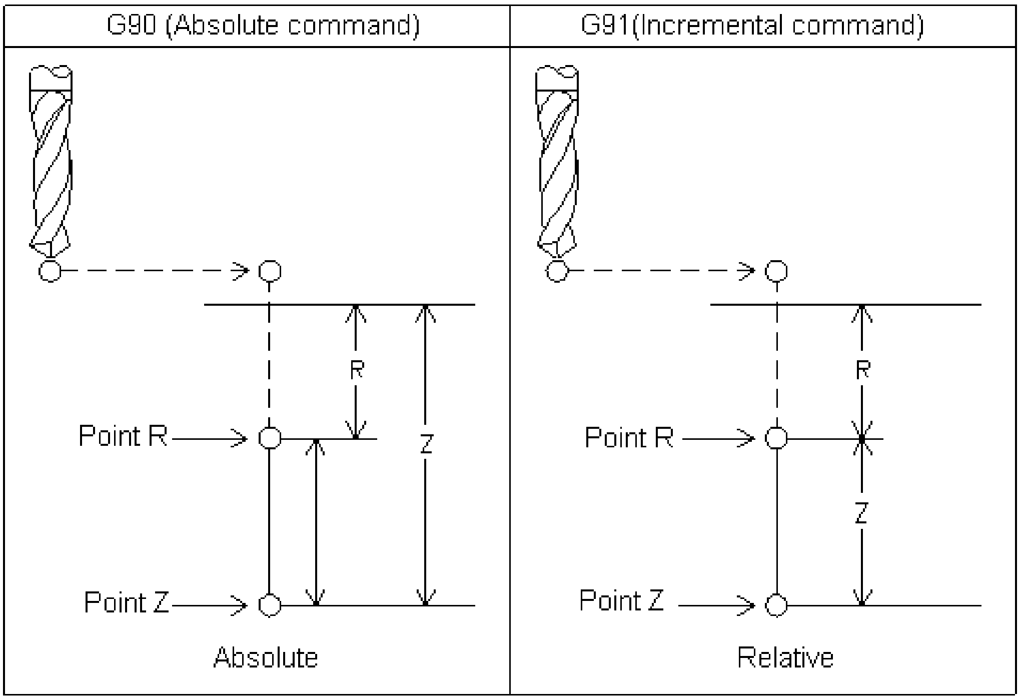
List 3-7

SR.NO	Specifying the data	Remark
①	G00X-Y-Z-M03S-;	Initial program point
②	G98/G99G81X-Y-Z-R-F-;	At the beginning, specify the required value for Z, R and F.
③	Y-;	The hole machining mode and the hole machining data are same as those specified in hole ②, so G81 and Z-R-F- all can be omitted. The hole position is moved to Y, the hole is machined for one time in G81 mode.
④	G82X-P-;	Comparatively, hole ③ position is just moved in X axis direction. The hole is machined in G28 mode, Z, R and F specified in ② and X, P specified in ④ are assumed as the hole machining data.

⑤	G80	The hole machining isn't operated. All the hole machining data are canceled.
⑥	G85X-Z-R-P-;	All the data are canceled in ⑤. Therefore, Z and R should be specified again, F is same as that specified in ②, so it can be omitted. P isn't required in the block but saved.
⑦	X- Z-;	Compared with ⑥, just Z value is different in the hole machining, and the hole position is just moved in X axis direction.
⑧	G89X-Y-;	Z specified in ⑦, R specified in ⑥, F specified in P and ② are assumed as the hole machining data, and the hole is machined in G89 mode.
⑨	G01X-Y-;	Cancel the hole machining mode and the hole machining data.

A. The absolute and incremental commands of the canned cycle G90/G91

Along with the move distance of the drilling axis, the change of G90 and G91 is shown as fig. 3-3-6. (Normally, programmed with G90, if it is programmed by G91, Z and R are handled as the negative value.)



(A) (B)

Fig. 3-4-3

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

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

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

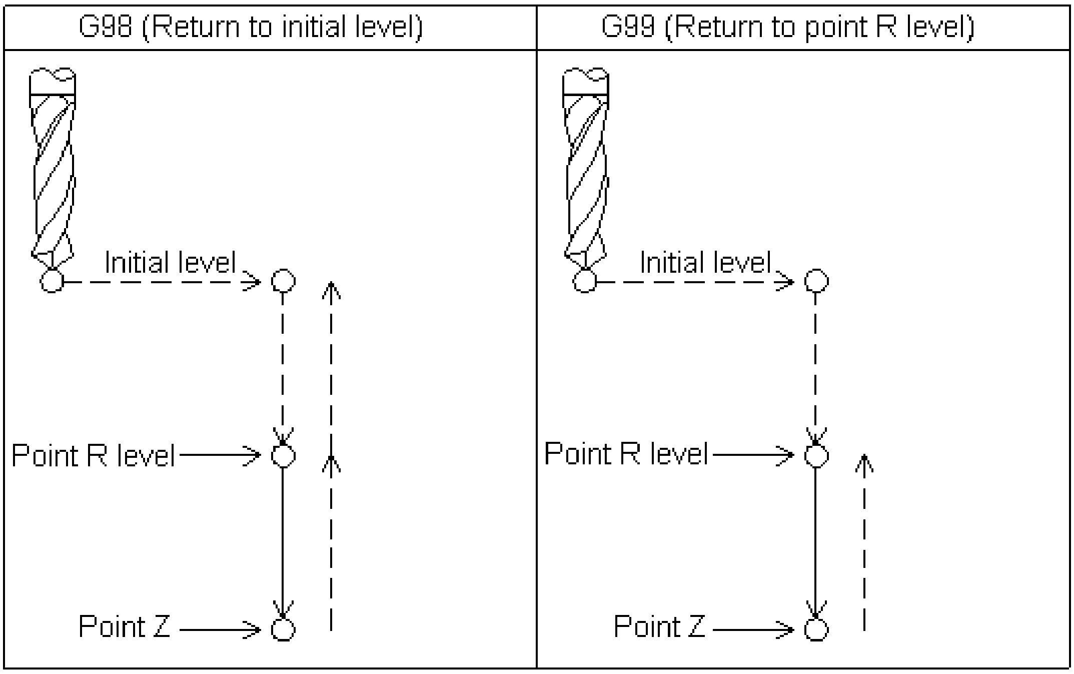


Fig. 3-4-4

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

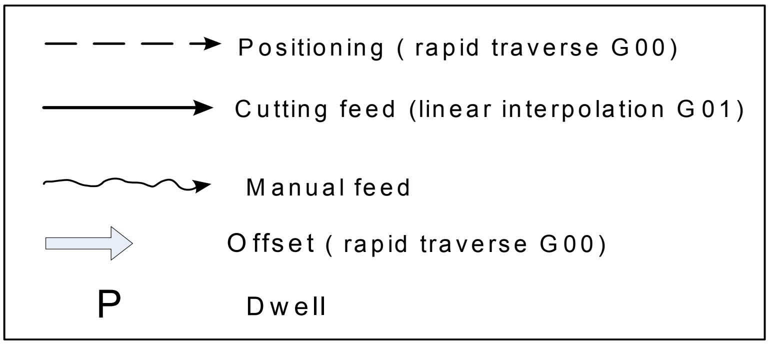


Fig. 3-4-5

Descriptions of each machining mode:

3.4.1 High-Speed Peck Drilling Cycle G73

Format : G73 X_Y_Z_R_Q_F_K_

Function:

The cycle performs high-speed peck drilling. It executes intermittent cutting feed to the bottom of a hole while removing chips from the hole. About the operation, refer to Fig. 3-4-6.

Explanation:

X_Y_: Hole position data;

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

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

Q_: Depth of cut for each cutting feed;

F_: Cutting feedrate;

K_: Number of repeats.

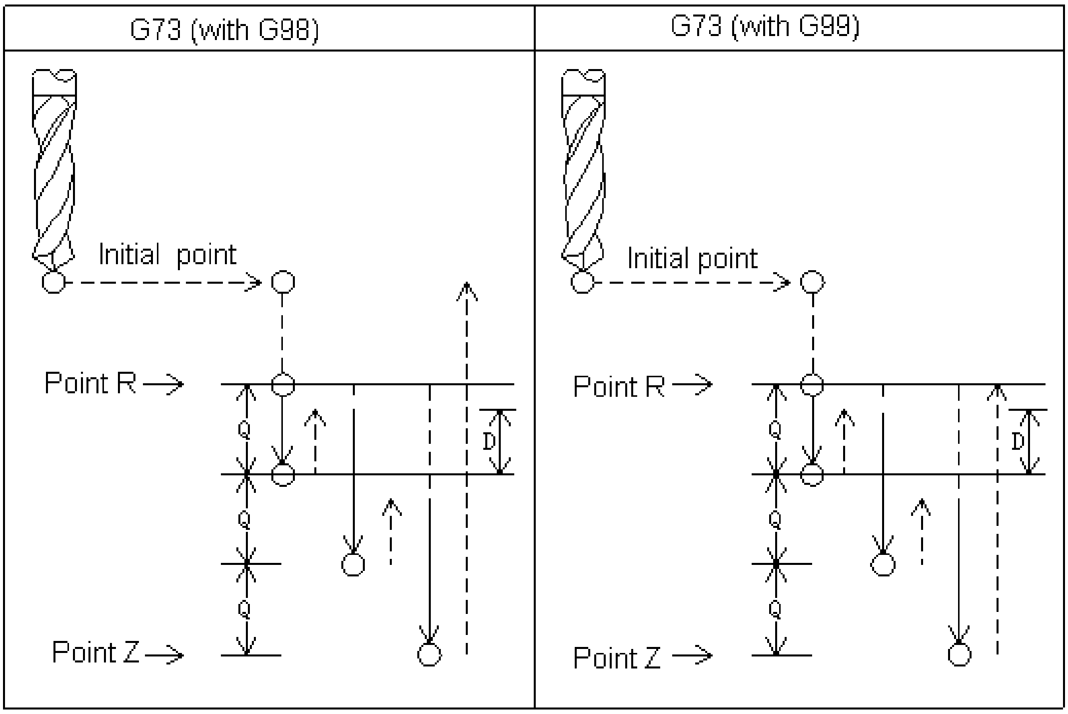


Fig. 3-4-6

The high-speed peck drilling cycle performs intermittent feeding along the Z-axis. When this cycle is used, chips can be removed from the hole easily, and a smaller value can be set for retraction. This allows drilling to be performed efficiently. Set the clearance, d, in parameter No.2010. The tool is retracted in rapid traverse.

Before specifying G73, rotate the spindle using a miscellaneous function (M code).
When a tool length offset (G43 or G44) is specified in the canned cycle, the offset is applied at the time of positioning to point R.

The canned cycle is cancelled with G80, G00, G01, G02 or G03.

In the canned cycle mode, tool offsets are ignored.

Restrictions:

1) Before the drilling axis can be changed, the canned cycle must be cancelled; otherwise, the system alarms.
2) In a block that does not contain X, Y, Z, R or commands of any other axes, drilling is not performed.
3) Specify Q in blocks that perform drilling. If they are specified in a block that does not perform drilling, they cannot be stored as modal data.

Example 1

M3 S1500；	Cause the spindle to start rotating;
G90 G99 G73 X0 Y0 Z-150. R-120.Q5 F120；	Position, drill hole 1, then return to point R;
Y-50.；	Position, drill hole 2, then return to point R;
Y-80.；	Position, drill hole 3, then return to point R;
X10.；	Position, drill hole 4, then return to point R;
Y10.；	Position, drill hole 5, then return to point R;
G98 Y75.；	Position, drill hole 6, then return to the initial level;

G80；

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

M5； Cause the spindle to stop rotating.

M30

Note:

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

3.4.2 Left-Handed Tapping Cycle G74

Format: G74 X_ Y_ Z_ R_ P_ F_ K_

Function:

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

Explanations:

X_Y_: Hole position data;

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

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

P_: Dwell time;

F_: Cutting feedrate;

K_: Number of repeats.

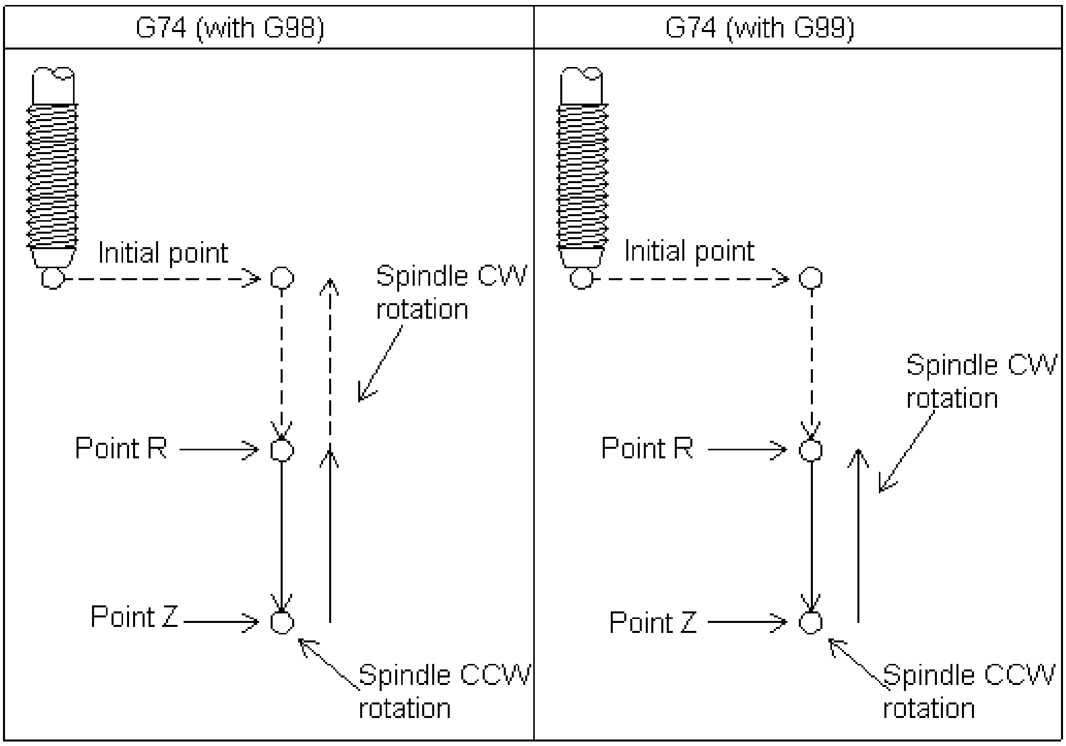


Fig. 3-4-7

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

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

Before specifying G74, use a miscellaneous function (code M04) to rotate the spindle. If the spindle CCW rotation isn't commanded, the system changes into spindle CCW rotation on point R level based on the current spindle commanded speed.

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

The canned cycle can be cancelled with G80, G00, G01, G02 and G03.

In the canned cycle mode, tool offsets are ignored.

Restrictions:

1) Before the drilling axis can be changed, the canned cycle must be cancelled; otherwise, the system alarms.
2) In a block that does not contain X, Y, Z, R or commands of any other axes, drilling is not performed.
3) Specify P in blocks that perform drilling. If they are specified in a block that does not perform drilling, they cannot be stored as modal data.

Example:

M4 S100；	Cause the spindle to start rotating;
G90 G99 G74 X30. Y-25. Z-150. R-120. P300 F150；	Position, tapping hole 1, then return to point R;
Y-55.；	Position, tapping hole 2, then return to point R;
Y-75.；	Position, tapping hole 3, then return to point R;
X100.；	Position, tapping hole 4, then return to point R;
Y-55.；	Position, tapping hole 5, then return to point R;
G98 Y-75.；	Position, tapping hole 6, then return to the initial level;
G80；	
G28 G91 X0 Y0 Z0 ；	Return to the reference position;
M5；	Cause the spindle to stop rotating.
M30；	

3.4.3 Fine Boring Cycle G76

Format : G76 X_Y_Z_Q_R_P_F_K_
Function:

The fine boring cycle bores a hole precisely. When the hole bottom has been reached, the spindle stops, and the tool is moved away from the machined surface of the workpiece and retracted.

Explanations:

X_Y_: Hole position data;
Z_: The incremental programming means the distance from point R to the bottom of the hole; the absolute programming means the absolute coordinate value of the bottom of the hole.
R_: The incremental programming means the distance from the initial level to point R level; the absolute programming means the absolute coordinate value of point R.
Q_: Shift amount at the bottom of a hole;
P_: Dwell time;

F_: Cutting feedrate;
K_: Number of repeats.

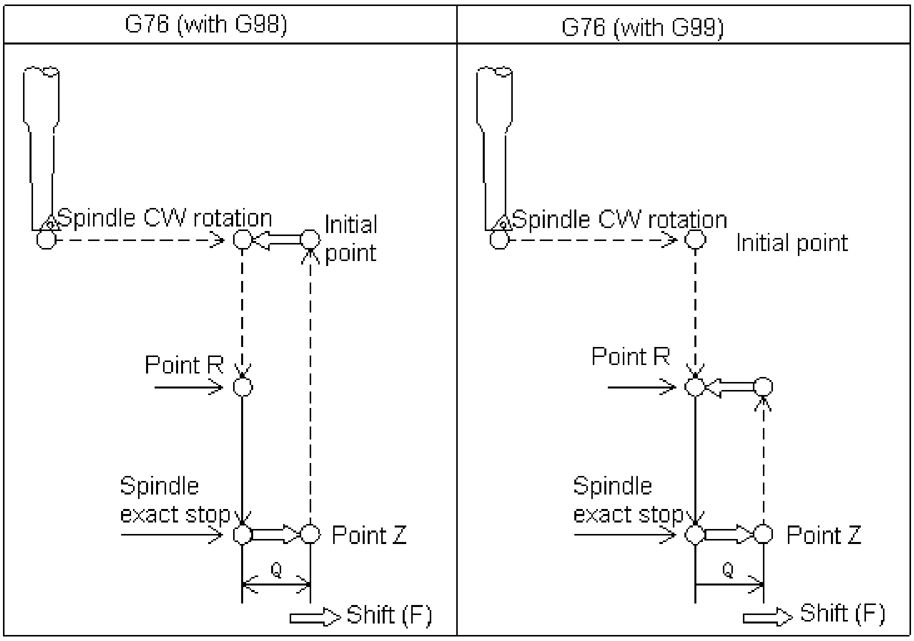


Fig. 3-4-8

When the bottom of the hole has been reached, the spindle is stopped at the fixed rotation position, and the tool is moved in the direction opposite to the tool tip and retracted. This ensures that the machined surface is not damaged and enables precise and efficient boring to be performed. The parameter Q specifies the distance of tool retraction. The offset direction is set by bit parameters 2000.4 and 2000.5（RD1, RD2）. Be sure to specify a positive value in Q. If Q is specified with a negative value, the sign is ignored. **Pay attention to that Q (shift at the bottom of a hole) is a modal value retained within canned cycles. It must be specified carefully because it is also used as the depth of cut for G73 and G83.**

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

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

The canned cycle is cancelled with G80, G00, G01, G02 or G03.

In the canned cycle mode, tool offsets are ignored.

Restrictions:

1) Before the drilling axis can be changed, the canned cycle must be cancelled; otherwise, the system alarms.

2) In a block that does not contain X, Y, Z, R or commands of any other axes, drilling is not performed.

3) Specify P and Q in blocks that perform drilling. If they are specified in a block that does not perform drilling, they cannot be stored as modal data.

Example:

M3 S500； Cause the spindle to start rotating;
G90 G99 G76 X30.Y-25 Z-150. R-120.Q5 P1000 F120.； Position, bore hole 1, stop at the bottom of the hole for 1s, orient at the bottom of the hole, then shift by 5mm, finally return to point R;
Y-55.； Position, bore hole 2, then return to point R.
Y-75.； Position, bore hole 3, then return to point R.
X100.； Position, bore hole 4, then return to point R.
Y-55.； Position, bore hole 5, then return to point R.
G98 Y-75.； Position, bore hole 6, then return to the initial level.
G80 G28 G91 X0 Y0 Z0； Return to the reference position
M5； Cause the spindle to stop rotating.

3.4.4 Canned cycle cancel G80

Format: G80

Function: Cancel the canned cycles.

Explanation:

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

Example:

M3 S100； Cause the spindle to start rotating;
G90 G99 G88 X30. Y-25. Z-150. R-120. F120； Position, bore hole 1, then return to point R.
Y-55.； Position, bore hole 2, then return to point R.
Y-75.； Position, bore hole 3, then return to point R.
X100.； Position, bore hole 4, then return to point R.
Y-55.； Position, bore hole 5, then return to point R.
G98 Y-75.； Position, bore hole 6, then return to the initial level.
G80；
G28 G91 X0 Y0 Z0 ； Return to the reference position and cancel the canned cycle;

3.4.5 Drilling Cycle, Spot Drilling (G81)

Format : G81 X_ Y_ Z_ R_ F_ K_

Function:

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

Explanation:

X_Y_: Hole position data

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

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

F_: Cutting feedrate.

K_: Number of repeats.

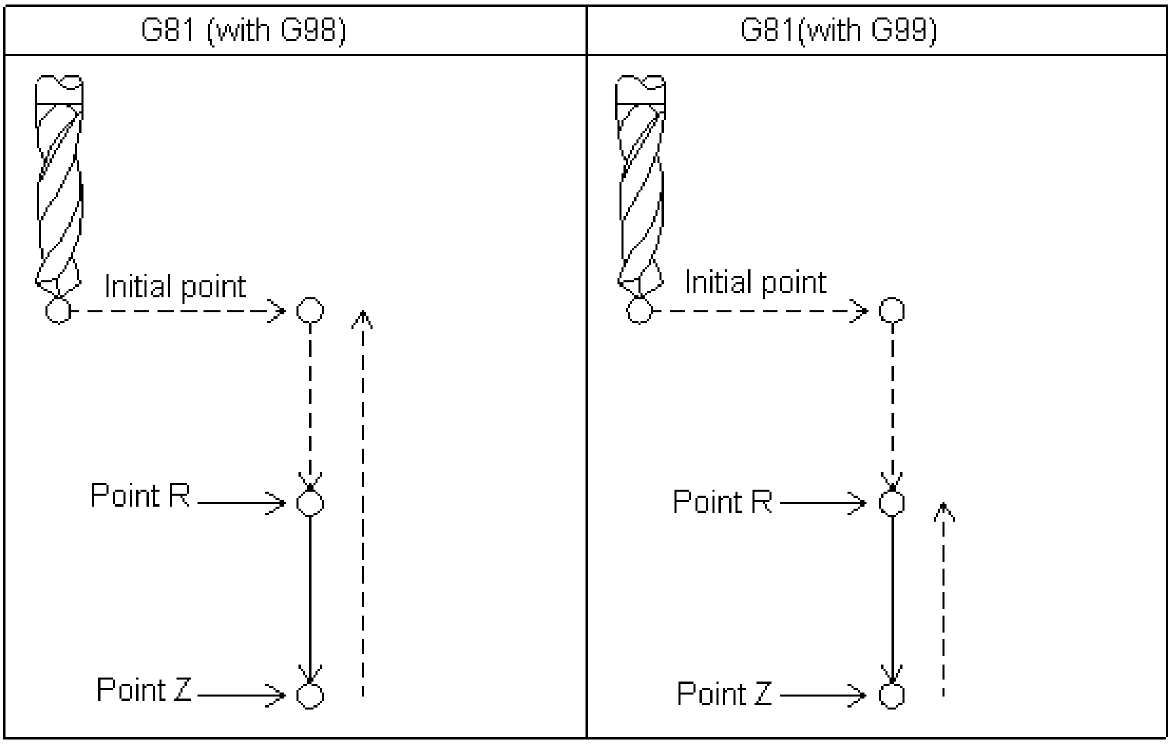


Fig. 3-4-9

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

Before specifying G81, rotate the spindle using a miscellaneous function (M code).

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

The canned cycle is cancelled with G80, G00, G01, G02 or G03.

In the canned cycle mode, tool offsets are ignored.

Restrictions:

1) Before the drilling axis can be changed, the canned cycle must be cancelled; otherwise, the system alarms.

2) In a block that does not contain X, Y, Z, R or commands of any other axes, drilling is not performed.

Example:

M3 S2000；	Cause the spindle to start rotating;
G90 G99 G81 X30. Y-25. Z-150. R-120. F120；	Position, drill hole 1, and then return to point R;
Y-55.；	Position, drill hole 2, and then return to point R;
Y-75.；	Position, drill hole 3, and then return to point R;
X100.；	Position, drill hole 4, and then return to point R;
Y-55.；	Position, drill hole 5, and then return to point R;
G98 Y-75.；	Position, drill hole 6, and then return to the initial level;
G80；	

Part Ⅰ Programming

G28 G91 X0 Y0 Z0 ； Return to the reference position
M5； Cause the spindle to start rotating;
M30；

3.4.6 Drilling Cycle, Counter Boring Cycle G82

Format: G82 X_ Y_ Z_ R_ P_ F_ K_；
Function:

The cycle is used for normal drilling. Cutting feed is performed to the bottom of the hole. At the bottom, a dwell is performed, and then the tool is retracted in rapid traverse. The cycle can improve the precision of the hole depth.

Explanations:

X_Y_: Hole position data.

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

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

F_: Cutting feedrate

P_: Dwell time

K_: Number of repeats

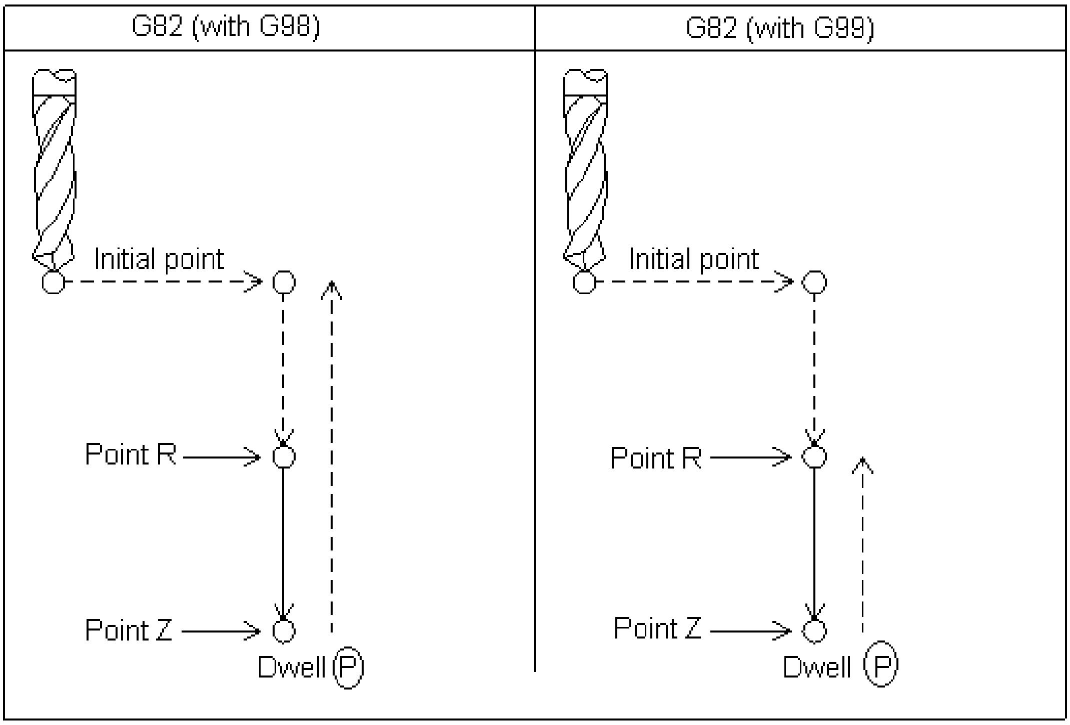


Fig. 3-4-10

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

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

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

The canned cycle is cancelled with G80, G00, G01, G02 or G03.

In the canned cycle mode, tool offsets are ignored.

Restrictions:

1) Before the drilling axis can be changed, the canned cycle must be cancelled; otherwise, the system alarms.

2) In a block that does not contain X, Y, Z, R or commands of any other axes, drilling is not performed.

3) Specify P in blocks that perform drilling. If they are specified in a block that does not perform drilling, they cannot be stored as modal data.

Example:

M3 S2000； Cause the spindle to start rotating;
G90 G99 G82 X30. Y-25. Z-150. R-120. P1000 F120； Position, drill hole 1, and dwell for 1s at the hole bottom, then return to point R;
Y-55； Position, drill hole 2, and dwell for 1s at the hole bottom, then return to point R;
Y-75； Position, drill hole 3, and dwell for 1s at the hole bottom, then return to point R;
X100.； Position, drill hole 4, and dwell for 1s at the hole bottom, then return to point R;
Y-55； Position, drill hole 5, and dwell for 1s at the hole bottom, then return to point R;
G98 Y-75； Position, drill hole 6, and dwell for 1s at the hole bottom, then return to initial level;
G80； Canned cycle cancel
G28 G91 X0 Y0 Z0 ； Return to the reference position;
M5； Cause the spindle to stop rotating.
M30；

3.4.7 Peck Drilling Cycle (G83)

Format: G83 X_ Y_ Z_ R_ Q_ F_ K_

Function:

This cycle performs peck drilling. It performs intermittent cutting feed to the bottom of a hole while removing shavings from the hole.

Explanations:

X_Y_: Hole position data.

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

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

Q_: Depth of cut for each cutting feed.

F_: Cutting feedrate.

K_: Number of repeats.

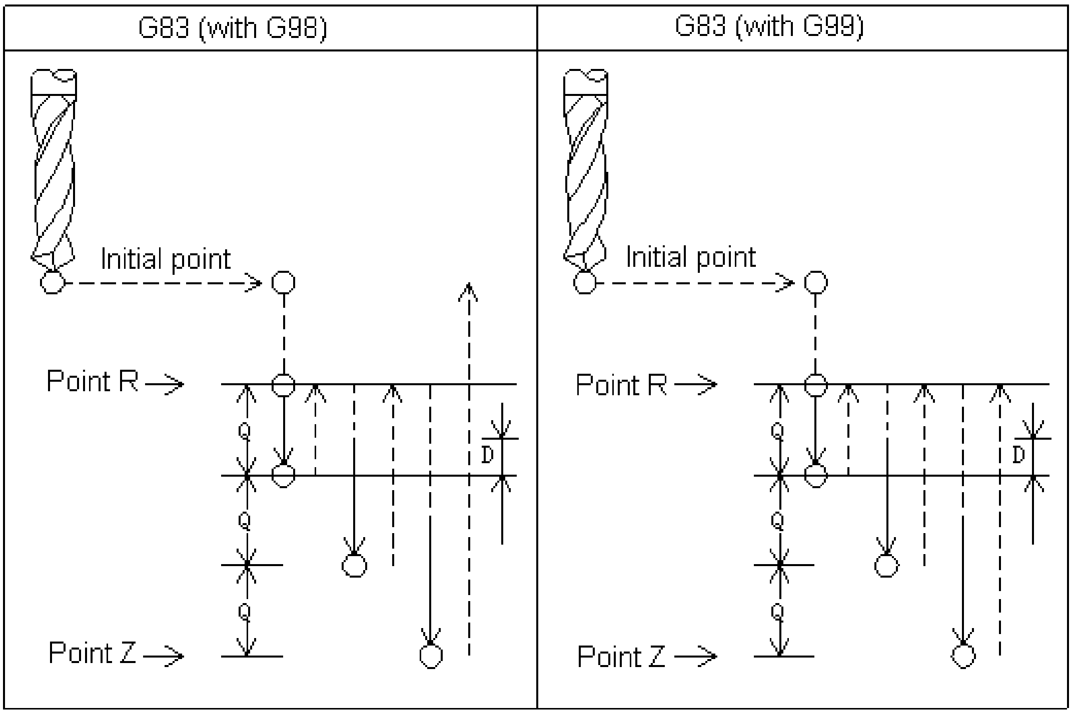


Fig. 3-4-11

Q represents the depth of cut for each cutting feed. It must always be specified as an incremental value. When the rapid traverse is performed to point R at the first time, cut for Q distance, and retraction is executed to point R at the rapid traverse rate. In the second and subsequent cutting feeds, rapid traverse is performed up to a d point just before where the last drilling ended, and cutting feed d+Q is performed again and retraction is executed to point R at the rapid traverse rate. Such cycle is executed repeatedly until the depth Z value is reached. d is set in parameter (**No.P2011)**. The operation is shown as figure **3-24**.

Be sure to specify a positive value in Q. Negative sign is ignored; the system operates as the positive one.

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

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

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

The canned cycle is cancelled with G80, G00, G01, G02 or G03.

In the canned cycle mode, tool offsets are ignored.

Restrictions:

1) Before the drilling axis can be changed, the canned cycle must be cancelled; otherwise, the system alarms.

2) In a block NOT contain X, Y, Z, R or commands of any other axes, drilling is not performed.

3) Specify Q in blocks that perform drilling. If they are specified in a block that does not perform drilling, they cannot be stored as modal data.

Example

M3 S2000; Cause the spindle to start rotating;
G90 G99 G83 X30. Y-25. Z-150. R-120. Q15 F120; Position, drill hole 1, and then return to point R;
Y-55; Position, drill hole 2, and then return to point R;
Y-75; Position, drill hole 3, and then return to point R;
X100; Position, drill hole 4, and then return to point R;
Y-55; Position, drill hole 5, and then return to point R;
G98 Y-75; Position, drill hole 6, and then return to initial level;
G80;
G28 G91 X0 Y0 Z0 ; Return to the reference position;
M5; Cause the spindle to stop rotating,
M30;

3.4.8 Right-Handed Tapping Cycle G84

Format : G84 X_ Y_ Z_ R_ P_ F_ K_
Function:

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

Explanations:

X_Y_: Hole position data;

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

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

P_: Dwell time;

F_: Cutting feedrate;

K_: Number of repeats.

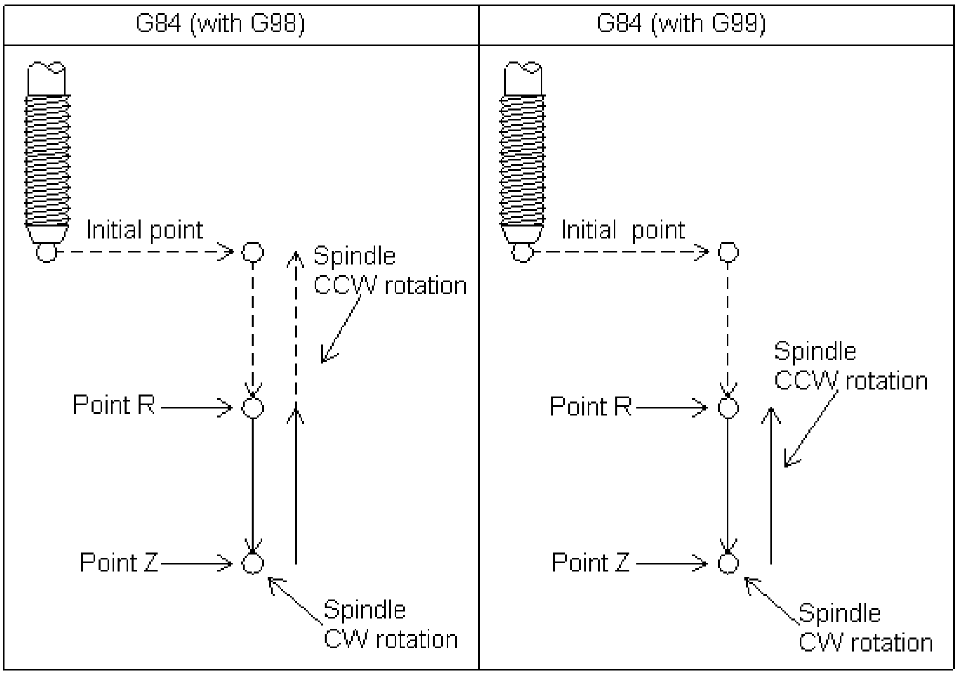


Fig. 3-4-12

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

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

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

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

The canned cycle can be cancelled by G80, G00, G01, G02 or G03.

In the canned cycle mode, tool offsets are ignored.

Restrictions:

1) Before the drilling axis can be changed, the canned cycle must be cancelled; otherwise, the system alarms.

2) In a block that does not contain X, Y, Z, R or commands of any other axes, drilling is not performed.

3) Specify P in blocks that perform drilling. If they are specified in a block that does not perform drilling, they cannot be stored as modal data.

Example:

M3 S100；	Cause the spindle to start rotating;
G90 G99 G84 X30. Y-25. Z-150. R-120 P3000F120；	Position, tap hole 1, and then return to point R;
Y-55.；	Position, tap hole 2, and then return to point R;
Y-75.；	Position, tap hole 3, and then return to point R;
X100.；	Position, tap hole 4, and then return to point R;
Y-55.；	Position, tap hole 5, and then return to point R;
G98 Y-75.；	Position, tap hole 6, and then return to initial level;
G80；	
G28 G91 X0 Y0 Z0 ；	Return to the reference position;
M5；	Cause the spindle to stop rotating;
M30；	

3.4.9 Boring Cycle G85

Format : G85 X_ Y_ Z_ R_ F_ K_

Function: The cycle is used to bore a hole.

Explanations:

X_Y_: Hole position data;

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

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

F_: Cutting feedrate;

K_: Number of repeats.

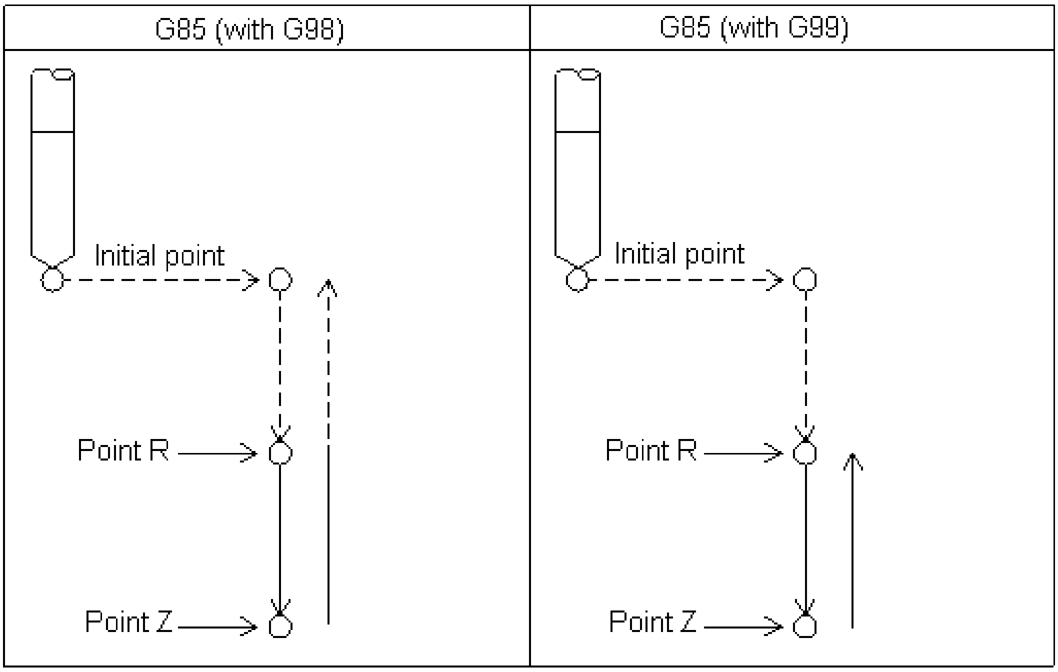


Fig. 3-4-12

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

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

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

The canned cycle can be cancelled by G80, G00, G01, G02 or G03.

In the canned cycle mode, tool offsets are ignored.

Restrictions:

1) Before the drilling axis can be changed, the canned cycle must be cancelled; otherwise, the system alarms.

2) In a block that does not contain X, Y, Z, R or commands of any other axes, drilling is not performed.

Example:

M3 S100；	Cause the spindle to start rotating;
G90 G99 G85 X30. Y-25. Z-150. R-120. F120；	Position, bore hole 1, then return to point R;
Y-55.；	Position, bore hole 2, then return to point R;
Y-75.；	Position, bore hole 3, then return to point R;
X100.；	Position, bore hole 4, then return to point R;
Y-55.；	Position, bore hole 5, then return to point R;
G98 Y-75.；	Position, bore hole 6, then return to initial level;
G80；	
G28 G91 X0 Y0 Z0 ；	Return to the reference position;
M5；	Cause the spindle to stop rotating;
M30；	

3.4.10 Boring Cycle G86

Format : G86 X_ Y_ Z_ R_ F_ K_;

Function: This cycle is used to bore a hole.

Explanations:

X_Y_: Hole position data.

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

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

F_: Cutting feedrate.

K_: Number of repeats.

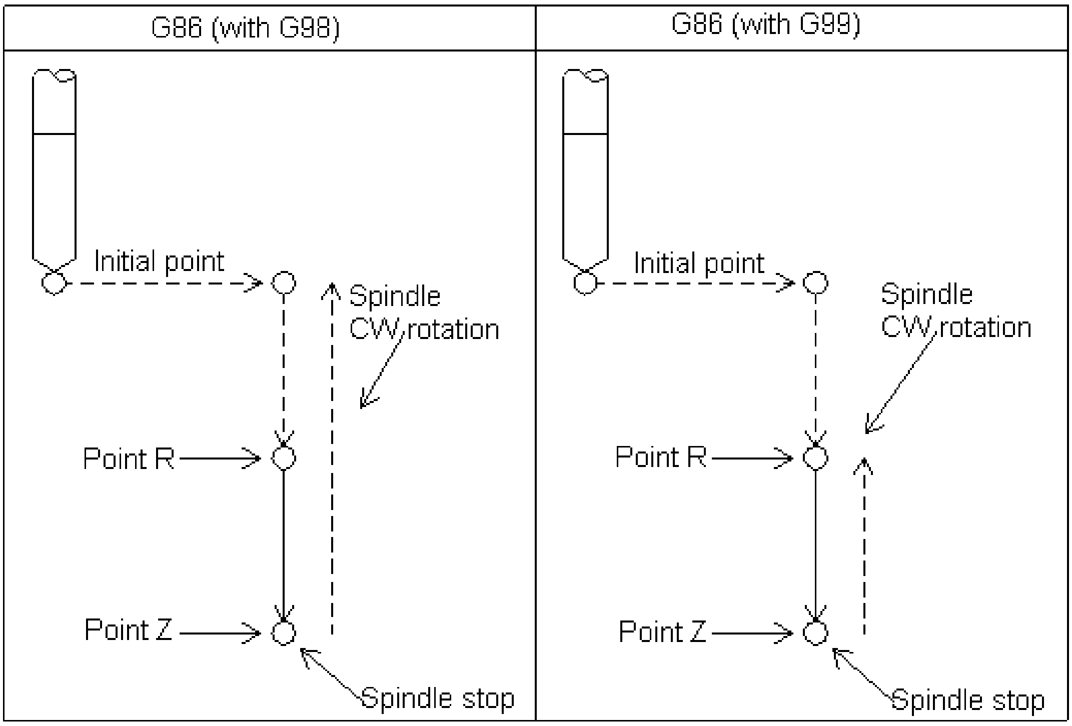


Fig. 3-4-13

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

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

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

The canned cycle can be cancelled by G80, G00, G01, G02 or G03.

In the canned cycle mode, tool offsets are ignored.

Restrictions:

1) Before the drilling axis can be changed, the canned cycle must be cancelled; otherwise, the system alarms.

2) In a block that does not contain X, Y, Z, R or commands of any other axes, drilling is not performed.

Example:

M3 S2000;	Cause the spindle to start rotating;
G90 G99 G86 X30. Y-25. Z-150. R-120. F120;	Position, bore hole 1, then return to point R;
Y-55.;	Position, bore hole 2, then return to point R;
Y-75.;	Position, bore hole 3, then return to point R;
X100.;	Position, bore hole 4, then return to point R;
Y-55.;	Position, bore hole 5, then return to point R;
G98 Y-75.;	Position, bore hole 6, then return to initial level;
G80;	
G28 G91 X0 Y0 Z0 ;	Return to the reference position;
M5;	Cause the spindle to stop rotating;

3.4.11 Boring Cycle, Back Boring Cycle (G87)

Format : G87 X_Y_Z_R_Q_ P_F_ K_
Function: The cycle performs accurate boring.

Explanations:

X_Y_: Hole position data.

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

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

Q_: Offset value at the hole bottom.

P_: Dwell time;

F_: Cutting feedrate.

K_: Number of repeats.

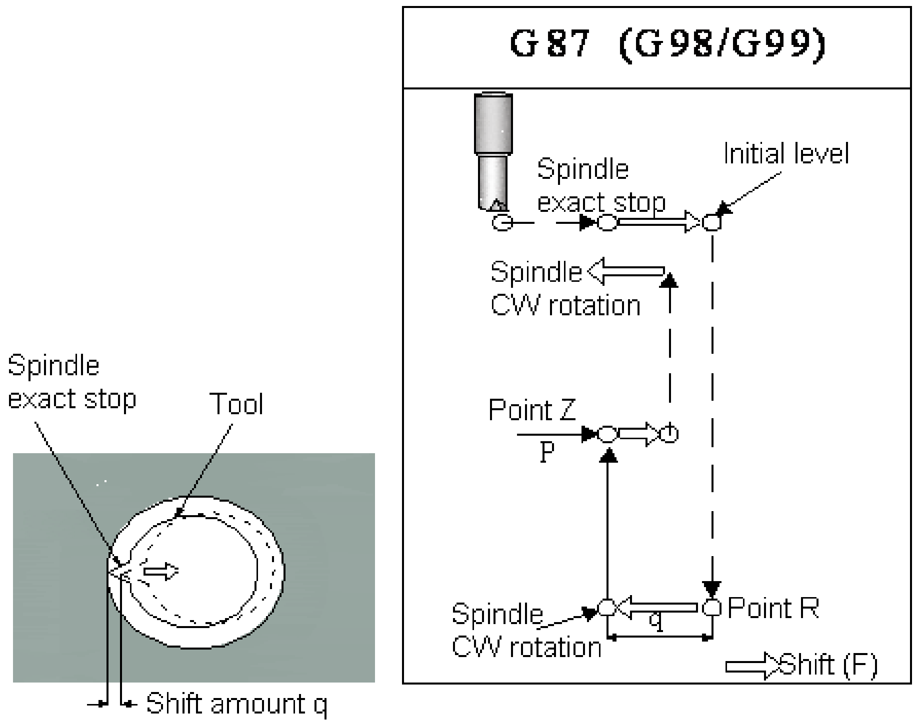


Fig. 3-4-13

After positioning along the X- and Y- axes, the spindle is stopped at the fixed rotation position. The tool is moved in the direction opposite to the tool tip; positioning (rapid traverse) is performed to the bottom of the hole (point R). The tool is then shifted in the direction of the tool tip and the spindle is rotated clockwise. Boring is performed in the positive direction along the Z-axis until point Z is reached. At point Z, the spindle is stopped at the fixed rotation position again, the tool is shifted in the direction opposite to the tool tip, and then the tool is returned to the initial level. The tool is then shifted in the direction of the tool tip to return the initial point and the spindle is rotated clockwise to proceed to the next block operation. X and Y axes offset values and directions are exactly same as those in G76, and the setting direction is same as that of G76 and G87. **Q (shift at the bottom of a hole) is a modal value retained in canned cycles. It must be specified carefully because it is also used as the depth of cut for G73 and G83.**

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

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

The canned cycle can be cancelled by G80, G00, G01, G02 or G03.

In the canned cycle mode, tool offsets are ignored.

Restrictions:

1) Before the drilling axis can be changed, the canned cycle must be cancelled; otherwise, the system alarms.

2) In a block that does not contain X, Y, Z, R or commands of any other axes, drilling is not performed.

3) Specify P and Q in blocks that perform drilling. If they are specified in a block that does not perform drilling, they cannot be stored as modal data.

Note:

During the back boring cycle programming, please remember the specification of Z and R values. Normally, Z position must be above R in the situation; otherwise, the unexpected consequence may occur.

Example:

M3 S500； Cause the spindle to start rotating;

G90 G99 G87 X30. Y-25. Z-120. R-150. Q5. P1000 F120； Position, bore hole 1; Orient at the initial level, then shift by 5mm. Stop at point Z for 1s;

Y-55.; Position, bore hole 2, then return to point R;

Y-75.; Position, bore hole 3, then return to point R;

X100.; Position, bore hole 4, then return to point R;

Y-55.; Position, bore hole 5, then return to point R;

G98 Y-75.; Position, bore hole 6, then return to the initial level;

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

M5; Cause the spindle to stop rotating.

3.4.12 Boring Cycle (G88)

Format : G88 X_Y_Z_R_ P_ F_ K_
Function: This cycle is used to bore a hole.
Explanations:

X_Y_: Hole position data

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

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

P_: Dwell time

F_: Cutting feedrate

K_: Number of repeats.

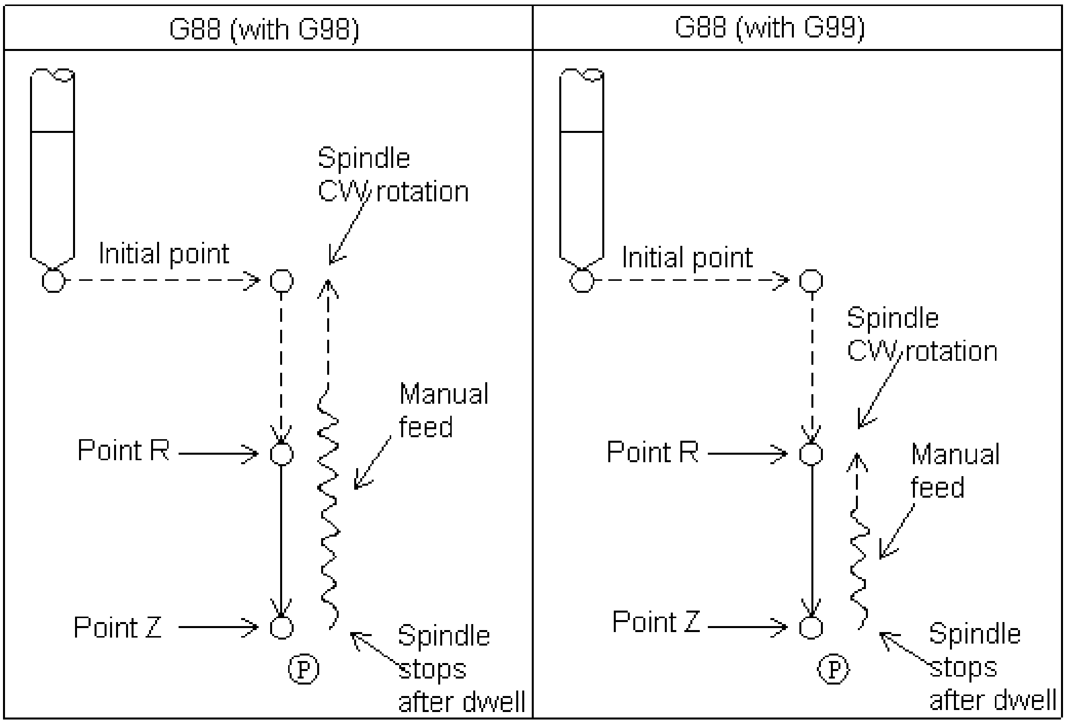


Fig. 3-4-14

After positioning along the X- and Y- axes, rapid traverses is performed to point R, Boring is performed from point R to point Z. When boring is completed, a dwell is performed, then the spindle is started to decelerate till stop, and the system enters the feed hold state and the indicator lamp is on. The tool is manually retracted from the bottom of the hole to point R (**For safety, it's better to move the tool outside**), and then cycle start is performed, after returning, the next block is executed. Or, when cutting feed is performed to the bottom of the hole, cycle start is executed directly without manual return, the tool automatically returns to the initial level.

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

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

The canned cycle can be cancelled by G80, G00, G01, G02 or G03.

In the canned cycle mode, tool offsets are ignored.

Restrictions:

1) Before the drilling axis can be changed, the canned cycle must be cancelled; otherwise, the system alarms.

2) In a block that does not contain X, Y, Z, R or commands of any other axes, drilling is not performed.

3) Specify P in blocks that perform drilling. If they are specified in a block that does not perform drilling, they cannot be stored as modal data.

4) The return operation is executed in JOG mode, K value is invalid.

Note:

During G88 hole cycle cutting, if it is not in JOG mode (such as in MDI mode), cycle start doesn't function. Normally, G88 should be executed in sequence strictly; otherwise, the tool may get damaged.

Example:

M3 S2000; Cause the spindle to start rotating;
G90 G99 G88 X30. Y-25. Z-150. R-120. P1000 F120;
Position, bore hole 1, and then return to point R;
Y-55.; Position, bore hole 2, and then return to point R;
Y-75.; Position, bore hole 3, and then return to point R;
X100.; Position, bore hole 4, and then return to point R;
Y-55.; Position, bore hole 5, and then return to point R;
G98 Y-75.; Position, bore hole 6, and then return to initial level;
G80 G28 G91 X0 Y0 Z0 ; Return to the reference position;
M5; Cause the spindle to stop rotating.

3.4.13 Boring Cycle (G89)

Format : G89 X_ Y_ Z_ R_ P_ F_ K_

Function: This cycle is used to bore a hole.

Explanations:

X_Y_: Hole position data.
Z_: The incremental programming means the distance from point R to the hole bottom; the absolute programming means the absolute coordinate value of the hole bottom.
R_: The incremental programming means the distance from the initial level to point R; the absolute programming means the absolute coordinate value of point R.
P_: Dwell time
F_: Cutting feedrate.
K_: Number of repeats.

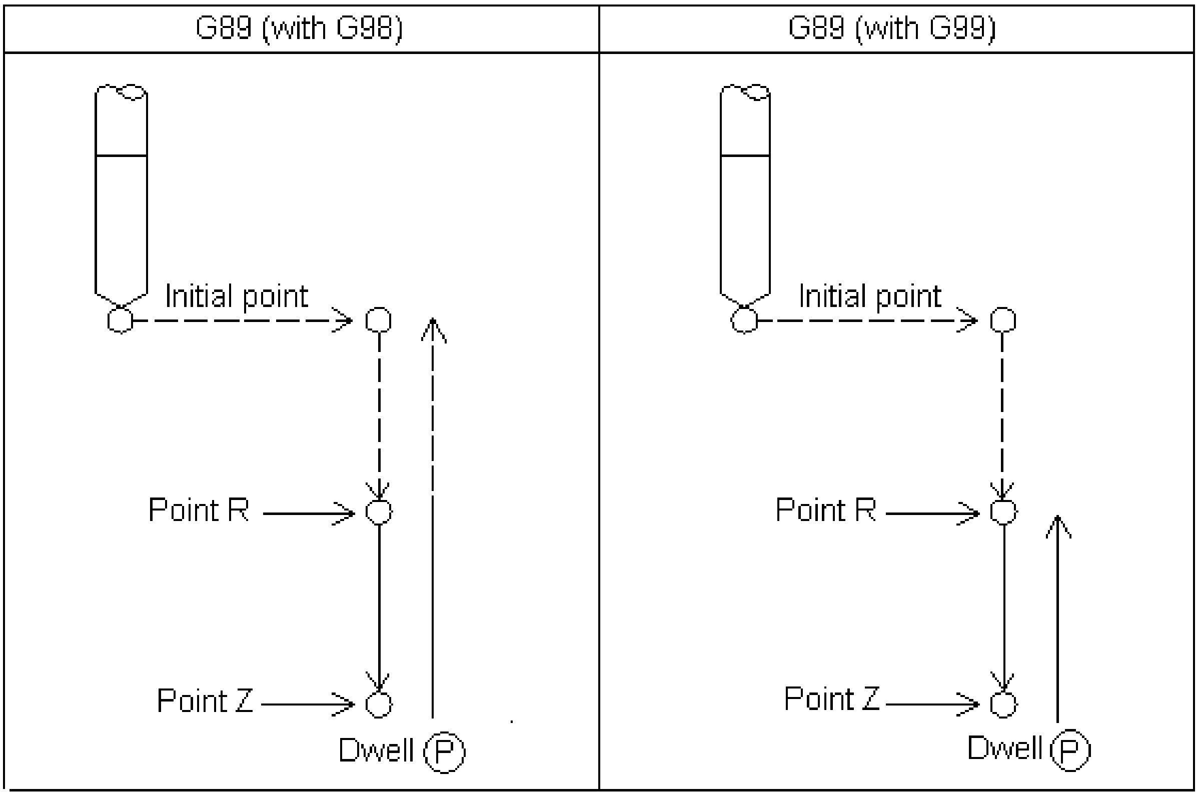


Fig. 3-4-15

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

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

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

The canned cycle can be cancelled by G80, G00, G01, G02 or G03.

In the canned cycle mode, tool offsets are ignored.

Restrictions:

1) Before the drilling axis can be changed, the canned cycle must be canceled; otherwise, the alarm is issued.

2) In a block that does not contain X, Y, Z, R or any other axes command, drilling is not performed.

3) Specify P in blocks that perform drilling. If it is specified in a block that does not perform drilling, it cannot be stored as modal data.

Example:

M3 S100；	Cause the spindle to start rotating;
G90 G99 G89 X30. Y-25. Z-150. R-120. P1000 F120；	Position, bore hole 1, return to point R, and then stop at the hole bottom for 1s;
Y-55.；	Position, bore hole 2, and return to point R;
Y-75.；	Position, bore hole 3, and return to point R;
X100.；	Position, bore hole 4, and return to point R;

Y-55.;	Position, bore hole 5, and return to point R;
G98 Y-75.;	Position, bore hole 6, and return to initial level;
G80;	
G28 G91 X0 Y0 Z0 ;	Return to the reference position;
M5;	Cause the spindle to stop rotating.
M30;	

3.4.14 Left-handed Rigid Tapping Cycle（G74）

Format: G74 X_Y_Z_R_P_F_K_

Function:

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

Explanations:

X_Y_: Hole position data.

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

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

P_: Dwell time

F_: Cutting feedrate.

K_: Number of repeats.

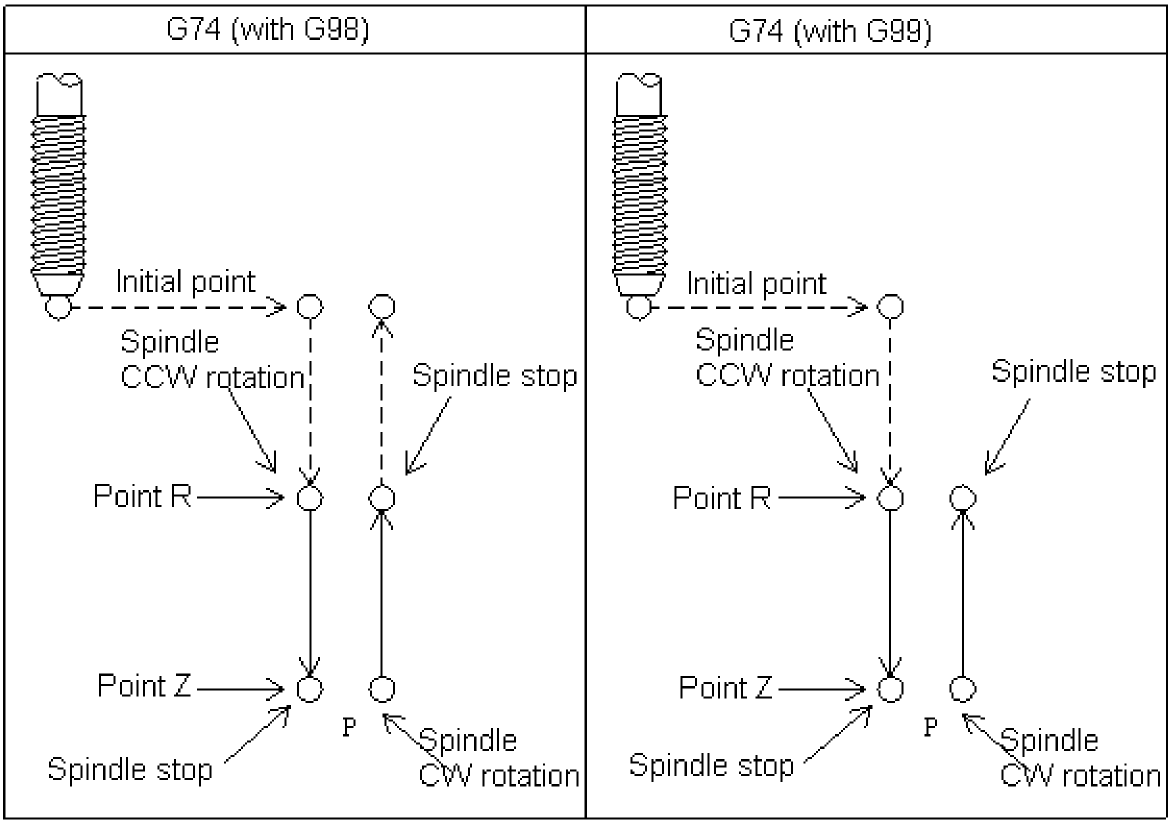


Fig. 3-4-16

After positioning along the X- and Y- axes, Z axis moves to point R in rapid traverse. Spindle starts CCW rotation with G74, and tapping is executed from point R to Z. When tapping is completed, the spindle is stopped and a dwell is performed. The spindle is then rotated in the reverse direction, the tool is retracted to point R, and then the spindle is stopped. Rapid traverse to the initial level is then performed. While tapping is being performed, the feedrate override and spindle override are assumed to be 100%.

Rigid mode:

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

(1) Specify M29 S***** before a tapping command;

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

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

The formula of the thread lead: Feedrate/spindle speed

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

The canned cycle can be cancelled by G80, G00, G01, G02 or G03.

In the canned cycle mode, tool offsets are ignored.

Restrictions:

1) Before the drilling axis can be changed, the canned cycle must be canceled; otherwise, the alarm is issued.

2) In a block that does not contain X, Y, Z, R or any other axes command, drilling is not performed.

3) Specify P in blocks that perform drilling. If it is specified in a block that does not perform drilling, it cannot be stored as modal data.

F: If the specified F value exceeds the upper limit value of the cutting feedrate, the system operates as the upper limit value of the cutting feedrate.

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

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

Example

Spindle speed 1000rpm

Thread lead 1.0mm

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

G00 X120 Y100； Positioning

M29 S1000 Specifying the rigid mode

G74 Z-100 R-20 F1000； Rigid tapping

3.4.15 Right-handed Rigid Tapping Cycle (G84)

Format : G84 X_Y_Z_R_P_F_K_

Function:

When a servo motor is controlled by the spindle motor in rigid tapping mode, tapping cycles can be sped up in high precision.

Explanations:

X_Y_: Hole position data.

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

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

P_: Dwell time

F_: Cutting feedrate.

K_: Number of repeats.

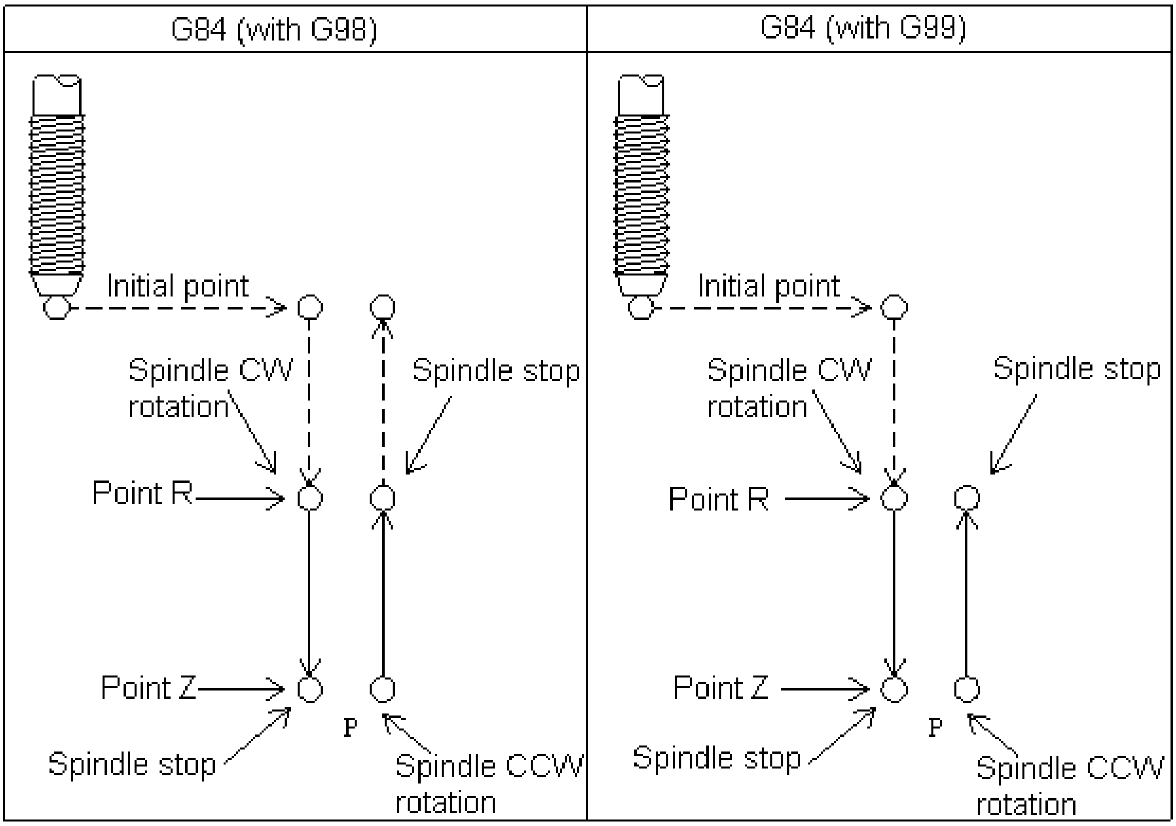


Fig. 3-4-17

After positioning along the X- and Y- axes, Z axis moves to point R in rapid traverse. Spindle starts CCW rotation with G84, and tapping is executed from point R to Z. When tapping is completed, the spindle is stopped and a dwell is performed. The spindle is then rotated in the reverse direction, the tool is retracted to point R, and then the spindle is stopped. Rapid traverse to the initial level is then performed.

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

Rigid mode:

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

(1) Specify M29 S***** before specifying a tapping command;

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

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

In the feeding/min mode, the formula of the thread lead: Feedrate/spindle speed.

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

The canned cycle can be cancelled by G80, G00, G01, G02 or G03.

In the canned cycle mode, tool offsets are ignored.

Restrictions:

1) Before the drilling axis can be changed, the canned cycle must be canceled; otherwise, the alarm is issued.

2) In a block that does not contain X, Y, Z, R or any other axes command, drilling is not performed.

3) Specify P in blocks that perform drilling. If it is specified in a block that does not perform drilling, it cannot be stored as modal data.

F: If the specified F value exceeds the upper limit value of the cutting feedrate, the system operates as the upper limit value of the cutting feedrate.

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

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

Example:

Spindle speed 1000r/min

Thread lead 1.0mm

Then, Z axis feedrate = 1000*1=1000mm/min

G00 X120 Y100； Positioning

M29 S1000 Specifying the rigid mode

G84 Z-100 R-20 F1000； Rigid tapping

3.4.16 Peck rigid tapping cycle (G84 or G74)

Format : G84/G74 X_Y_Z_R_P_Q_F_K_

Function: In this cycle, cutting is performed several times until the bottom of the hole is reached, and two tapping cycles are selected using the parameter PCP2000#7.

Explanations:

X_Y_: Hole position data.

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

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

P_: Dwell time

Q_: Cutting depth of cutting feed each time;

F_: Cutting feedrate.

K_: Number of repeats.

High-speed peck tapping cycle (parameter PCP2000#7=0)

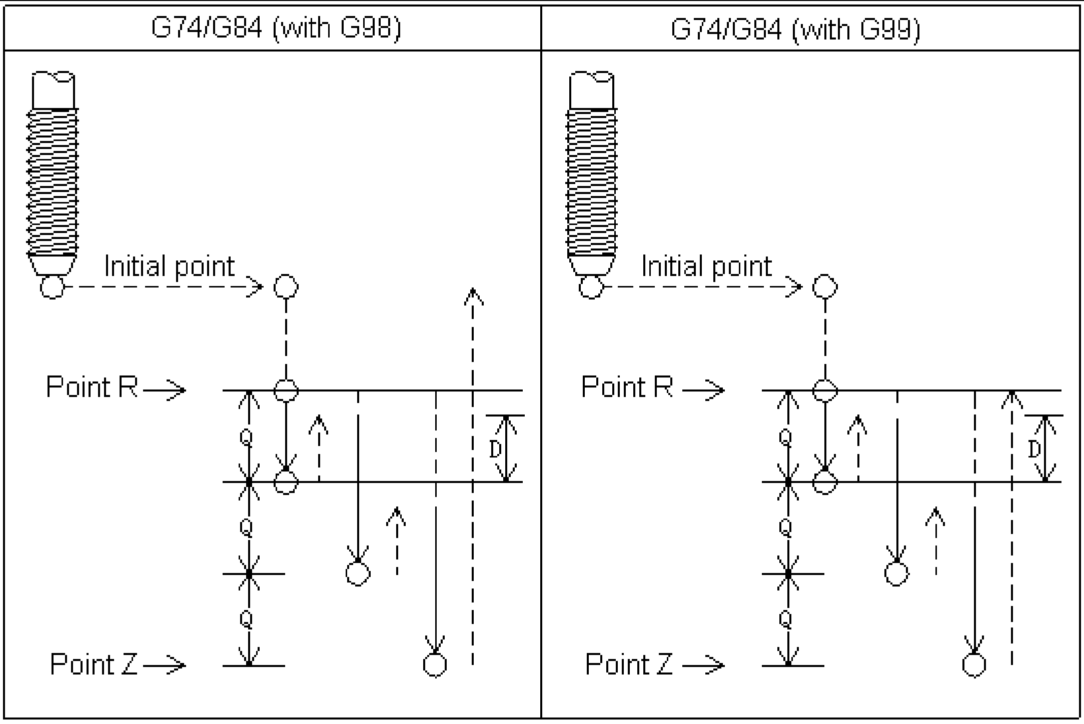


Fig. 3-4-18

After positioning along the X- and Y- axes, Z axis moves to point R in rapid traverse. From point R, cutting is performed with depth Q, then the tool is retracted by distance d, when point Z--the hole bottom has been reached, the spindle is stopped, then rotated in the reverse direction for retraction.

Peck tapping cycle (parameter PCP2000#7 = 1 Non-high speed tapping)

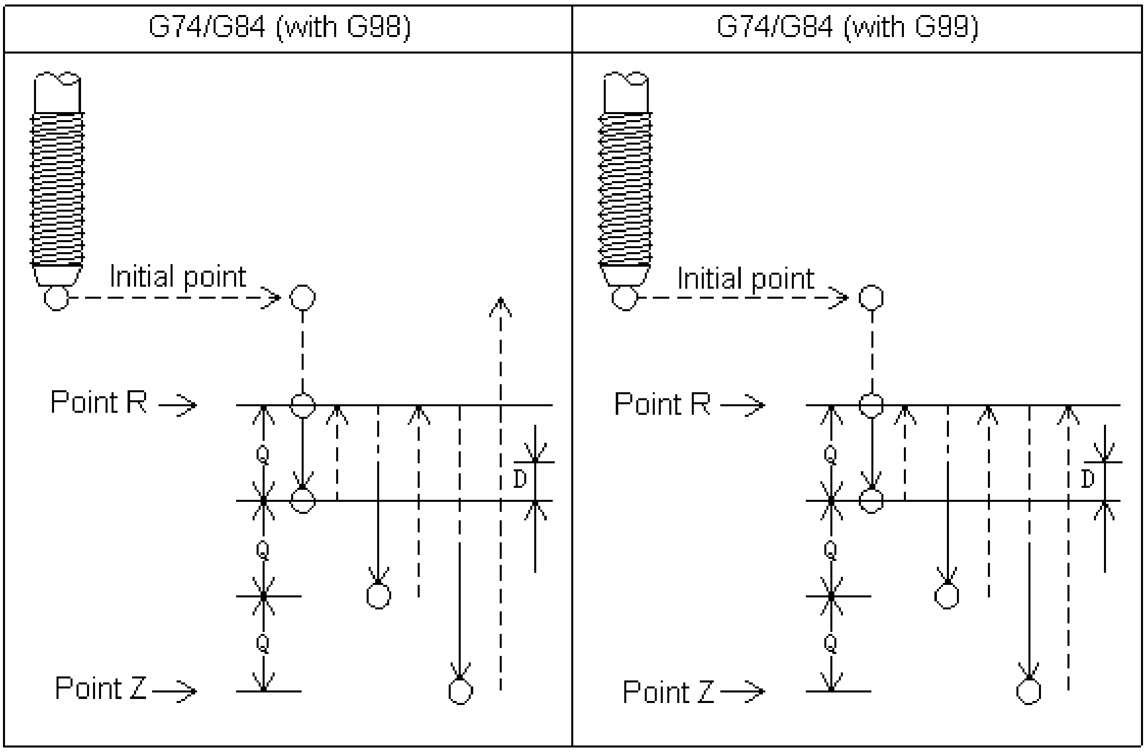


Fig. 3-4-19

After positioning along the X- and Y- axes, Z axis moves to point R in rapid traverse. From point R, cutting is performed with depth Q, then the tool is retracted to point R. And from point R to a position distance d from the end point of the last cutting, it is where cutting is restarted. when point Z--the hole bottom has been reached, the spindle is stopped, then rotated in the reverse direction for retraction.

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

The canned cycle can be cancelled by G80, G00, G01, G02 or G03.

In the canned cycle mode, tool offsets are ignored.

Restrictions:

1) Before the drilling axis can be changed, the canned cycle must be canceled; otherwise, the alarm is issued.

2) In a block that does not contain X, Y, Z, R or any other axes command, drilling is not performed.

3) Specify P and Q in blocks that perform drilling. If they are specified in a block that does not perform drilling, they cannot be stored as modal data.

F: If the specified F value exceeds the upper limit value of the cutting feedrate, the system operates as the upper limit value of the cutting feedrate.

S: The system alarms if the speed exceeds the maximum speed of the specified gear. The gear speed is set by the data parameters No: 2140~2142.

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

Example:

Explain the canned cycle usage through the tool length compensation.

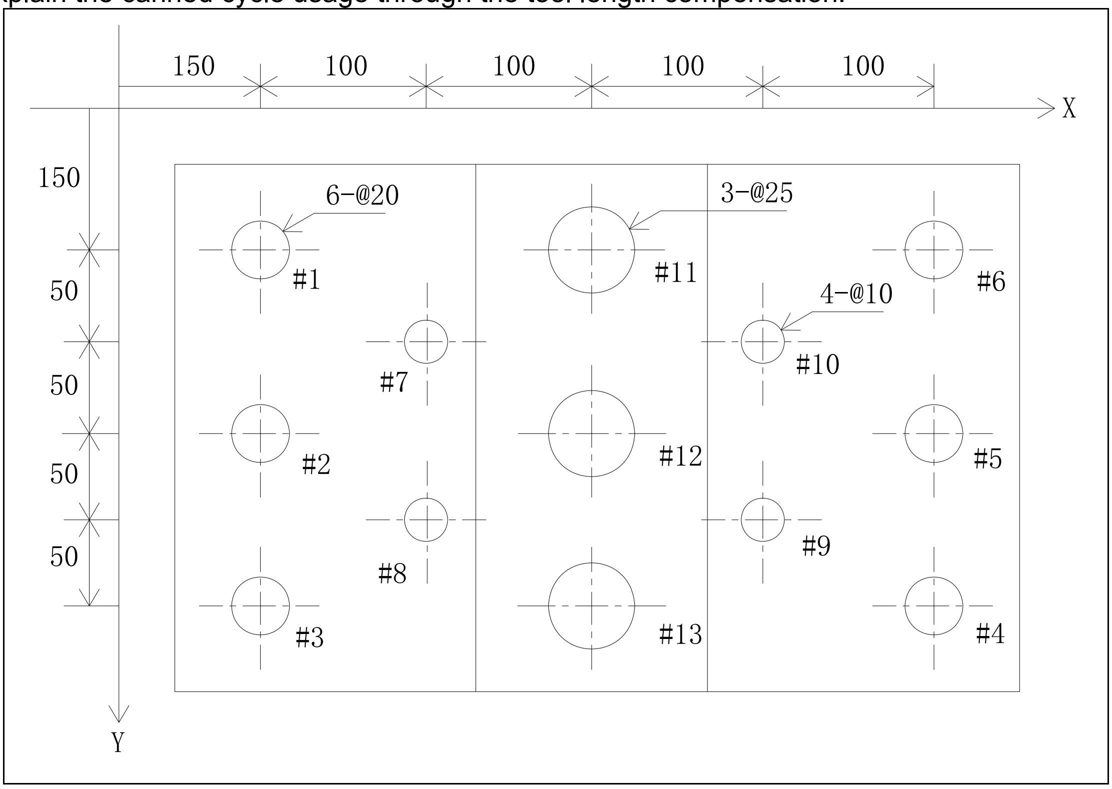


Fig. 3-4-20

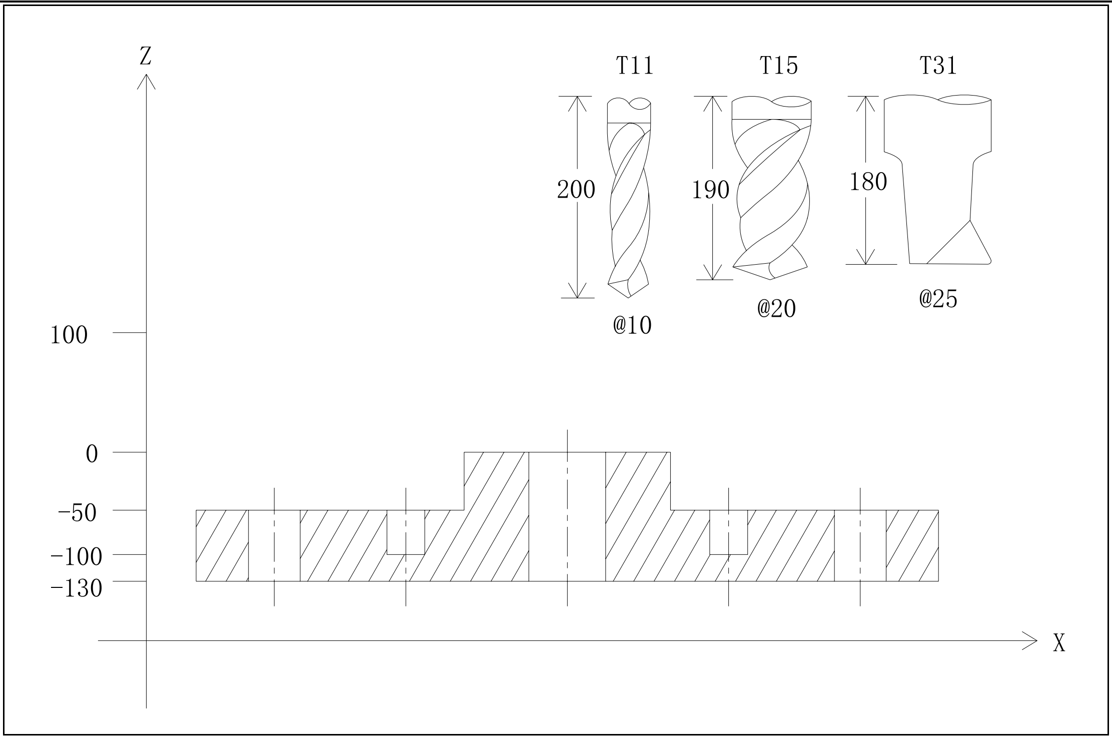


Fig. 3-4-21

The corresponding workpiece coordinate system is set in G54, and the corresponding tool length is set in the length offset. It is assumed the material is aluminum. The machining program is as below (Note: @ is represented as the diameter):

N001 G90 G54 G00 G17 G40 G80 G49 ；	Program initialization
N002 M06 T15 ；	Tool change.
N003 G43 H15 Z100 ；	H15 tool length compensation valid
N004 S5500 M3 ；	The spindle is started.
N005 G98 G83 X150 Y-150 Z-140 R-45 Q8 F300 ；	Machine #1 hole after positioning.
N006 G91 Y-100 K2 ；	Machine #2 and #3 holes after positioning, and return to initial level.
N007 G90 X550 ；	Machine #4 hole after positioning, and return to initial level.
N008 G91 Y100 K2 ；	Machine #5 and #6 holes after positioning, and return to initial level.
N011 G90 G49 Z100 M5 ；	The tool length compensation is cancelled, the spindle stops.
N012 M06 T11 ；	Tool change.
N013 G43 H11 Z100 ；	H11 tool length compensation becomes valid.
N014 S800 M3 ；	The spindle is started.
N015 G98 G83 X250 Y-200 Z-140 R-45 Q5 F150 ；	Machine #7 hole after positioning, and return to initial level.
N016 G91 Y-100 ；	Machine #8 hole after positioning, and return to initial level.
N017X200 ；	Machine #9 hole after positioning, and return to initial level.
N018 Y100 ；	Machine #10 hole after positioning, and return to initial level.
N017 G90 X350 Y-150 ；	Rough #11 hole after positioning, and return to initial level.

N018 G91 Y-100 K2 ；	Rough #12 and #13 holes after positioning, and return to initial level.
N011 G90 G49 Z100 M5 ；	Cancel the tool length compensation, the spindle stops.
N020 M06 T31 ；	Cancel the tool length compensation, tool change.
N021 G43 H31 Z100 ；	H31 tool length compensation valid.
N022 S600 M3 ；	The spindle is started.
N023 G81 G99 X350 Y-150 Z-140 R5 F100 ；	Machine #11 hole after positioning, and return to point R level.
N024 G91 Y-100 K2 ；	Machine #12 and #13 holes after positioning, and return to point R level.
N011 G49 G80 Z100 M5 ；	Cancel the tool length compensation, the spindle stops.
N011 G91 G28 Z0 ；	Z axis zero return.
N027 M30 ；	Program end.

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

Format:

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

Function:

Start from the center of the circle; execute the arc interpolation for many times in the helical mode till the round groove of the programmed dimension is processed.

Explanations:

G110: CCW rough of the groove in the circle

G111: CW rough of the groove in the circle

X, Y: The starting position of XY plane is the circle center;

Z: The machining depth, the absolute position in G90; the position relative to the R standard plane in G91;

R: R standard plane position, the absolute position in G90; the position relative to the start position of the current block in G91;

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

L: The cut step width on XY plane, its absolute value should be less than the current tool diameter;

W: Feed in Z axis direction for the first time, the distance is below R standard plane. (If the feeding exceeds the groove bottom, directly process at the groove bottom.)

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

V: When cut rapidly, the distance is from point R to the unprocessed face;

D: Tool radius series number, the current tool radius value is taken based on the given series number;

K: Number of repeats.

Parameter range:

X,Y,Z,R:-999999.9999mm-999999.9999mm；

I, W, Q, V:-999999.9999mm-999999.9999mm,take the absolute value when the value is less than 0; the system alarms when it defaults or the value is equal to 0;

L: 0mm-999999.9999mm, the system alarms when it defaults or L is equal to 0;

D: 0-400, the system alarms when D is less than 0; it defaults or the current tool radius value is 0 when D is equal to 0;

K: 0-9999, it executes one time when it defaults; when K is equal to 0 only position is executed rather than the canned cycle, while the canned cycle and the parameter modal values are saved; the system alarms when K is less than 0.

Cycle process:

⑴ Rapidly position on the starting position (circle center) on XY plane;

⑵ Rapidly move downward point R level;

⑶ Cut down for the height of (R-W) in the cutting speed;

⑷ Helical mill the circle section from the center outside based on L value and L value increases each time.

⑸ Z axis rapidly returns to point R standard plane;

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

⑺ Z axis rapidly moves toward the height of（R-W）+V. If V value is bigger, the height will exceed that of point R;

⑻ Z axis is set as Q value each time, the increment will be continued to move downward the machined face;

⑼ Cycle the steps of（5）~（8）till process the round section of the total cutting depth;

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

Commanded path:

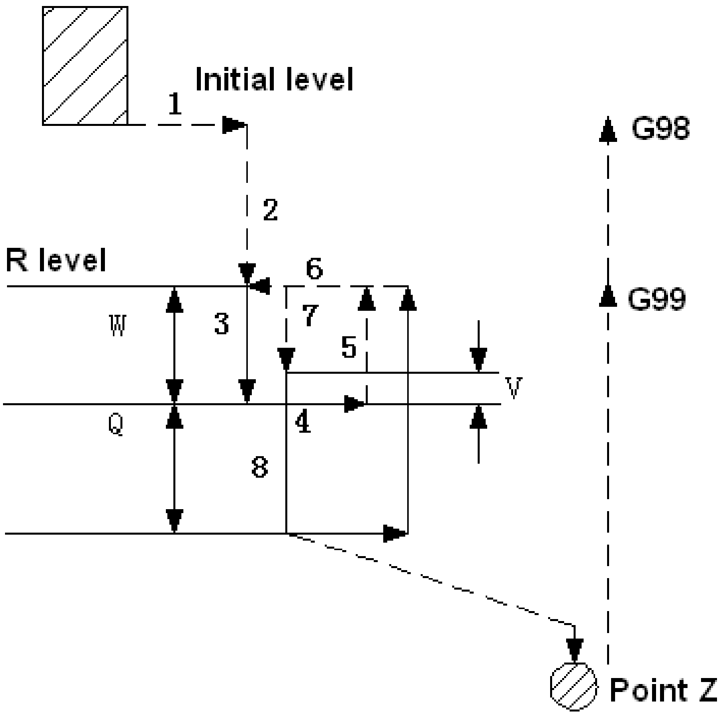


Fig. 3-4-22

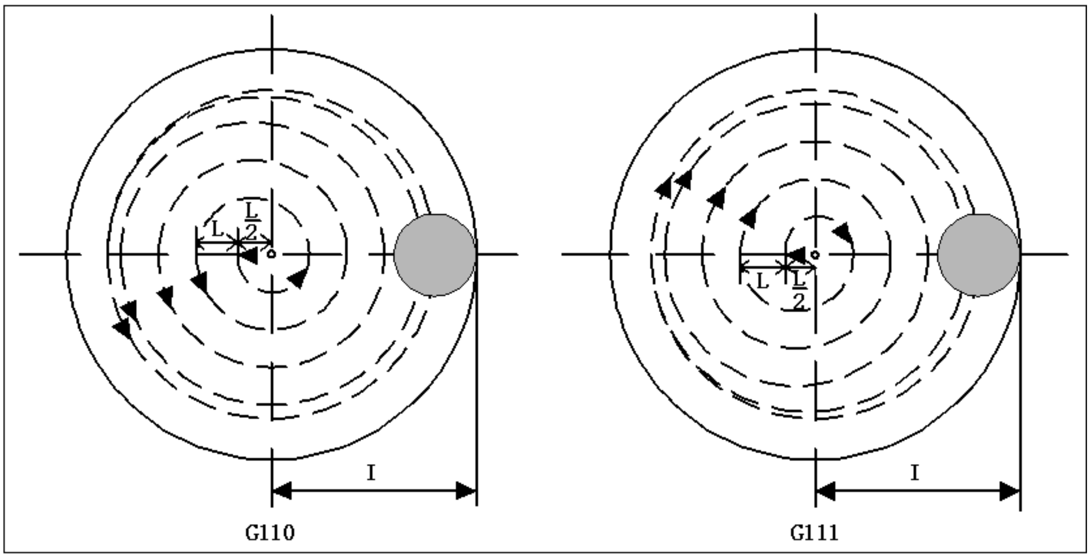


Fig. 3-4-23

Relative explanations:

1) In the cycle, commanding P is invalid; while P value is saved as the canned cycle modal one.

2) In G51 mode, when the command is used, I is scaled, while L and D aren't.

Restrictions:

1) The special canned cycle can only specified on G17 plane.

2) In a block that does not contain X, Y, Z, R or any other axes command, milling groove is not performed.

3) Specify I, L, W, Q, P, V and D in blocks that perform milling groove. If they are specified in a block that does not perform milling groove, they cannot be stored as modal data.

Example:

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

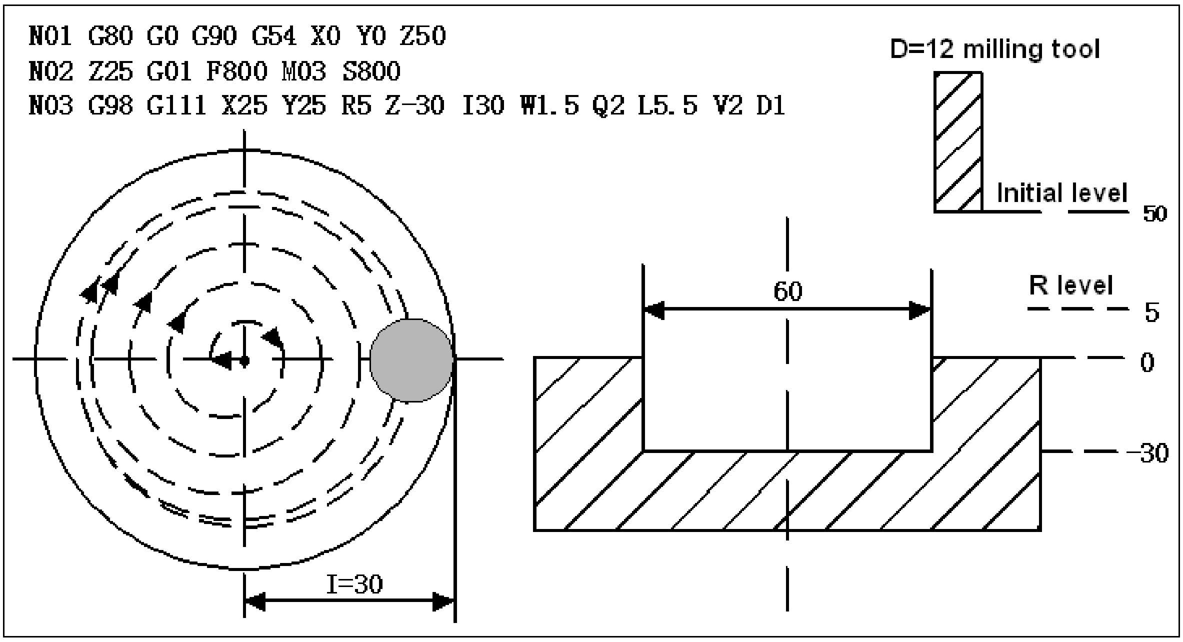


Fig. 3-4-24

3.4.18 Finish Milling the Whole Circle Cycle(G112/G113)

Format:

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

Function:

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

Explanations:

G112: Finish milling cycle in the full circle in CCW direction.

G113: Finish milling cycle in the full circle in CW direction.

X, Y: The starting position of XY plane is the finish milling circle center;

Z: The machining depth, the absolute position in G90; the position relative to the R standard plane in G91;

R: R standard plane position, the absolute position in G90; the position relative to the start position of the current block in G91;

I: Finish milling circle radius;

J: The distance between the finish milling start-up position and the finish milling circle;

D: The tool radius serial number, based on the specified serial number, the current tool radius value is taken.

K: Number of repeats.

Parameter range

X,Y,Z,R:-999999.9999mm-999999.9999mm；

I:-999999.9999mm-999999.9999mm, take the absolute value when I is less than 0; the system alarms when it defaults or I is equal to 0;

L: 0mm-999999.9999mm, the system alarms when it defaults or L is equal to 0;

D: 0-400, the system alarms when D is less than 0; it defaults or the current tool radius value is 0 when D is equal to 0;

K: 0-9999, it executes one time when it defaults; when K is equal to 0 only position is executed rather than the canned cycle, while the canned cycle and the parameter modal values are saved; the system alarms when K is less than 0.

Cycle process

(1) Rapidly position on the starting position (finish milling circle center) on XY plane;

(2) Rapidly position on the location deviating distance (I-L) from the finish milling circle center;

(3) Rapidly position on point R;

(4) Feed into the hole bottom point Z;

(5) Feed the transition arc 1;

(6) Feed the circles 2 and 3 with radius I;

(7) Feed the transition arc 4;

(8) Based on G98 or G99, rapidly return to the initial level or point R level.

(9) Rapidly position on the finish milling circle center (execute (I-L) distance).

Commanded path:

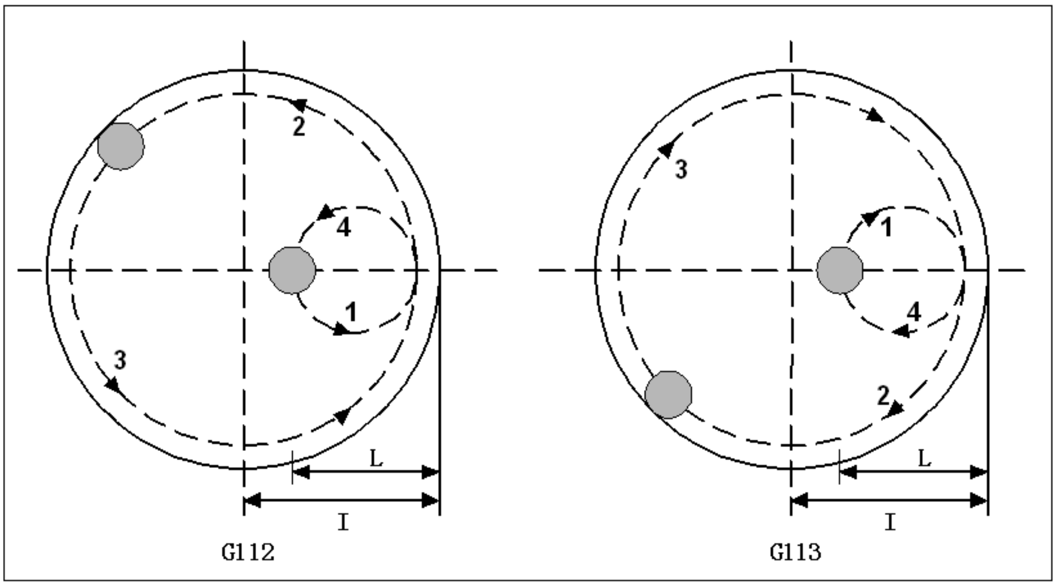


Fig. 3-4-25

Relative explanations:

1) In the cycle, commanding Q and P are invalid; while Q and P values are saved as the canned cycle modal one.

2) In G51 mode, when the command is used, I is scaled, while L and D aren't.

Restrictions:

1) The special canned cycle can only specified on G17 plane.

2) In a block that does not contain X, Y, Z, R or any other axes command, milling groove is not performed.

3) Specify I, L, W, Q, P, V and D in blocks that perform milling groove. If they are specified in a block that does not perform milling groove, they cannot be stored as modal data.

Example:

The canned cycle G112 commands finish milling one round groove, which is shown as the following figure:

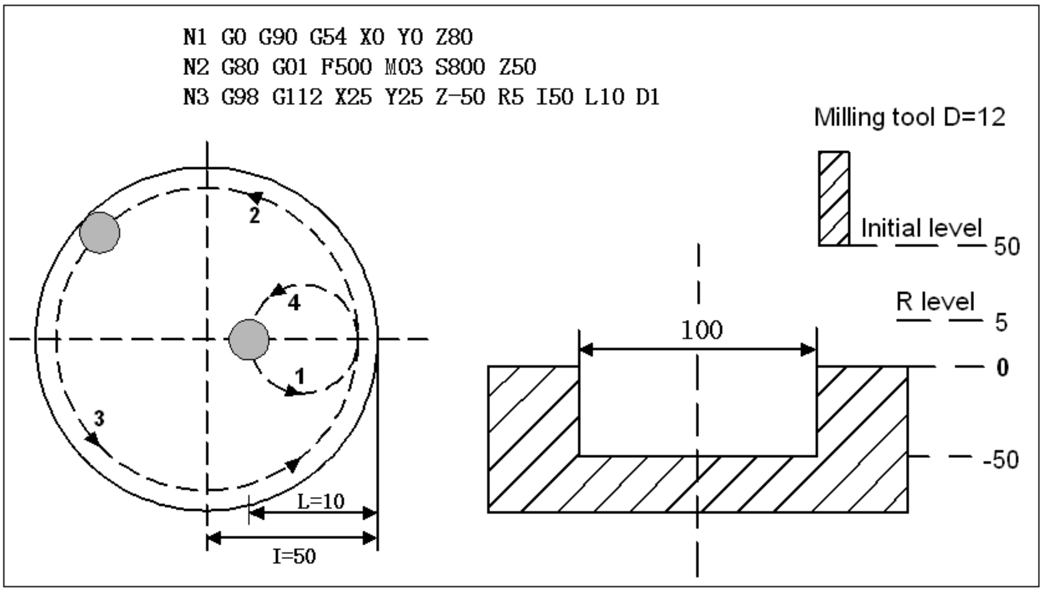


Fig. 3-4-26

3.4.19 Finish Milling Cycle Outside of the Circle (G114/G115)

Format:

$$G98/G99 \left\{ \begin{array}{l} \textbf{G116} \\ \textbf{G117} \end{array} \right. X\_ \ \ Y\_ \ \ R\_ \ \ Z\_ \ \ I\_ \ \ L\_ \ \ D\_ \ \ F\_ \ \ K\_;$$

Function:

The tool finish mills one whole circle based on the specified radius value and the direction outside of the circle, and the tool returns after complete the finish milling.

Explanations:

G116: Finish milling cycle outside of the circle in CCW direction.

G117: Finish milling cycle outside of the circle in CW direction.

X, Y: The starting position of XY plane is the finish milling circle center;

Z: The machining depth, the absolute position in G90; the position relative to the R standard plane in G91;

R: R standard plane position, the absolute position in G90; the position relative to the start position of the current block in G91;

I: Finish milling circle radius;

J: The distance between the finish milling start-up position and the finish milling circle;

D: The tool radius serial number, based on the specified serial number, the current tool radius value is taken.

K: Number of repeats.

Parameter range

X,Y,Z,R:-999999.9999mm-999999.9999mm;

I:-999999.9999mm-999999.9999mm, take the absolute value when I is less than 0; the system alarms when it defaults or I is equal to 0;

L: 0mm-999999.9999mm, the system alarms when it defaults or L is equal to 0;

D: 0-400, the system alarms when D is less than 0; it defaults or the current tool radius value is 0 when D is equal to 0;

K:0-9999, it executes one time when it defaults; when K is equal to 0 only position is executed rather than the canned cycle, while the canned cycle and the parameter modal values are saved; the system alarms when K is less than 0.

Cycle process

(1) Rapidly position on the starting position (finish milling circle center) on XY plane;

(2) Rapidly position on the location deviating distance (I-L) from the finish milling circle center;

(3) Rapidly position on point R;

(4) Cut and feed into the hole bottom;

(5) Arc interpolation is executed based on the transition arc 1 as the path;

(6) Whole circle interpolation is executed based on the arcs 2 and 3 as the path;

(7) Arc interpolation is executed based on the transition arc 4 as the path, and then return to the starting position;

(8) Based on G98 or G99, return to the initial level or point R level.

(9) Rapidly position on the finish milling circle center (execute (I-L) distance).

Commanded path:

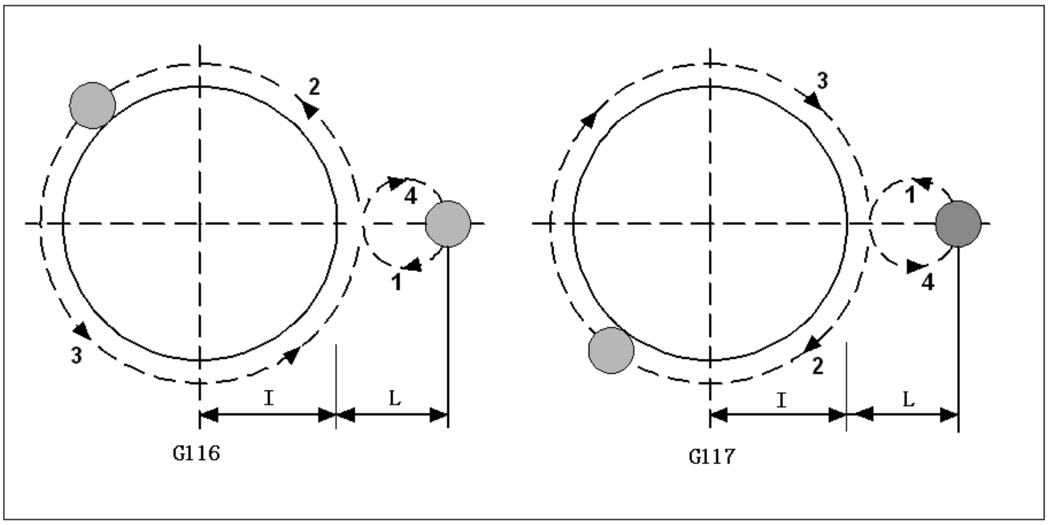


Fig. 3-4-27

Relative explanations:

1) During the finish milling outside of the circle, the interpolation direction of the transition arc and that of the finish milling arc are different, the interpolation direction in the explanation is the one of the finish milling arc.

2) In the cycle, commanding Q and P are invalid; while Q and P values are saved as the canned cycle modal one.

3) In G51 mode, when the command is used, I is scaled, while L and D aren't.

Restrictions:

1) The special canned cycle can only specified on G17 plane.

2) In a block that does not contain X, Y, Z, R or any other axes command, milling groove is not performed.

3) Specify I, L, W, Q, P, V and D in blocks that perform milling groove. If they are specified in a block that does not perform milling groove, they cannot be stored as modal data.

Example:

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

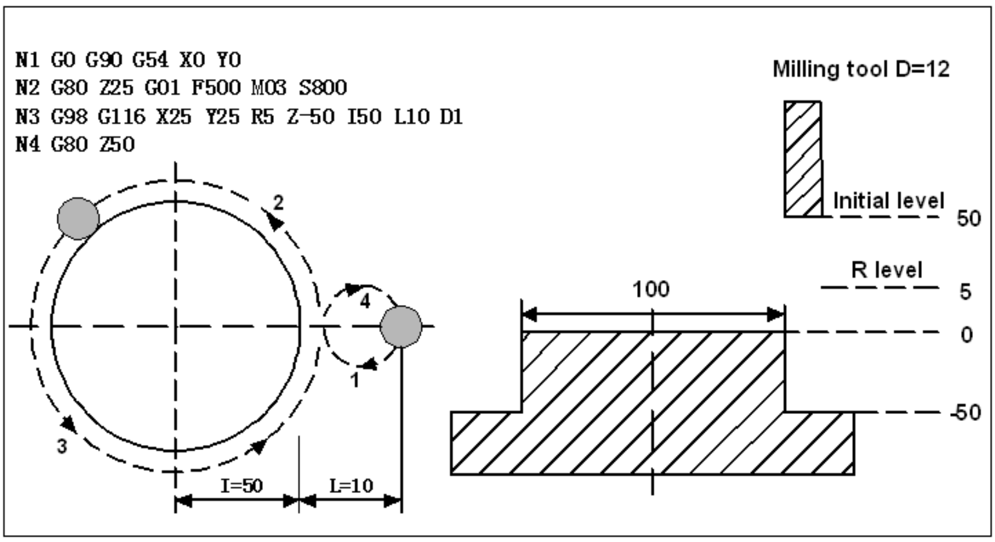


Fig. 3-4-28

3.4.20 Roughing Rectangle Groove (G130/G131)

Format:

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

Function:

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

Explanation:

G130: CCW roughing the rectangular groove;

G131: CW roughing the rectangular groove;

X, Y: The start position of XY plane is the rectangle center;

Z: The machining depth, it is the absolute position (G90), it is the position relative to R standard plane (G91);

R: R is the standard plane position, it is the absolute position (G90), and it is the position relative to the start position of the block (G91);

I: The length of the rectangular groove in X axis direction;

J: The length of the rectangular groove in Y axis direction;

L:The cut width increment on XY plane, $L \leq \frac{2+\sqrt{2}}{4}D$ （D is the tool diameter）;

W: Cutting depth in Z axis direction for the first time is the distance from R standard plane (If the cutting depth for the first time exceeds the groove bottom, directly process at the groove bottom.)

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

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

U: The corner arc radius, if it is omitted, it means there isn't any corner arc transition; |U| is greater than or equal to D/2, less than or equal to the smaller value of I/2 or J/2;

D: The cutter radius serial number, the present cutter radius value is taken based on the specified serial number.

K: Number of repeats.

Parameter range:

X, Y, Z, R: -999999.9999mm-999999.9999mm；

I, J, W, Q, V:-999999.9999mm-999999.9999mm, when the value is less than 0, take the absolute one; the system alarms when it defaults or equal to 0;

U: -499999.9999mm-499999.9999mm, when U is less than 0, take the absolute value; there isn't any corner arc transition when it defaults or U is equal to 0, while the system doesn't alarm;

L: 0mm-999999.9999mm, the system alarms when it defaults or L is equal to 0;

D: 0-400, the system alarms when D is less than 0; it defaults the present cutter radius value as 0 when it defaults or D is equal to 0;

K: 0-9999, it executes one time when it defaults; when K is equal to 0, just positioning is executed without the canned cycle, while the modal values of the canned cycle and those of parameters are saved; the alarm is issued if K is less than 0.

Cycle process:

1) Rapidly position on the starting position of XY plane (that is the rectangle center);
2) Rapidly cut down point R level;
3) Cut downward（R-W）height at the cutting speed;
4) From the center outside, roughing is completed based on L value increase each time;
5) Z axis rapidly return to point R level;
6) X and Y axes rapidly position on the rectangle center;
7) Z axis rapidly moves to the height（R-W）+V. If V value is bigger, height will exceed point R;
8) Z axis is set by Q value each time, and the increment continuously moves downward the machining face;
9) Cycle（5）~（8）till complete machining the rectangle face of the total cutting depth;
10) Return to the initial level or point R level set by G98 or G99;
11) Return to the center position of the rectangle groove.

Commanded path:

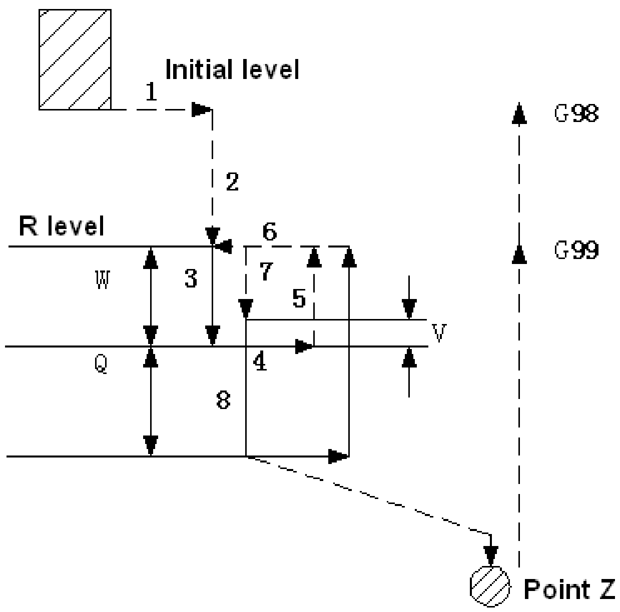


Fig. 3-4-29

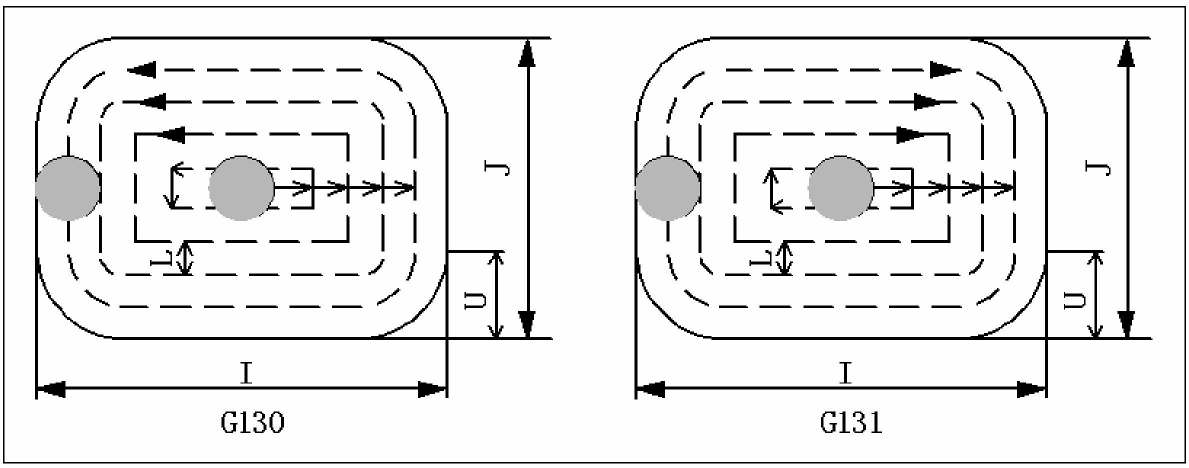


Fig. 3-4-30

Relative explanations:

1) In the cycle, commanding P is invalid; while P value is saved as the canned cycle modal one.

2) In G51 mode, when the command is used, I, J and U are scaled, while L and D aren't.

Restrictions:

1) The special canned cycle can only be specified on G17 plane.

2) In a block that does not contain X, Y, Z, R or any other axes command, milling groove is not performed.

3) Specify I, L, W, Q, P, V and D in blocks that perform milling groove. If they are specified in a block that does not perform milling groove, they cannot be stored as modal data.

Example:

The canned cycle G130 commands roughing one rectangular groove, which is shown as the following figure:

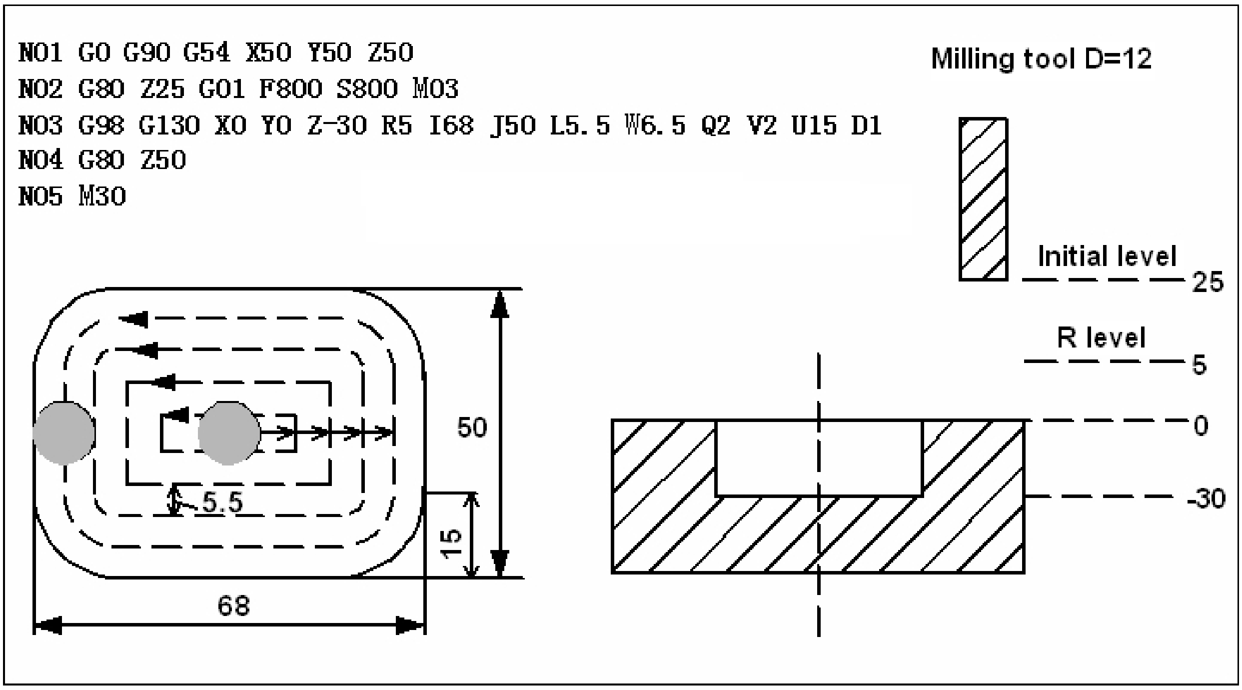


Fig. 3-4-31

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

Format:

G132
G98/G99 X_ Y_ R_ Z_ I_ J_ D_ L_ U_ F_ K_;
G133

Function:

The tool finishes in the rectangle in the specified width and direction and returns after the finishing completes.

Explanations:

G132: CCW finishing cycle in the rectangle;

G133: CW finishing cycle in the rectangle

X, Y: The start position of XY plane is the rectangle center;

Z: The machining depth, it is the absolute position (G90), and it is the position relative to R standard plane (G91);

R: R is the standard plane position, it is the absolute position (G90), and it is the position relative to the start position of the block (G91);

I: Width of rectangle X axis direction;

J: Width of rectangle Y axis direction;

L: The distance between the finishing cutting point and the rectangular side in X axis direction;

U: The corner arc radius; if U is omitted, it means there isn't any corner arc transition; |U| is greater than or equal to D/2, less than or equal to the smaller value of I/2 or J/2;

D: The cutter radius serial number, the present cutter radius value is taken based on the specified serial number;

K: Number of repeats

Parameter range:

X,Y,Z,R: -999999.9999mm-999999.9999mm；

I, J:-999999.9999mm-999999.9999mm; take the absolute one when the value is less than 0; the alarm is issued when it defaults or the value is equal to 0;

U: -499999.9999mm-499999.9999mm; When U is less than 0, take the absolute value; there isn't any corner arc transition when it defaults or U is equal to 0 and the system doesn't alarm.

L: 0mm-999999.9999mm; The system alarms when it defaults or L is equal to 0;

D:0-400；The system alarms when D is less than 0; it defaults the present cutter radius value as 0 when it defaults or D is equal to 0;

K:0-9999; It executes one time when it defaults; when K is equal to 0, just positioning is executed without the canned cycle, while the modal values of the canned cycle and those of parameter are saved; the alarm is issued if K is less than 0.

Cycle process:

1) Rapidly position on the starting position of XY plane (that is the rectangle center);
2) Rapidly position at the location deviating (I/2-L) distance from the rectangle center;
3) Rapidly cut downward point R level;
4) Cutting feed into the bottom of the hole;
5) From the starting point, the arc interpolation takes the transition arc 1 as the path;
6) The linear and arc interpolations take the transition arc 2-3 as the paths;
7) The arc interpolation takes the transition arc 4 as the path to return the starting point;
8) Return to the initial level or point R level selected by G98 or G99;
9) Rapidly position on the rectangle center (execute (I/2-L) distance).

Commanded path:

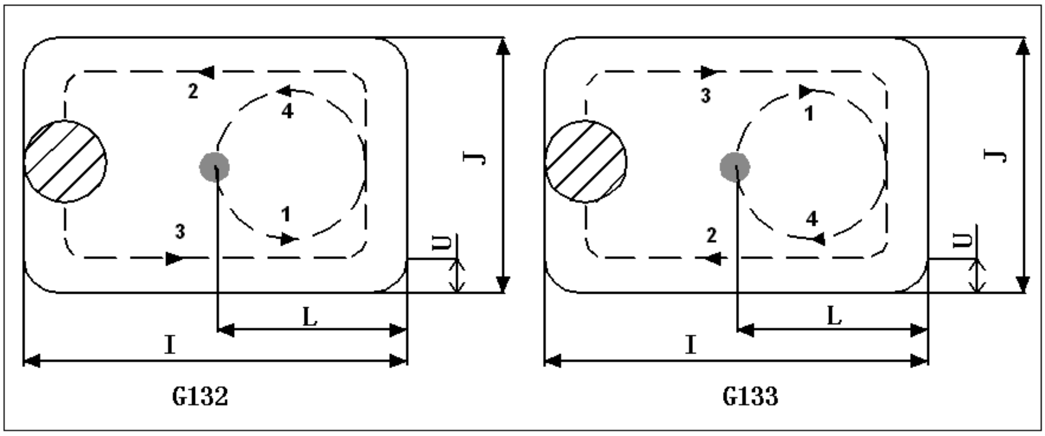


Fig. 3-4-32

Relative explanations:

1) In the cycle, commanding Q and P are invalid, while Q and P values are saved as the canned cycle modal values.

2) In G51 mode, when the command is used, I, J and U are scaled, while L and D aren't.

Restrictions:

1) The special canned cycle can only be specified on G17 plane.

2) In a block that does not contain X, Y, Z, R or any other axes command, milling groove is not performed.

3) Specify I, L, W, Q, P, V and D in blocks that perform milling groove. If they are specified in a block that does not perform milling groove, they cannot be stored as modal data.

Example:

The canned cycle G133 commands finishing the rectangular groove which has already been roughed, which is shown as below:

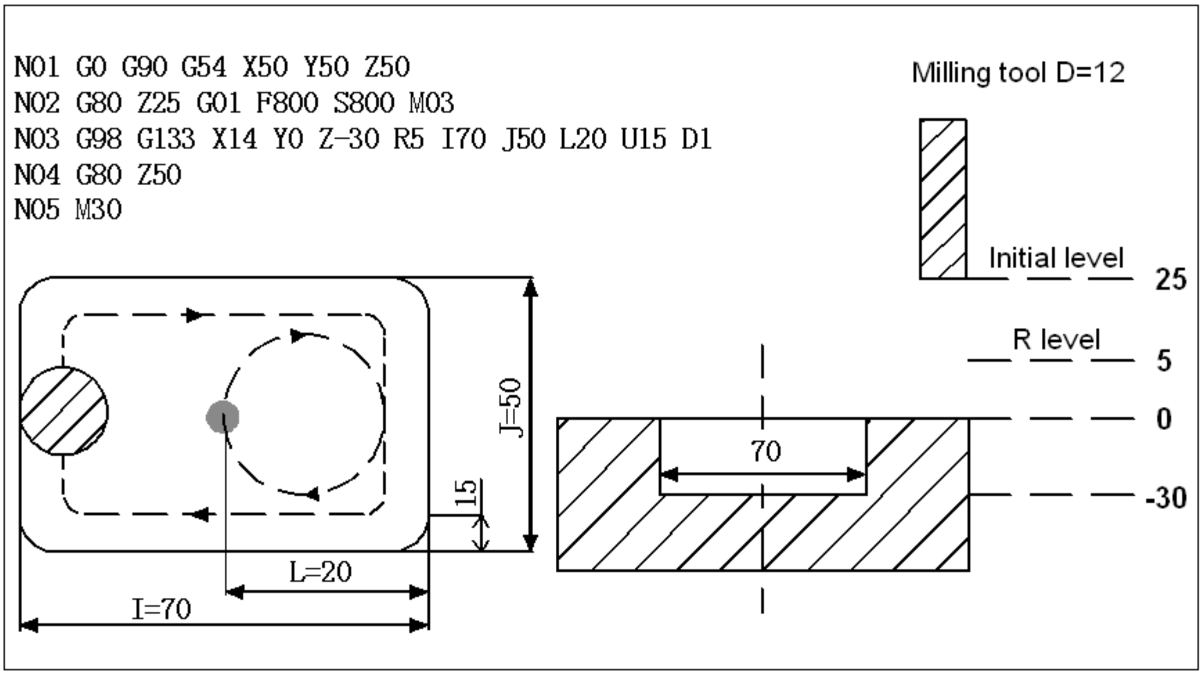


Fig. 3-4-33

3.4.22 Finishing Cycle outside of the Rectangle (G136/G137)

Format:

	G136	
G98/G99		X_ Y_ R_ Z_ I_ J_ D_ L_ U_ F_ K_
	G137	

Function:

The tool finishes outside of the rectangle in the specified width and direction, and it returns after the finishing completes.

Explanations:

G136: CCW finishing cycle outside of the rectangle;

G137: CW finishing cycle outside of the rectangle;

X, Y: The starting position of XY plane is the rectangle center;

Z: The machining depth, it is the absolute position (G90), and it is the position relative to R standard plane (G91);

R: R is the standard plane position, it is the absolute position (G90), and it is the position relative to the starting position of the block (G91);

I: Width of rectangle X axis direction;

J: Width of rectangle Y axis direction;

L: The distance between the finishing cutting point and the rectangular side in X axis direction;

U: The corner arc radius; if U is omitted, it means there isn't any corner arc transition; |U| is greater than or equal to D/2, less than or equal to the smaller value of I/2 or J/2;

D: The cutter radius serial number, the present cutter radius value is taken based on the specified serial number.

K: Number of repeats.

Parameter range:

X,Y,Z,R: -999999.9999mm-999999.9999mm;

I, J: -999999.9999mm-999999.9999mm, take the absolute one when the value is less than 0; the alarm is issued when it defaults or the value is equal to 0;

U: -499999.9999mm-499999.9999mm, when U is less than 0, take the absolute value; there isn't any corner arc transition when it defaults or U is equal to 0 and the system doesn't alarm.

L: 0mm-999999.9999mm; the system alarms when it defaults or L is equal to 0;

D: 0-400; The system alarms when D is less than 0; it defaults the present cutter radius value as 0 when it defaults or D is equal to 0;

K: 0-9999; It executes one time when it defaults; when K is equal to 0, just positioning is executed without the canned cycle, while the modal values of the canned cycle and those of parameter are saved; the alarm is issued if K is less than 0.

Cycle process:

1) Rapidly position at the starting position on XY plane (that is the rectangle center);
2) Rapidly position on the location deviating (I/2-L) distance from the rectangle center;

3) Rapidly cut downward point R level;
4) Cutting feed into the bottom of the hole;
5) From the start point, the arc interpolation takes the transition arc 1 as the path;
6) The linear and arc interpolations take the transition arc 2-3 as the paths;
7) The arc interpolation takes the transition arc 4 as the path to return the starting point;
8) Return to the initial level or point R level, which is selected by G98 or G99;
9) Rapidly position on the rectangle center (execute (I/2-L) distance).

Commanded path:

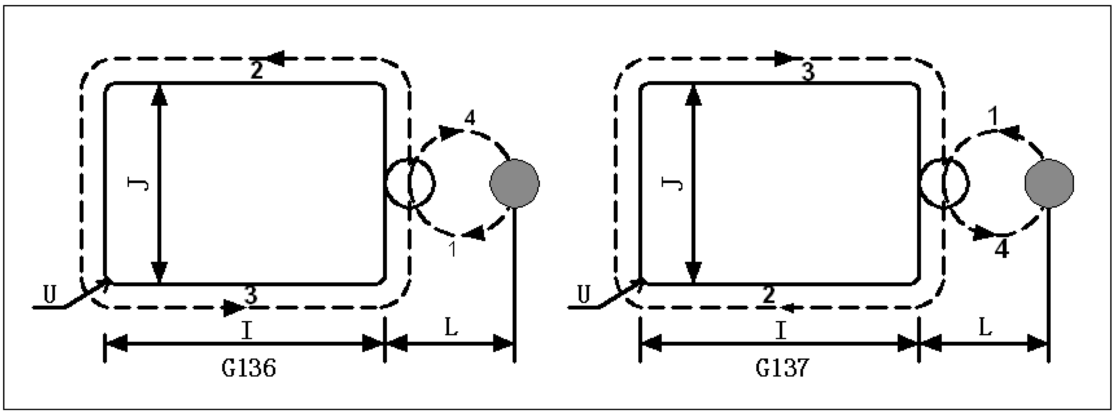


Fig. 3-4-34

Relative explanations:

1) During the finishing outside of the rectangle, the interpolation directions of the transition arc and the finishing arc are different, the interpolation direction in the explanation is that of the finish milling arc.

2) In the cycle, commanding Q and P are invalid, while Q and P values are saved as the canned cycle modal values.

3) In G51 mode, when the command is used, I, J and U are scaled, while L and D aren't.

Restrictions:

1) The special canned cycle can only be specified on G17 plane.

2) In a block that does not contain X, Y, Z, R or any other axes command, milling groove is not performed.

3) Specify I, L, W, Q, P, V and D in blocks that perform milling groove. If they are specified in a block that does not perform milling groove, they cannot be stored as modal data.

Example:

The canned cycle G136 commands finishing the rectangular section which has already been roughed, which is shown as below:

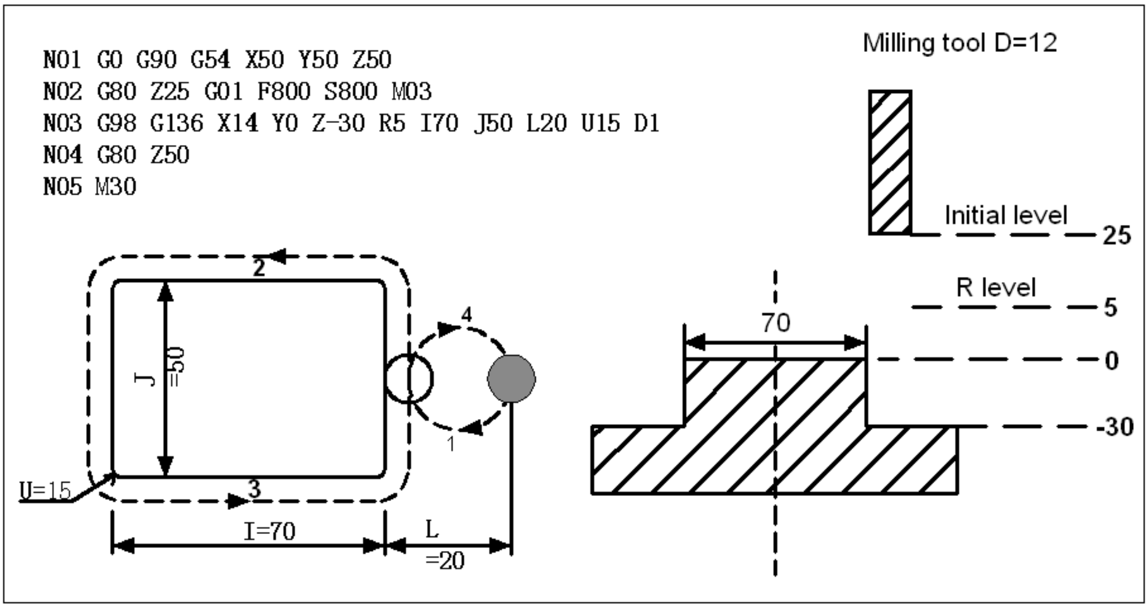


Fig. 3-4-35

3. 5 Tool compensation function

3.5.1 Tool length compensation G43, G44, G49

Format:

At present, the system supports the two tool length offset modes A/B, which is set by parameter No: 2600#1:

Mode A:

G43 } Z_ H_ ;
G44

Mode B:

G17 G43 Z_H;
G17 G44 Z_H;
G18 G43 Y_H;
G18 G44 Y_H;
G19 G43 X_H;
G19 G44 X_H;

Cancel the tool length offset mode: G49; or H0;

Function:

G43 specifies the positive compensation of the tool length.
G44 specifies the negative compensation of the tool length.
G49 cancels the tool length compensation.

Explanations:

The finishing position of Z axis movement is commanded to move positively or negatively, which is set by the offset value in the memorizer. The function can be used by setting the difference between the tool length assumed during programming and the actual tool length of the tool used into the offset memory. It is possible to compensate the difference without changing the program.

Specify the direction of offset with G43 or G44. Select a tool length offset value from the offset memory by H commands.

1. Offset direction

G43: Positive offset (the most ordinary offset mode)

G44: Negative offset

When G43 is specified, the tool length offset value (stored in offset memory) specified with the H code is added to the coordinates of the end position specified by a command in the program. When G44 is specified, the same value is subtracted from the coordinates of the end position. The resulting coordinates indicate the end position after compensation, regardless of whether the absolute or incremental mode is selected. When Z axis movement command is omitted, the same method is introduced as below:

$$\left.\begin{matrix}G43\\G44\end{matrix}\right\} \text{G91 ZO H——;}$$

The offset value in G43 is positive direction while negative in G44.

G43 and G44 are modal G codes, and they remain valid before there are other G codes in same group.

2. Specifying the tool length offset value

The tool length offset value assigned to the number (offset number) specified in the H code is selected from offset memory and added to or subtracted from the programming value of Z axis. The offset number is specified by H00～H400.

The offset value corresponds to the offset number and the length offset value is set in the memorizer through MDI/LCD, the span is shown as below:

	Input in mm	Input in inch
Offset value (profile)	-999.9999mm～999.9999mm	-999.9999 inch～999.9999 inch
Offset value (wearing)	The range is set by parameter 2611; The max. setting range is: 0～100.	

The tool length offset value corresponding to offset No.00, that is, H00 always means 0. It is impossible to set any other tool length offset value to H00.

Notes:

The value of parameter No.2611 is 100, the actual range of the wearing compensation is: -100～100.

During the machining process of the program running, the current length compensation value is rewritten in JOG mode; the rewritten compensation value won't become valid immediately because the system is with the pre-read process.

3. Valid sequence of the offset number

Once the length offset mode is set, the current offset number becomes valid; while the offset number is changed, the new offset value will replace the old one immediately.

O××××；		
H01；		
G43 Z10；	(1)	Offset number H01 becomes valid
G44 Z20 H02；	(2)	Offset number H02 becomes valid
H03；	(3)	Offset number H03 becomes valid
G49 Z0；	(4)	Offset is cancelled.
M30；		

4. Tool length offset cancel

To cancel tool length offset, specify G49 or H00. After G49 or H00 is canceled, the system immediately cancels the offset mode if it is with the movement command.

Notes:

After tool length offset B is executed along two or more axes, offset along all the axes is cancelled by specifying G49. If H00 is specified, only offset along an axis perpendicular to the specified plane is cancelled.

Setting or cancelling the length compensation must be executed with the movable command.

5. G53, G28 or G30 command in the tool length offset mode

In the tool length offset mode, specify G53, G28 or G30, the offset vector of the tool length offset axis is canceled after moving into the specified point. Among them, G53 is canceled after moving toward the commanded position; G28 and G30 are canceled after moving into the reference position. However, the mode code doesn't switch into G49, and the axes except for the tool length offset axes aren't canceled. When G53 and G49 are in the same block, the length offset is canceled after all axes move toward the commanded position; when G28 or G30 share the same block with G49, the length offset vector is canceled after all the axes move toward the intermediate point. In the tool length offset, the tool length offset vector canceled by G53, G28 or G30 is restored in the next block.

6. The practical example of the tool length compensation

The example of the tool length compensation (machining the holes of #1, #2, #3)

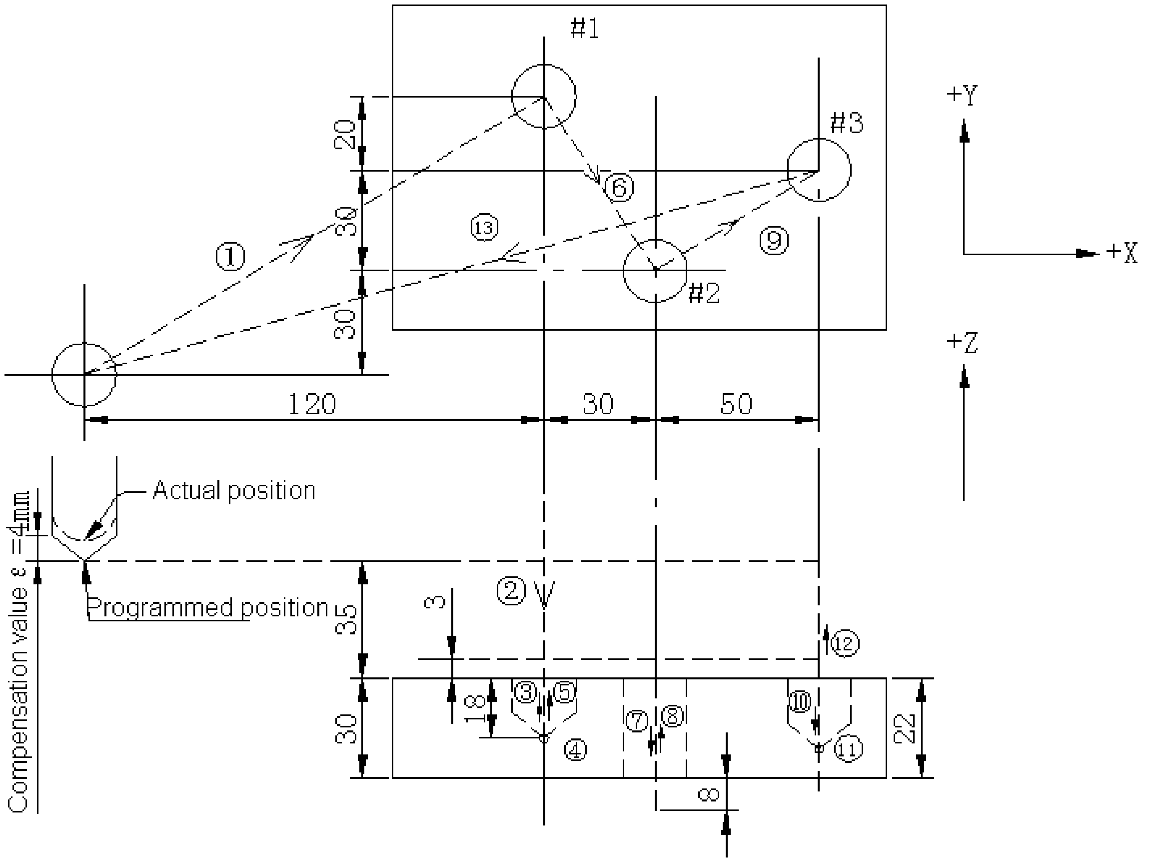


Fig. 3-5-1

H01= －4.0（offset value）

N1　G91　 G00　X120.0　Y80.0　；···············(1)

N2　G43　 Z-32.0　H01　；···············Z-3

N3　G01　 Z-21.0　F1000　；···············

N4　G04　 P2000　；···············P20

N5　G00　 Z21.0　；···············Z21

N6　X30.0　Y-50.0　；···············30.

N7　G01　 Z-41.0　；··············· Z-

N8　G00　 Z41.0　；··············· Z4

N9　X50.0　Y30.0　；···············50.

N10　G01　 Z-25.0　；···············Z-25

N11　G04　 P2000　；···············P200

N12　G00　 Z57.0　 H00　；··············· Z5

N13　X-200.0　 Y-60.0　；···············(13)

Notes:

1. When the tool length offset value is changed due to a change of the offset number, the new tool length offset value is not added to the old one.

H01············Offset value 20.0
H02············Offset value 30.0
G90　G43　Z100.0　H01；······ will move to 120.0

G90 G43 Z100.0 H02；…… will move to 130.0

2. Pay attention to two special examples:

Example 1:

```
G18 G40 G49 G1 X0 Y0 Z0
G43 H01 X0 Y0 Z10     (H01=6)
G41 D01 X10 Y10 Z5    (D01=8)
X15 Y15 Z15
G98 G81 X20 Y20 Z-10 R5
G80 X30 Y30 Z30
X40 Y40 Z40
M30
```

In the above example, the length compensation is mode A, the radius compensation is mode B. In G18/G19 plane, when the canned cycle is mode A and Z axis is positioned on the initial plane, Z axis will move because the tool compensation is cancelled, the operator should pay special attention to it.

Example 2

```
G54 G40 G49 G90 G01 X-10 Y-10 Z-10 G19
G43 H01 X0 Y0 Z0           H01=6
G42 D01 X10 Y10 Z10         D01=8
X15 Y15 Z15
X20 Y20 Z20
X30 Y30 Z30
G49 X40 Y40 Z40
M30
```

In the above example, the length compensation is mode A, the radius compensation is mode B. On G18/G19 plane, the length compensation and the radius compensation will occur on Z axis, in such situation, the system firstly processes the length compensation and then the radius one.

3.5.2 Overview of cutter compensation C（G40～G42）

Format:

1) G00 (G01) G41 D_IP__
2) G00 (G01) G42 D_IP_
3) G40 IP_

Function:

G41 specifies cutter compensation left.
G42 specifies cutter compensation right.
G40 cancels the cutter compensation.

Explanations:

1. The cutter compensation function

In the following figure, the tool of radius R processes the work piece specified by A, and the path corresponding to the tool center is position B relative to A and the distance is R. Like this, the distance which the tool leaves the work piece is called as the offset.

Therefore, the programmer can program the work piece shape through the cutter compensation function; during the machining, if the tool radius (offset value) is measured and set in CNC, the tool path is offseted as path B.

When the machining ends, the cutter compensation must be cancelled to make the tool return to the starting position

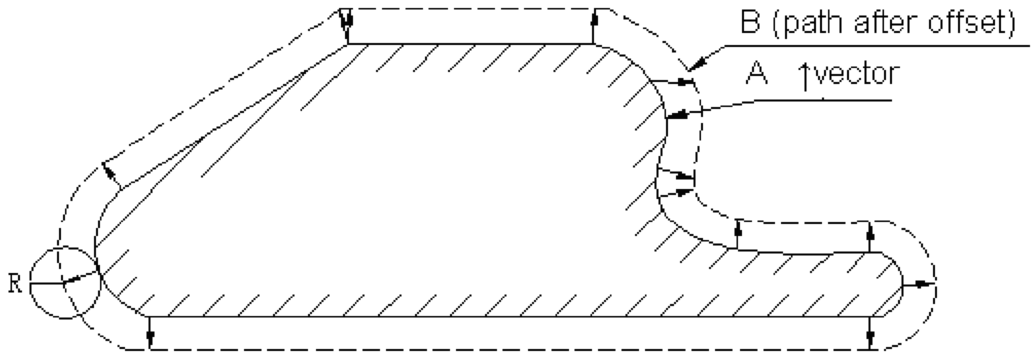


Fig. 3-5-2

2. Offset amount (D)

The offset amount is set by the command D in the program.

The offset amount range is set as below:

	Input in mm	input in inch
Offset value (profile)	-999.9999mm～999.9999mm	-999.9999 inch ～999.9999 inch
Offset value (wearing)	The range is set by parameter 2611, the maximum range is 0～100.	

Note:

The value of parameter No.2611, the actual range of wearing compensation is: -100～100.

The tool length offset value corresponding to offset No.00, that is, D00 always means 0. It is impossible to set any other tool length offset value to D00.

During the machining process of the program running, the current radius compensation value is rewritten in JOG mode; the rewritten compensation value won't become valid immediately because the system is with the pre-read process.

The offset number can specify D000～D400. The diameter value or the radius value is set by the bit parameter N0:2601#7.

3. Plane selection and vector

Offset calculation is carried out in the plane determined by G17, G18 or G18. This plane is called the offset plane. For example, XY is selected on G17 plane, calculate the offset value in the program（X, Y）or （I, J）. The axes not on the offset planes are still executed based on the commanded value. If the offset plane is changed in the offset mode, the compensation should be cancelled; otherwise, the alarm NO.037 is issued.

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

4. Setting cutter compensation

At the CNC system power-on, the cutter compensation mode is in the cancel one. Only in G41 or G42 and in the offset plane, the cutter compensation mode can be set with D code except for D0. In G02 or G03 mode, if the cutter compensation is set, the alarm NO.034 is issued.

When the cutter compensation is set, if there isn't any command of the movement distance, the tool doesn't move. The cutter compensation is set when the next block is with the command.

5. Cancelling cutter compensation

In the cutter compensation, the cutter compensation is cancelled in the block with command G40 or with compensation offset number D0.

In G02 or G03 mode, the cutter compensation is cancelled, the alarm No.034 is issued.

When the cutter compensation is cancelled, if there isn't any command of the movement distance, the tool doesn't move. The cutter compensation is cancelled when the next block is with the command.

6. Cutter compensation offsetting

In the cutter compensation, it can be realized by G00, G01, G02 or G03. When many blocks not move are specified, the quantity of blocks exceeds 8, the alarm No.054 is issued.

7. Switch between the cutter compensation left and right

Normally, the cutter compensation offset direction is from left to right, or from right to left, and the offset should be cancelled. However, positioning (G00) and the linear interpolation (G01) can be directly switched without cancelling the offset, the current tool path is shown as the following figure:

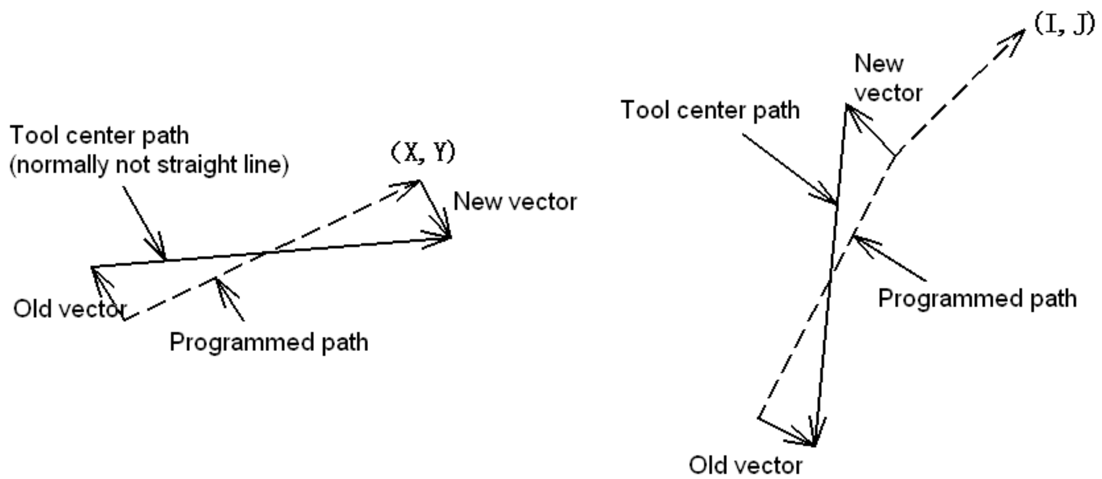


Fig. 3-5-3

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

8. Change of the cutter compensation value

In general, the cutter compensation value should be changed in the cancel mode, when changing tools. However, the positioning (G00) and the linear interpolation (G01) are executed in the offset state;

the situation is shown as below.

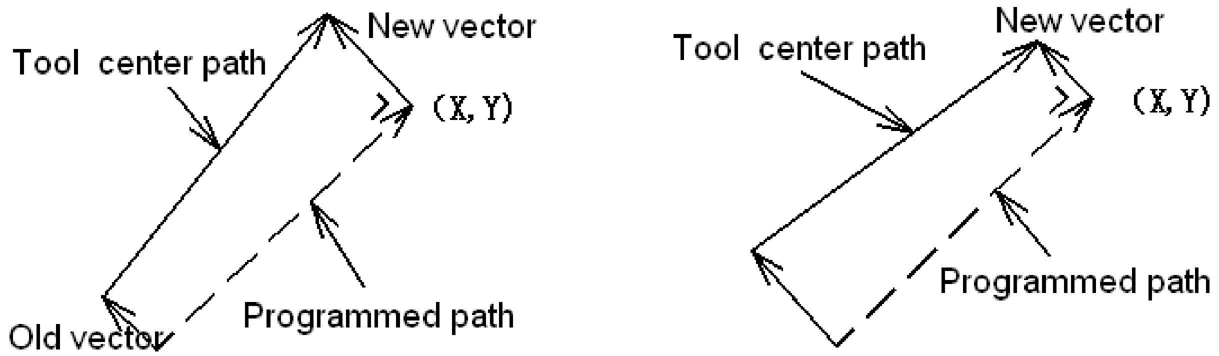


Fig. 3-5-4

9. Positive/negative offset amount and the tool center path

If the offset amount is negative (-), distribution is made for a figure in which G41's and G42's are all replaced with each other on the program. Consequently, if the tool path is passing around the outside of the work piece, it will pass around the inside, and vice versa:

When a tool path is programmed as in figure (A), if the offset amount is made negative (-), the tool center moves as in figure (B). Same, when a tool path is programmed as in figure (B), if the offset amount is made negative (-), the tool center moves as in figure (A).

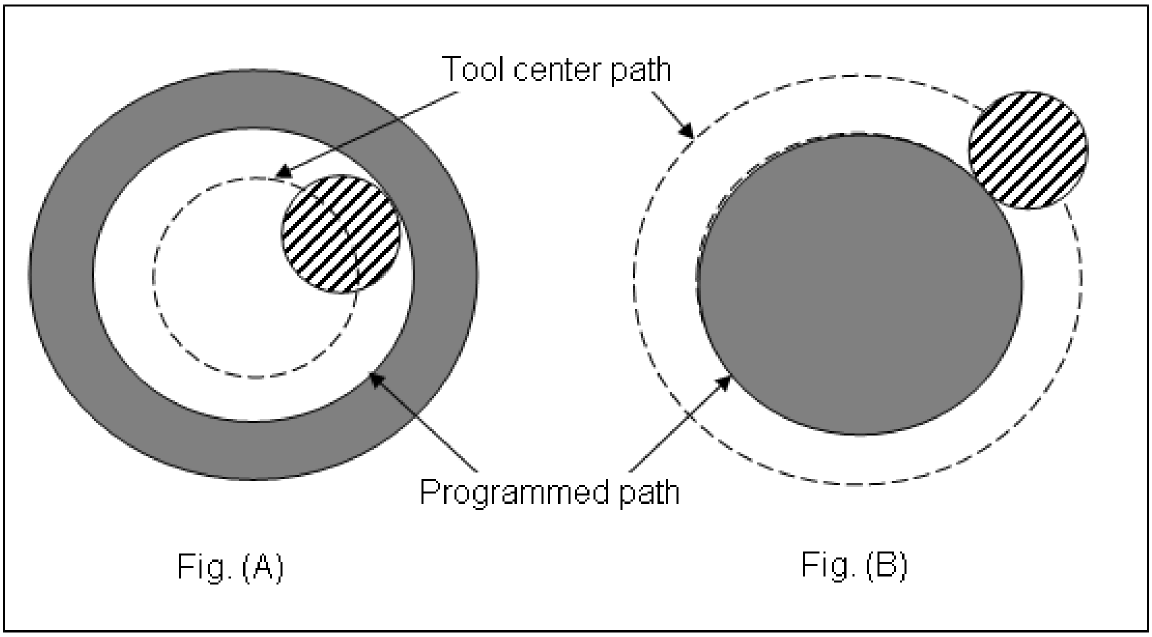


Fig. 3-5-5

Left or right compensation depends on that the offset direction is on the left or right side of the tool relative work piece movement direction and the work piece keeps still. The system enters the offset mode by G41 or G42, the system enters the offset cancel mode by G40.

10. Note: In the offset mode, the positioning (G06) occurs unidirectional and it functions, the system alarms.

11. The tool change command will automatically cancel the compensation amount, please specify D code of the relative cutter compensation, again after tool change.

12. The cutter compensation mistake alarm: Because at lease three block are pre-read for calculating the cutter compensation, if the mistake alarm occurs, the operator should comprehensively analyze the front and after blocks based on the cursor stopping position.

The program of the cutter compensation is shown as below:

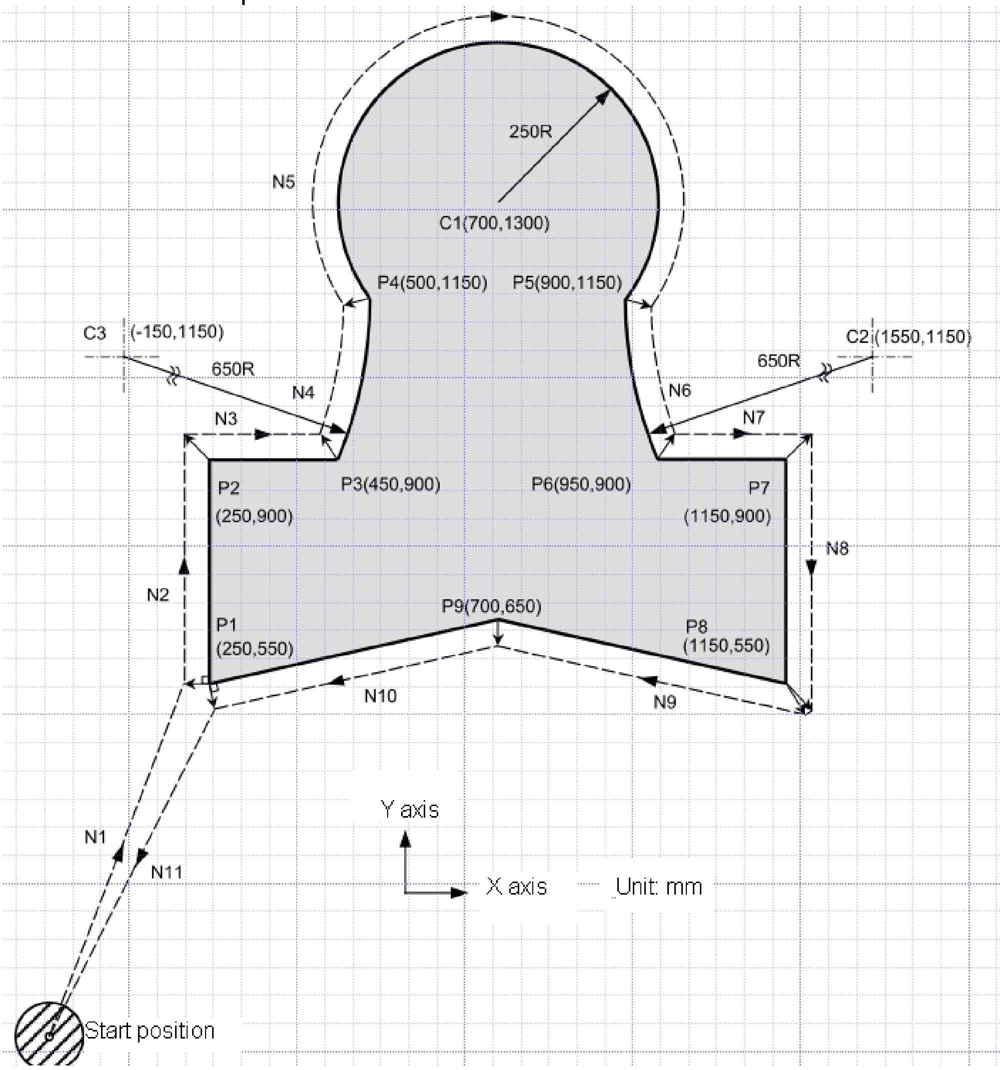


Fig. 3-5-6

G92 X0 Y0 Z0;…………………………………..Specifies absolute coordinates;
The tool is positioned at the start position (X0, Y0, Z0).

N1 G90 G17 G00 G41 D07 X250.0 Y550.0; Starts cutter compensation (start-up). The tool is shifted to the left of the programmed path by the distance specified in D07. In other words the tool path is shifted by the radius of the tool (offset mode) because D07 is set to 15 beforehand (the radius of the tool is 15mm).

N2 G01 Y900.0 F150;……………………….Specifies machining from P1 to P2.
N3 X450.0……………………………………Specifies machining from P2 to P3.
N4 G03 X500.0 Y1150.0 R650.0…………..Specifies machining from P3 to P4.
N5 G02 X900.0 R-250.0…………………….Specifies machining from P4 to P5.
N6 G03 X950.0 Y900.0 R650.0;…………...Specifies machining from P5 to P6.
N7 G01 X1150.0……………………………..Specifies machining from P6 to P7.
N8 Y550.0……………………………………Specifies machining from P7 to P8.
N9 X700.0 Y650.0…………………………...Specifies machining from P8 to P9.
N10 X250.0 Y550.0…………………………...Specifies machining from P9 to P1.
N11 G00 G40 X0 Y0;………………………..Cancels the offset mode.
The tool is returned to the start position (X0, Y0, Z0).

Start the block N1, in the block, the offset cancel mode is changed into offset mode (G41). At the block end position N1, the tool center is offseted based on the radius vertical to the next block path (N1 ～N2). The tool compensation value is specified by D01, namely, the offset number is 1 and G41 means the tool is compensated on the left side.

The system automatically completes the tool compensation after programming and starting the work piece shape N1→N2→N8→N9→N1.

In the block N10, the tool returns to the starting position through command G40 and the offset is canceled. At the end position of the block N11, the tool center is vertical to the programming path (N11～N1).

At the end position of the block, G40 must be commanded (the program is cancelled).

3.5.3 Details of cutter compensation

The detailed explanation of the cutter compensation C is as the following, which consists of the following subsections:

(1) General

(2) Setting the cutter compensation

(3) Tool movement in tool compensation

(4) Cancelling the cutter compensation

(5) Temporary cutter compensation cancel（G53 G28 G30 G29）

(6) Overcutting by cutter compensation

(7) Interference check

3.5.3.1 Overview

Inner side and outer side

Inner side: When an angle of intersection created by two blocks specified with move commands is more than 180°, it is referred to as “inner side”.

Outer side: When the angle is between 0°～180°, it is referred to as “outer side”.

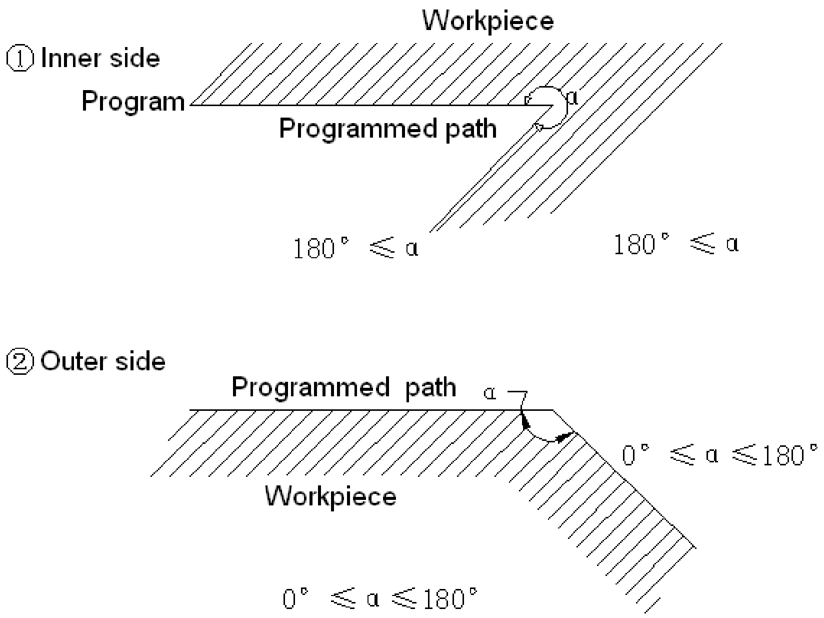


Fig. 3-5-7

The following symbols are used in subsequent figures:

S indicates a position at which a single block is executed once;

SS indicates a position at which a single block is executed twice;

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

L indicates the tool moves along a straight line;

C indicates the tool moves along an arc;

r indicates the cutter compensation value;

O indicates the tool center.

3.5.3.2 Setting the Cutter Compensation

(1)Tool movement around an inner side（180° ≤ α）

Linear-> Linear

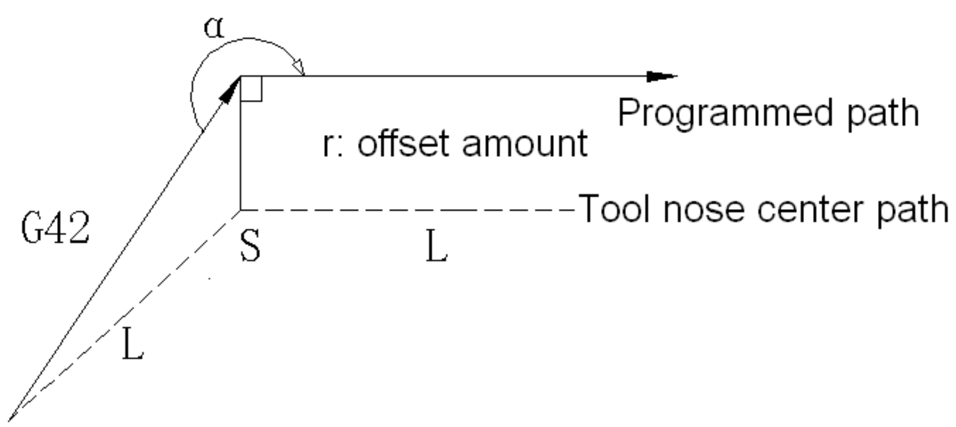


Fig. 3-5-8

Linear->Circular

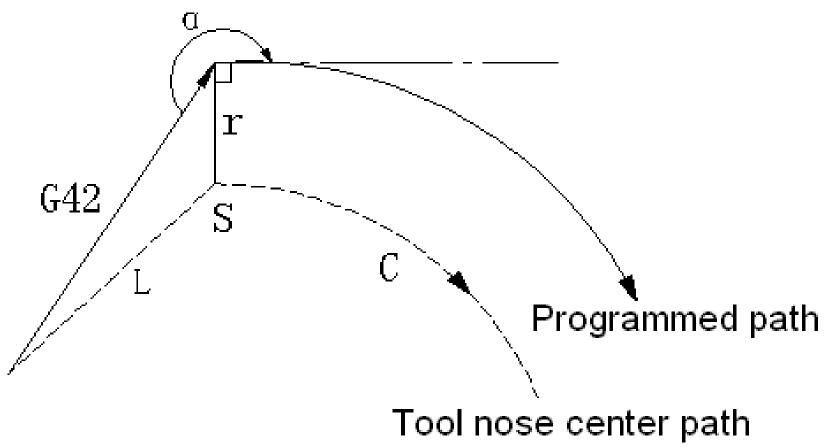


Fig. 3-5-9

(2) Tool movement around the outer side at an obtuse angle（90° ≤ α ≤ 180° ）

The tool path has two modes A and B, which are set by parameter 2600#0（SUP）.

Type A: Linear-> Linear

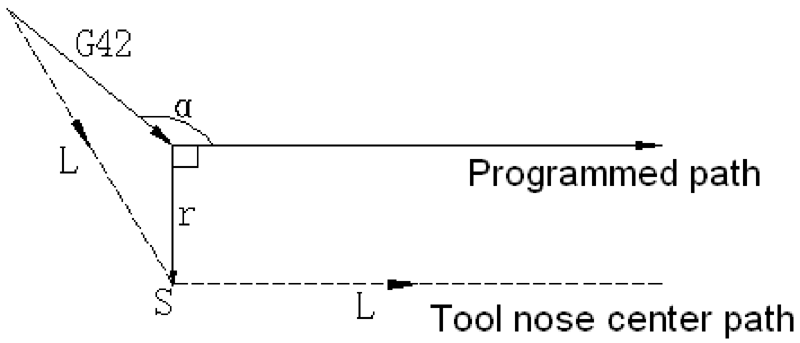


Fig. 3-5-10

Linear->Circular

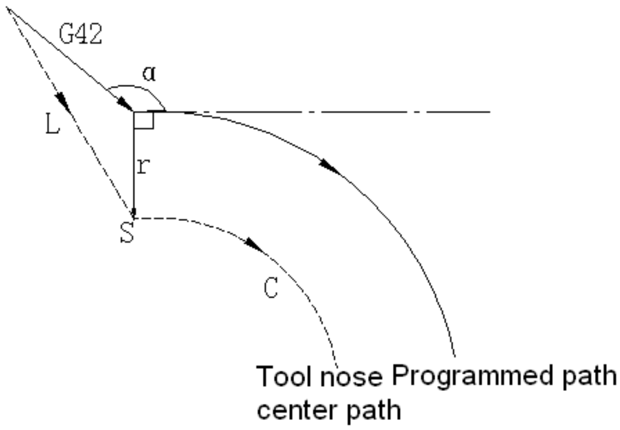


Fig. 3-5-11

Type B: Linear->Linear

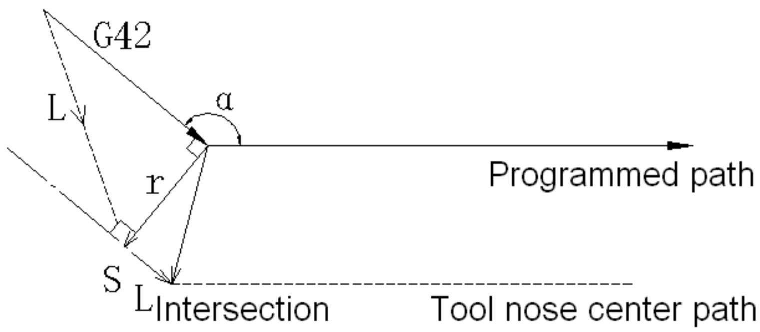


Fig. 3-5-12

Linear->Circular

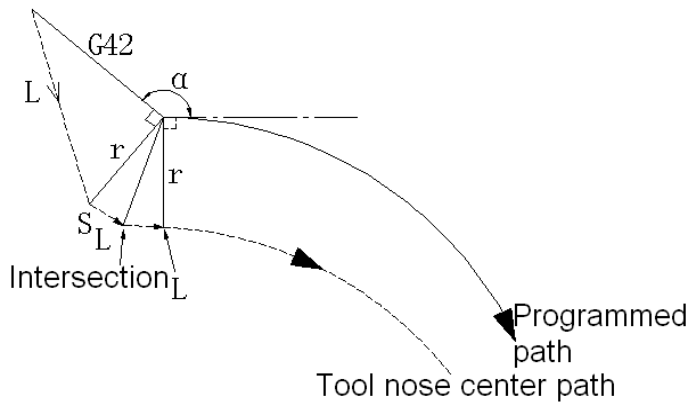


Fig. 3-5-13

(3) Tool movement around the outside of an acute angle（α＜ 90°）

The tool path has two modes A and B, which are set by parameter 2600#0（SUP）.

Type A: Linear-> Linear

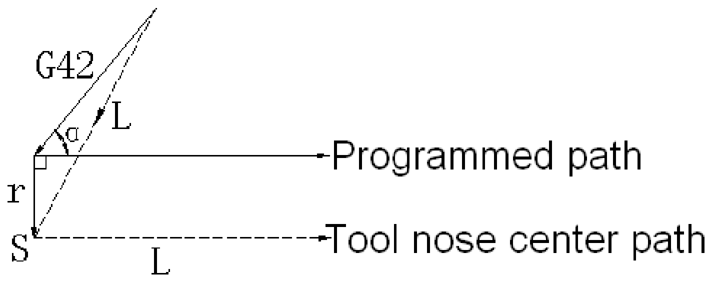


Fig. 3-5-14

Linear->Circular

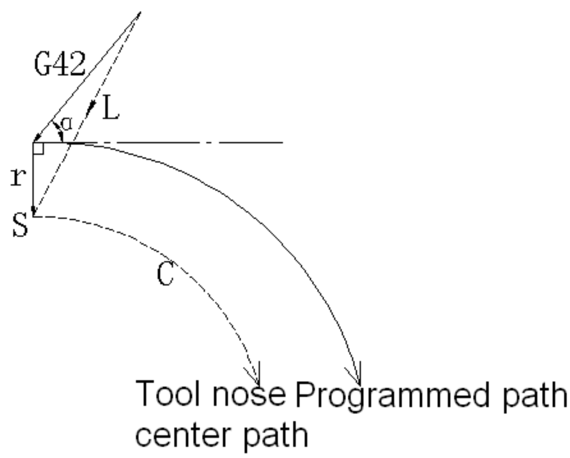


Fig. 3-5-15

Type B Linear->Linear

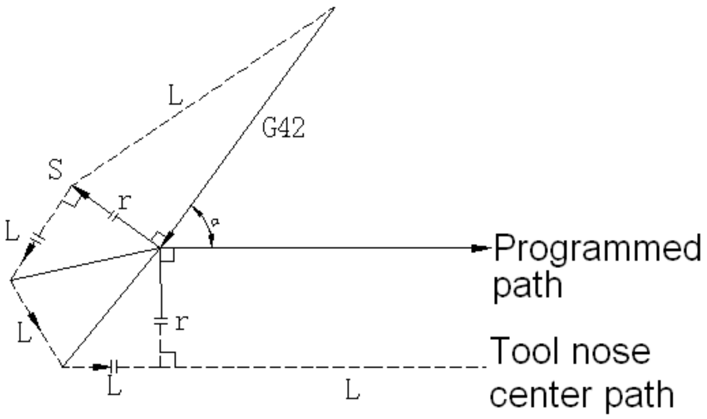


Fig. 3-5-16

Linear->Circular

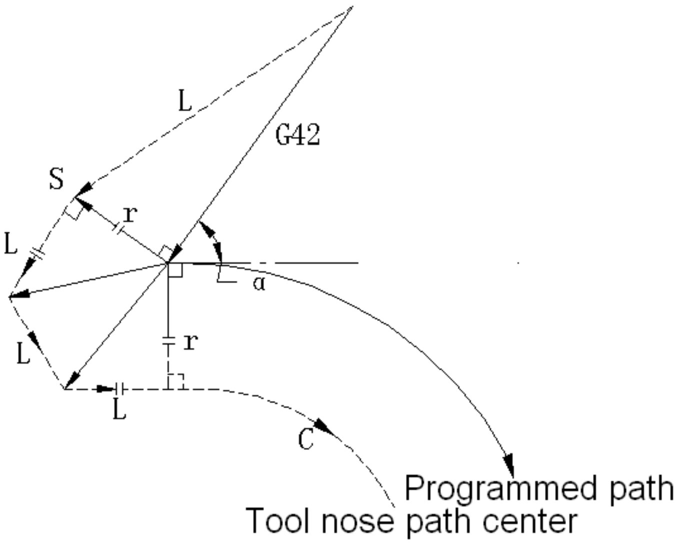


Fig. 3-5-17

(4) Tool movement around the outside at an acute angle less than 1° （α ＜ 1°）

Linear->linear

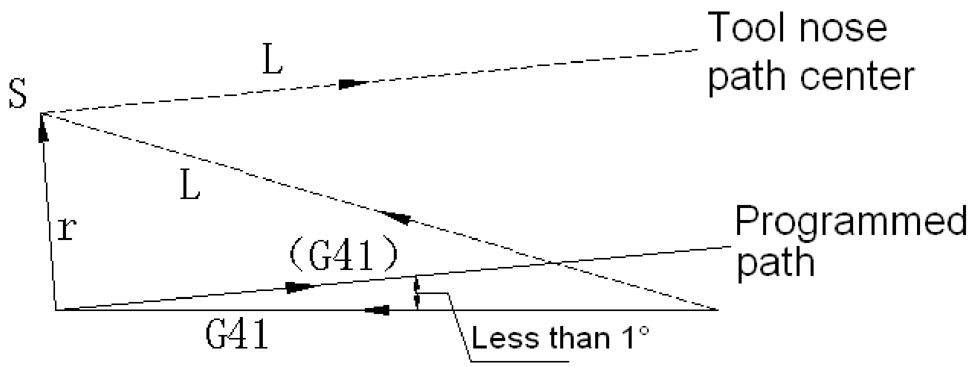
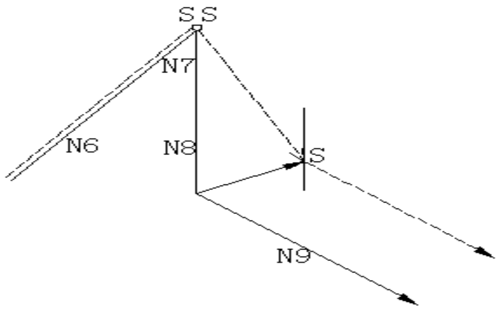


Fig. 3-5-18

(5) A block without tool movement around specified at start-up

If the command is not specified at start-up, the system sets the tool compensation in the next block.




```
G04   G91 --------------
 ⋮
N6   X100.0   Y100.0;
N7   G41 D1 X0;
N8   Y－100.0;
N9   X100.0   Y—100.0;
```

Fig. 3-5-19

3.5.3.3 Tool movement in offset mode

(1) Tool movement around the inside of a corner （180° ≤ α）

Linear->linear

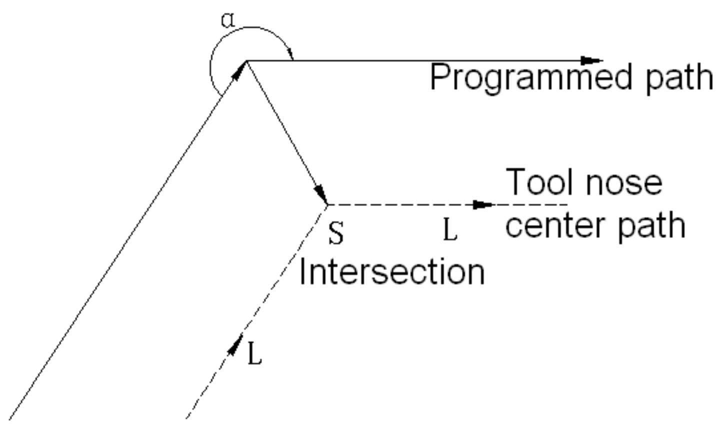


Fig. 3-5-20

Linear->arc

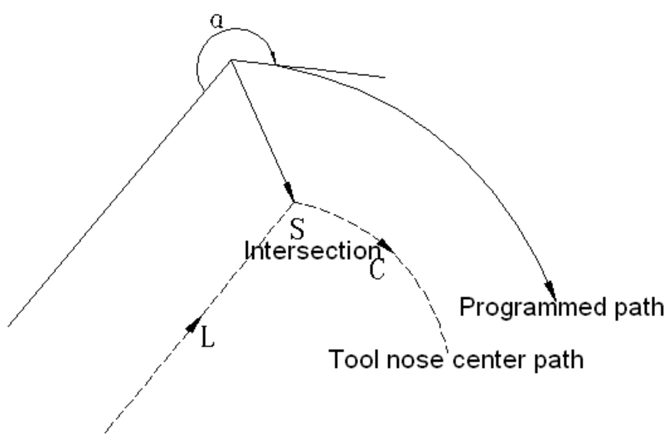


Fig. 3-5-21

Arc->linear

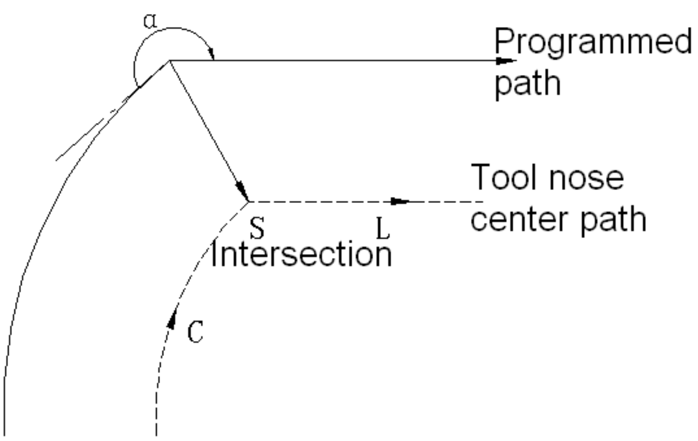


Fig. 3-5-22

Arc->arc

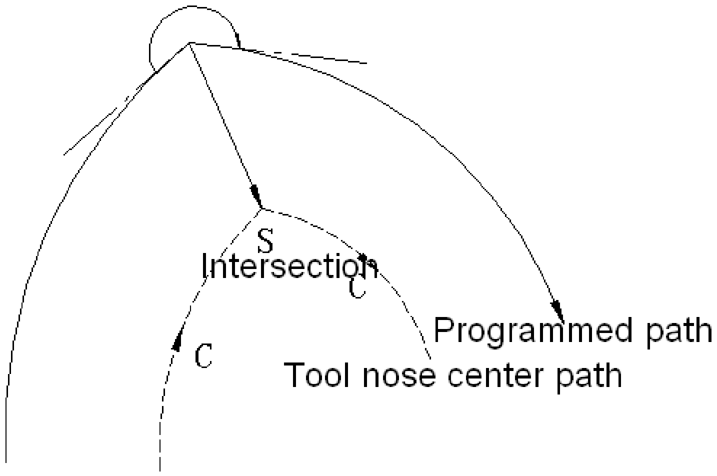


Fig. 3-5-23

Tool movement around the inside with an abnormally long vector （α＜1°）

Linear->linear

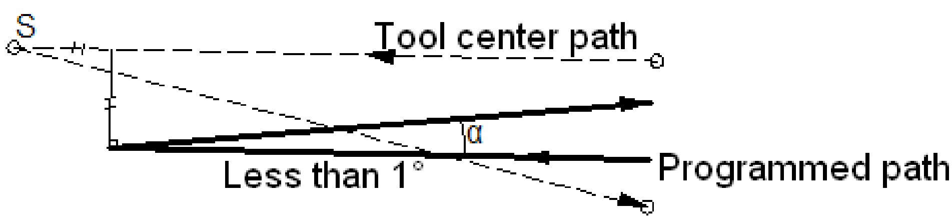


Fig. 3-5-24

Also in case of arc to straight line, straight line to arc and arc to arc, the operator should infer the same procedure.

（2) Tool movement around the outside corner at an obtuse angle（90° ≤α <180°)

Linear->linear

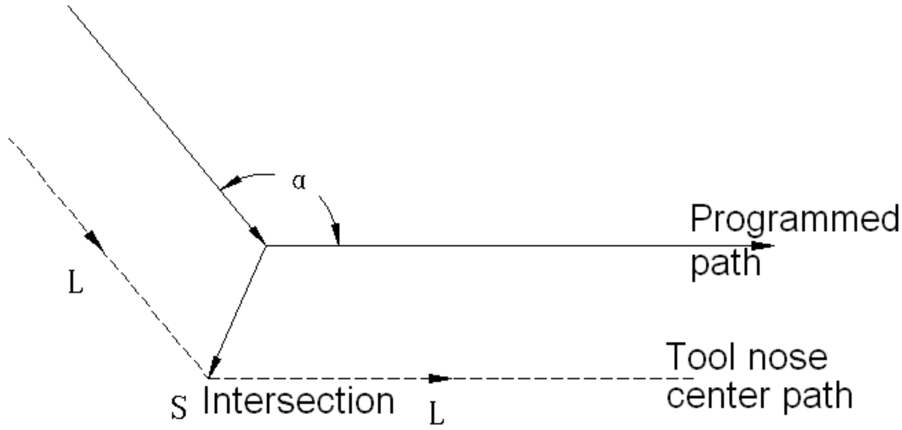


Fig. 3-5-25

Linear→arc

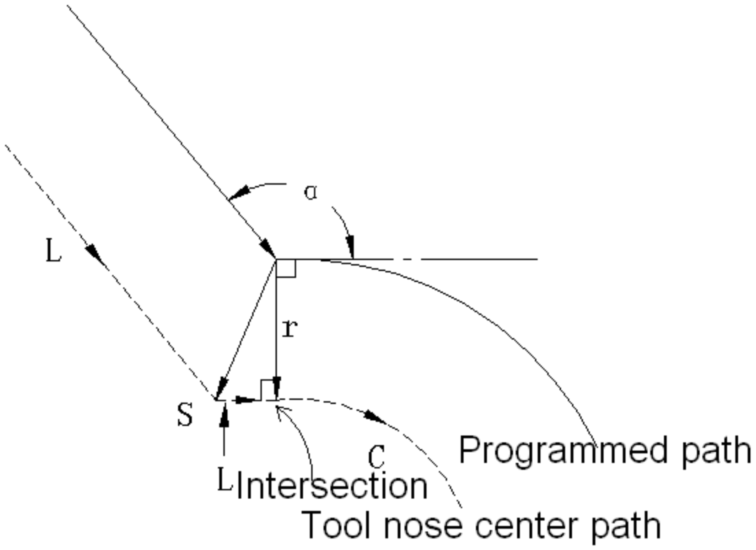


Fig. 3-5-26

Arc->linear

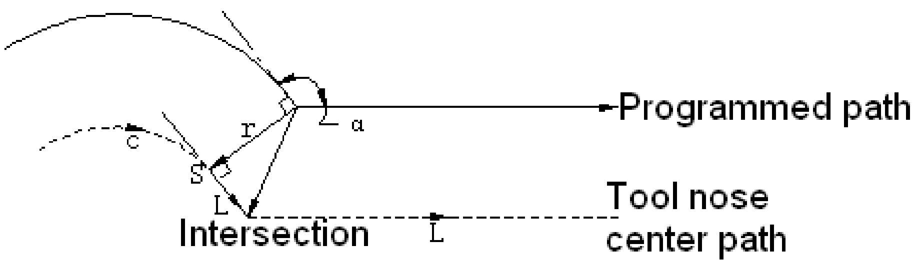


Fig. 3-5-27

Arc->arc

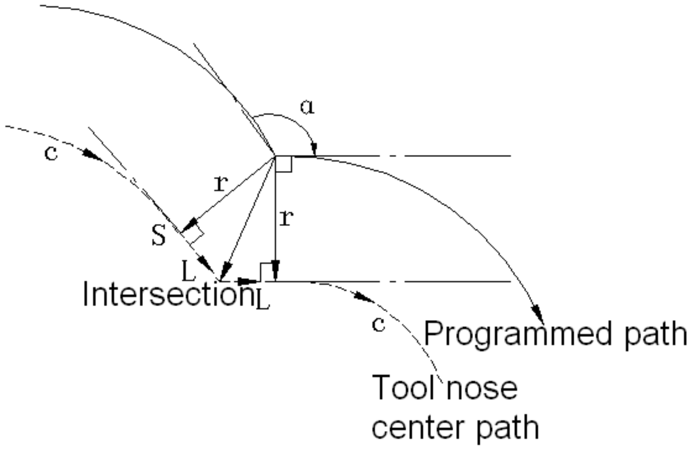


Fig. 3-5-28

(3) Tool movement around the outside corner at an acute angle（α ＜ 90°）

Linear→linear

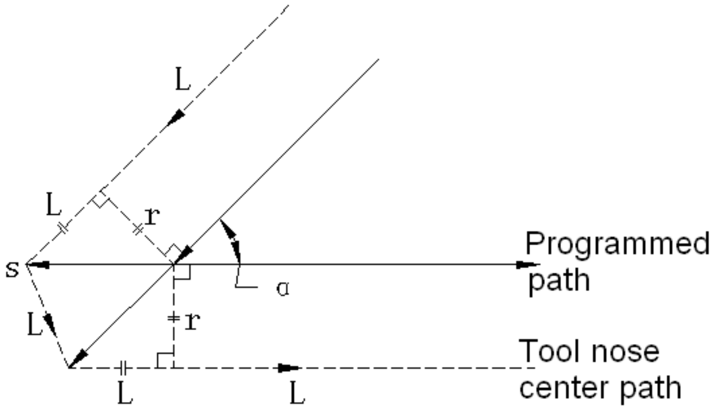


Fig. 3-5-29

Linear→arc

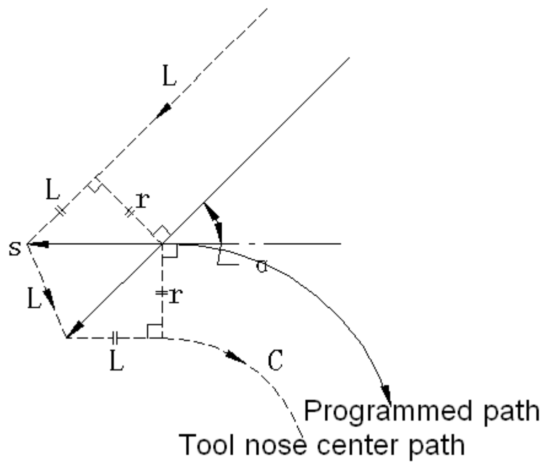


Fig. 3-5-30

Arc→linear

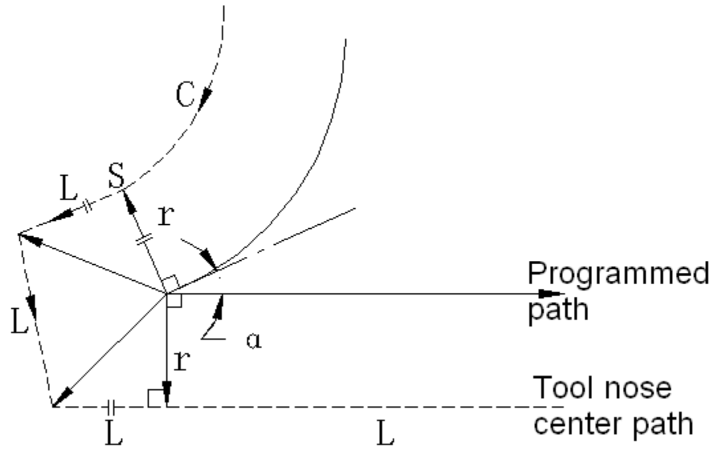


Fig. 3-5-31

Arc→arc

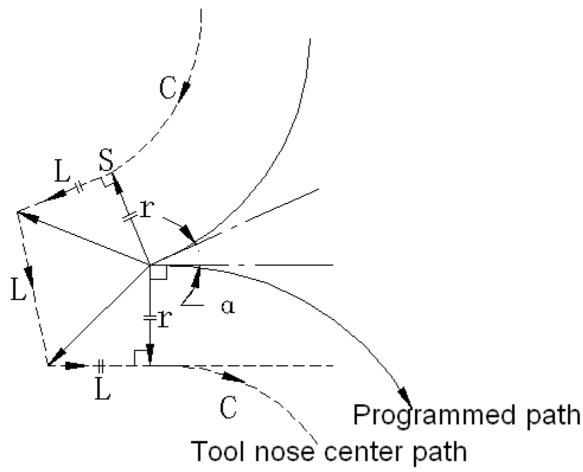


Fig. 3-5-32

Note: When it is exceptional:
End position for the arc is not on the arc

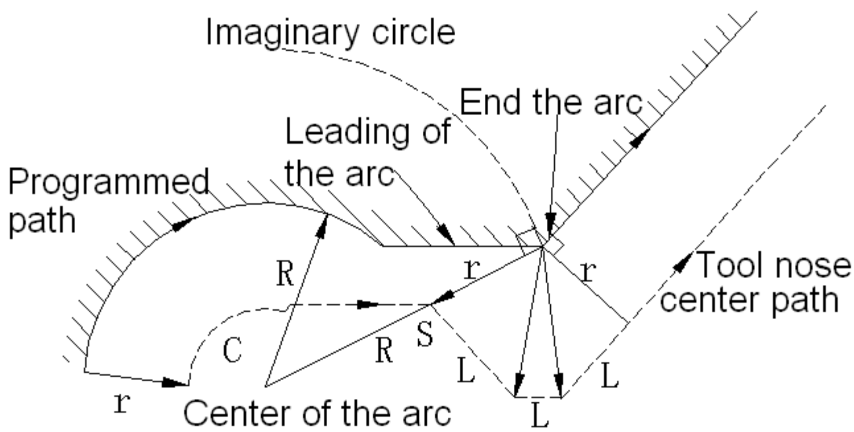


Fig. 3-5-33

It also applies to the situation of arc→arc.

There is no inner intersection

If the cutter compensation value is sufficiently small, the two circular center paths made after compensation intersect at a position (P). Intersection P may not occur if an excessively large value is specified for cutter compensation. When this is predicted, alarm No. 33 occurs at the end of the previous block and the operation is stopped.

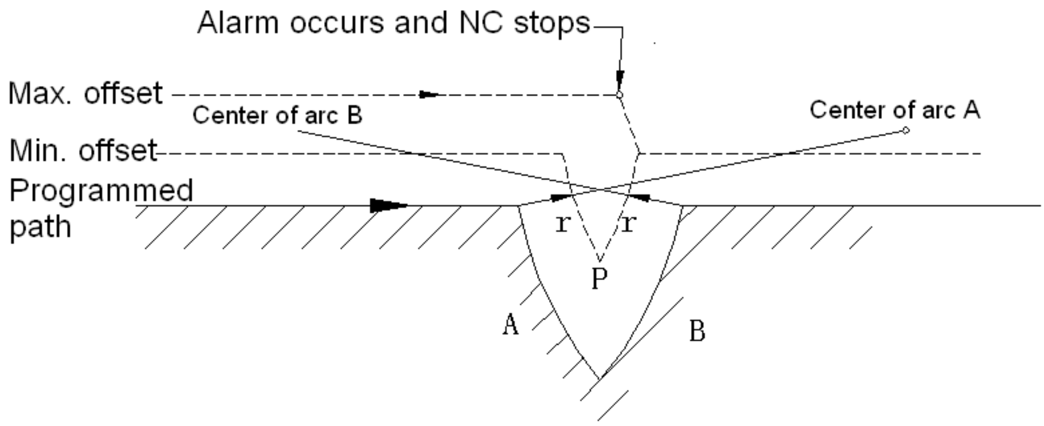


Fig. 3-5-34

The center of the arc is identical with the start position or the end position

If the center of the arc is identical with the start position or the end position, the alarm is displayed.

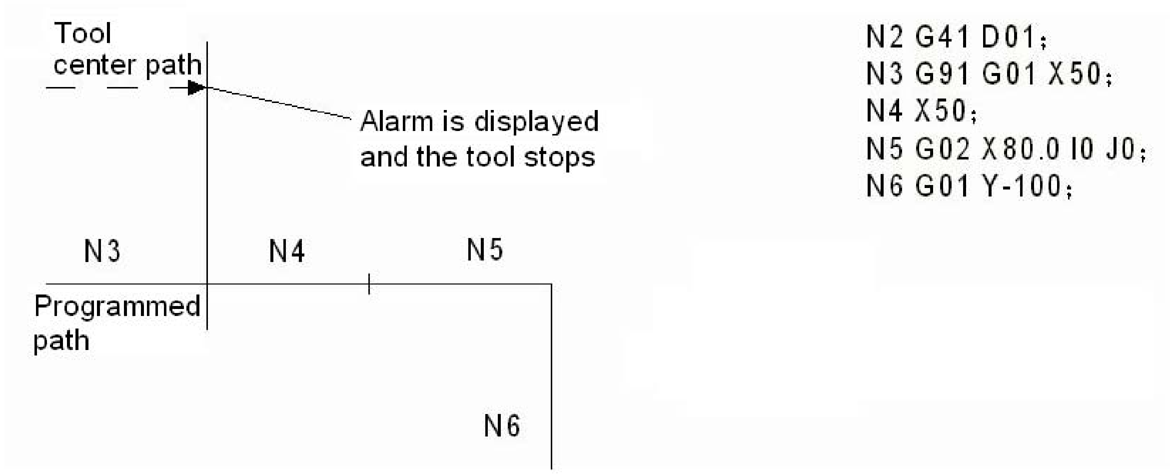


Fig. 3-5-35

3.5.3.4 Tool movement in offset mode cancel

(1) Tool movement around an inside corner（α≥180°）

Linear→linear

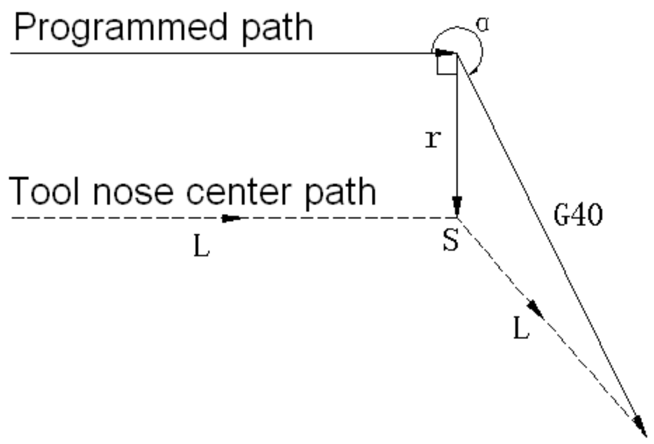


Fig. 3-5-36

Arc→linear

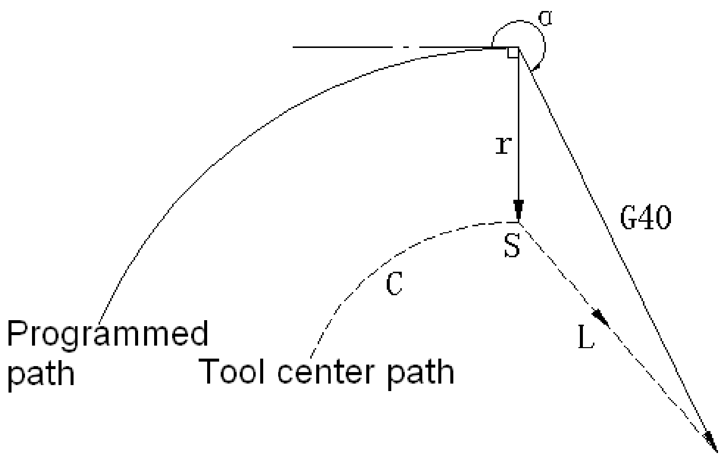


Fig. 3-5-37

(2) Tool movement around an outside corner at an obtuse angle （90°≤α<180°）

Tool path has two types, A and B; and they are selected by parameter 2600#0（SUP）.

Type A Linear→linear

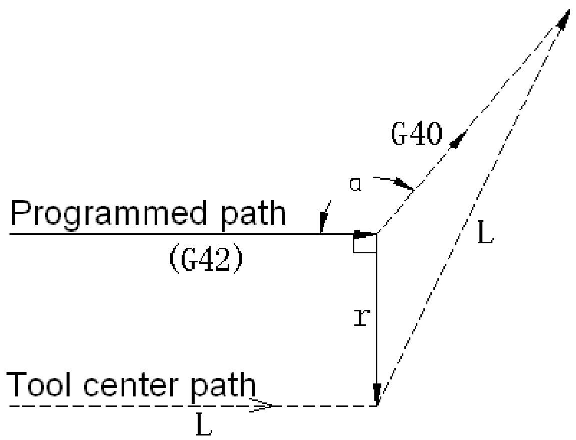


Fig. 3-5-38

Arc→linear

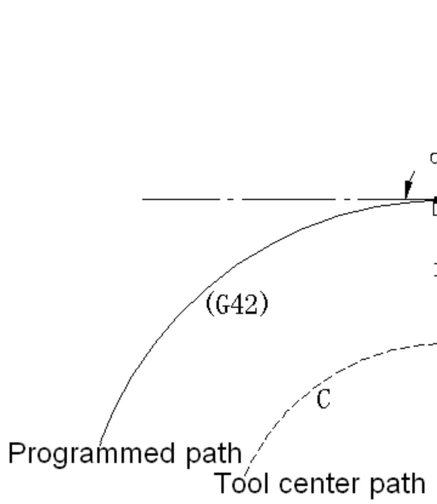


Fig. 3-5-39

Type B Linear→linear

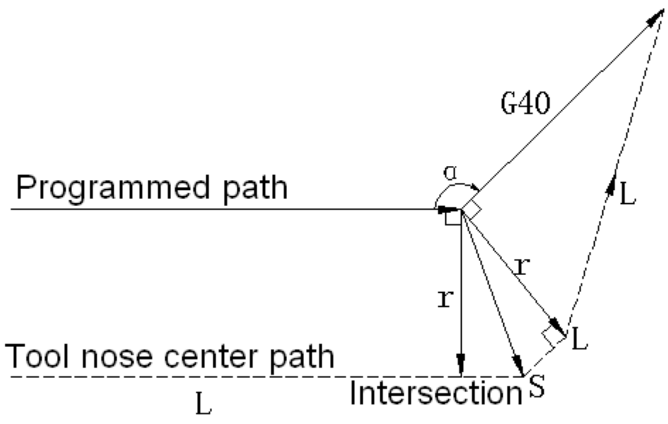


Fig. 3-5-40

Arc→linear

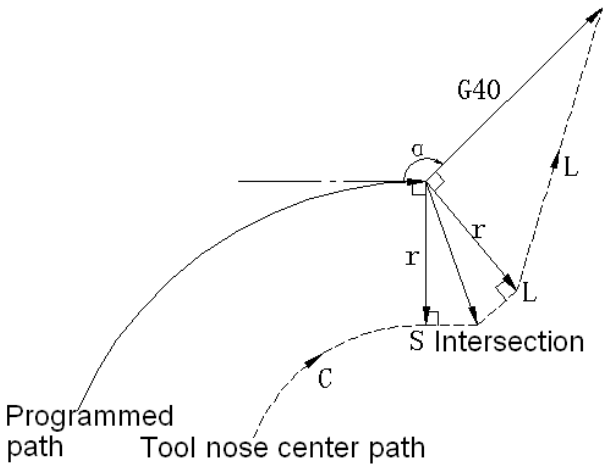


Fig. 3-5-41

(3) Tool movement around an outside corner at an acute angle （α ＜ 90° ）

Tool path has two types, A and B; and they are selected by parameter 2600#0（SUP）.

Type A Linear→linear

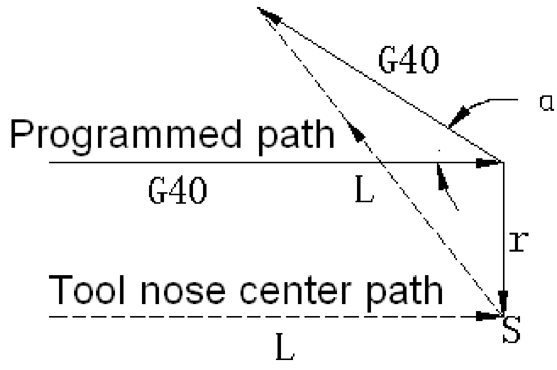


Fig. 3-5-42

Arc→linear

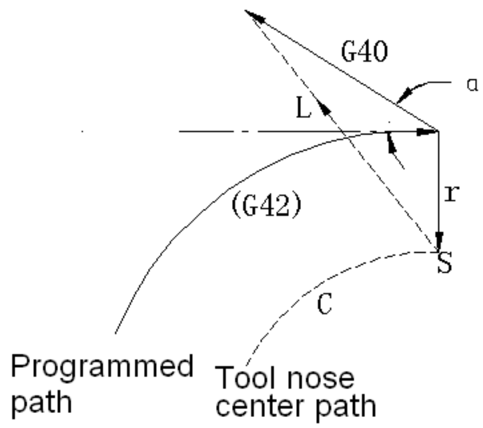


Fig. 3-5-43

Type B Linear→linear

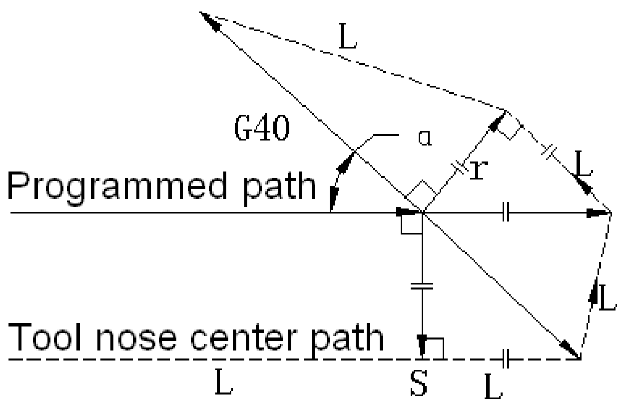


Fig. 3-5-44

Arc→linear

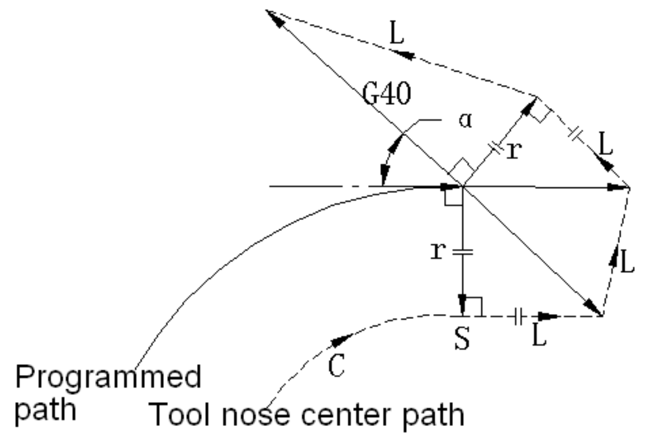


Fig. 3-5-45

Tool movement around the outside at an acute angle less than 1°, linear→linear:

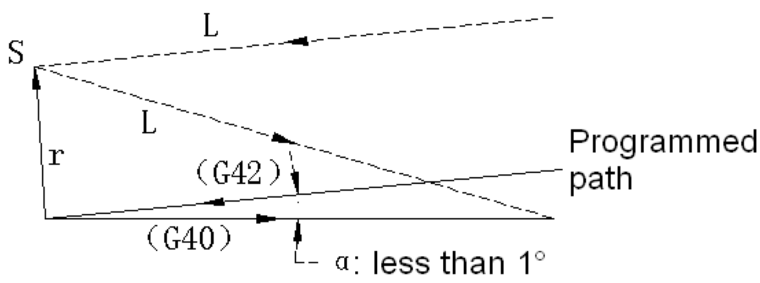
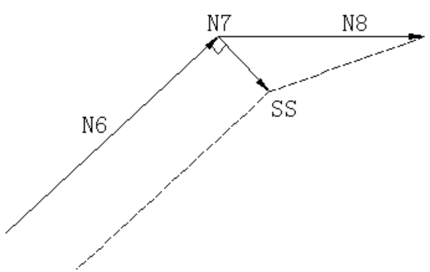


Fig. 3-5-46

(4) A block without tool movement specified together with offset cancel

When a block without tool movement is commanded together with an offset cancel, a vector whose length is equal to the offset value can be set to form a right angle to the moving direction in the previous block; the vector is cancelled in the next movement command.



N6 G91 X100.0 Y100.0;
N7 G40;
N8 X50.0;

Fig. 3-5-47

Change in the offset direction in the offset mode

The offset direction is decided by G codes (G41 and G42) for cutter radius and the sign of cutter compensation value as follows.

Offset amount sign / G code	+	—
G41	Left side offset	Right side offset
G42	Right side offset	Left side offset

The offset direction can be changed in the offset mode with G41 and G42. However, the change is not available in the start-up block and the block following it. The offset direction is changed, irrespective inner or outer side, and the offset amount is positive in the following example.

(1) Tool center path with an intersection

Linear→linear

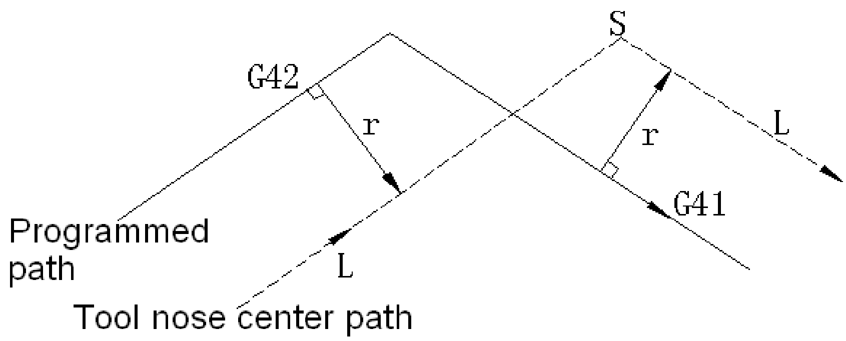


Fig. 3-5-48

Linear→arc

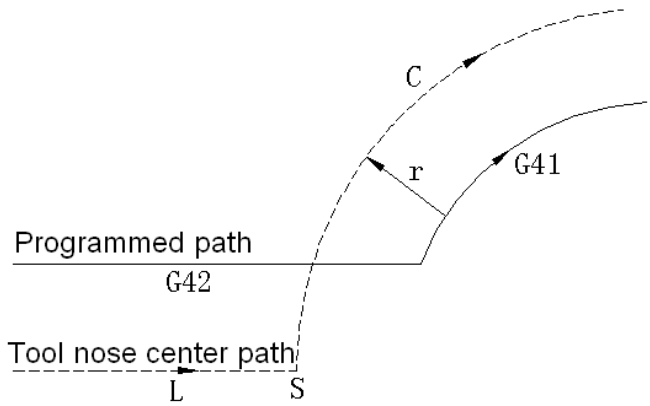


Fig. 3-5-49

Arc→linear

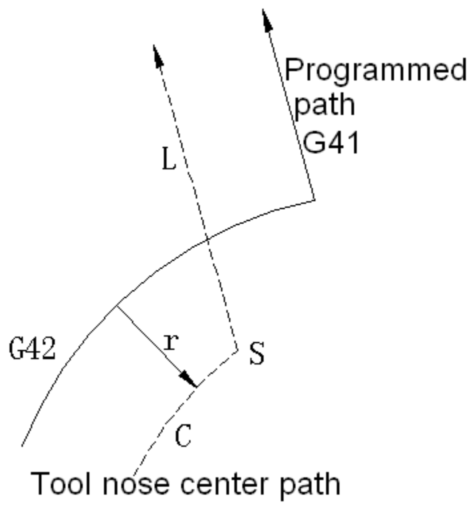


Fig. 3-5-50

Arc→arc

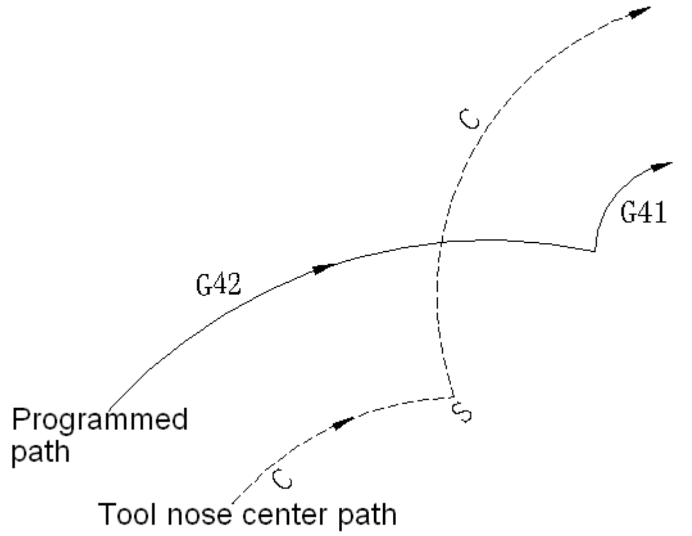
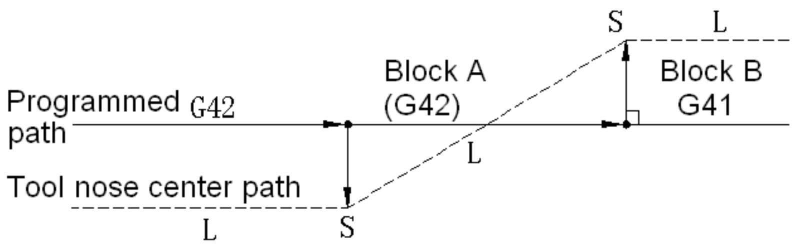


Fig. 3-5-51

(2) Tool center path without an intersection

Linear→linear



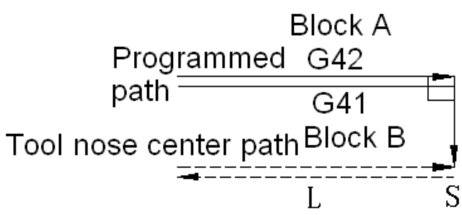


Fig. 3-5-52

Linear→arc

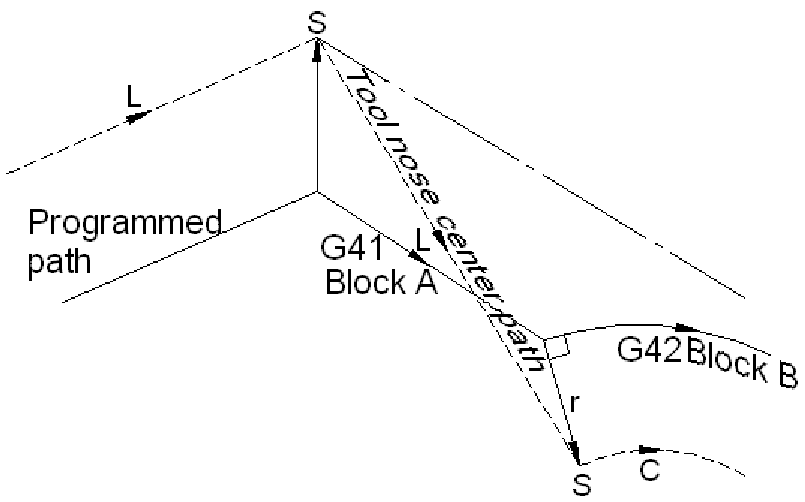


Fig. 3-5-53

Arc→arc

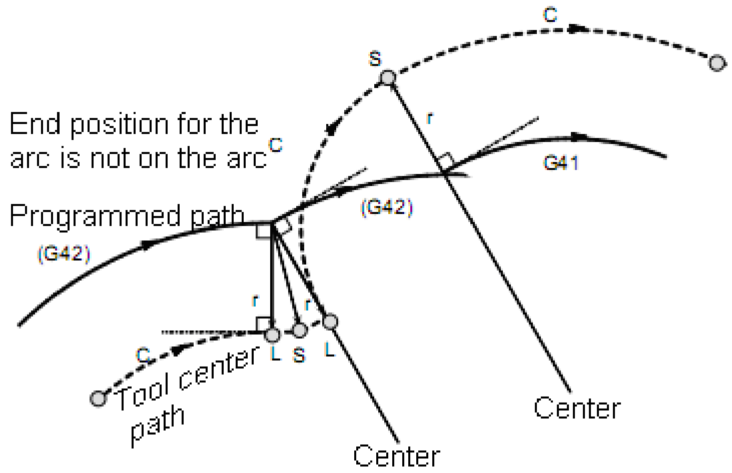


Fig. 3-5-54

In the situations of linear→arc and arc→arc, the system executes the above path with an intersection; While the system alarms if there isn't any intersection.

(3) The length of tool center path larger than the circumference of a circle:

Normally there is almost no possibility of generating this situation. However, when G41 and G42 are changed, or when a G40 was commanded with address I，J and K, this situation can occur.

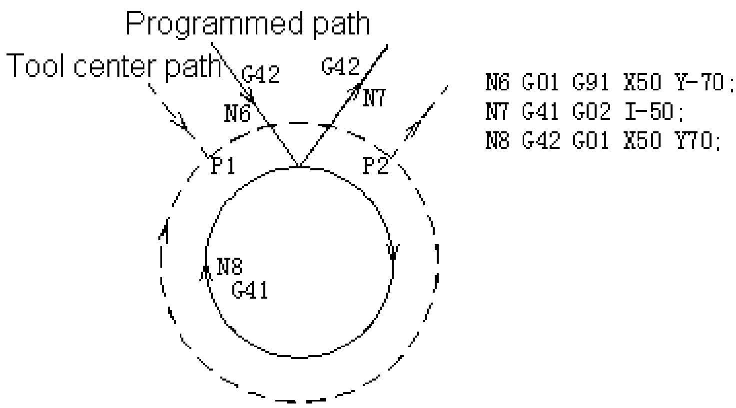


Fig. 3-5-55

In this case of the figure, the cutter compensation is not performed with more than one circle circumference: an arc is formed from P1～P2 as shown. Depending on the circumstances, an alarm may be displayed due to the "Interference Check" described later. To execute a circle with more than one circumference, the circle must be specified in segments.

3.5.3.5 Temporary cutter compensation cancel

If the following command is specified in the offset mode, the offset mode is temporarily cancelled then automatically restored.

(1) Specifying G28 (automatic return to the reference position) in the offset mode

If G28 is specified in the offset mode, the offset mode is cancelled at the intermediate position; after the reference position return, it will be automatically restored.

However, if the intermediate position is identical with the reference position, the machine returns to the intermediate position and remains still.

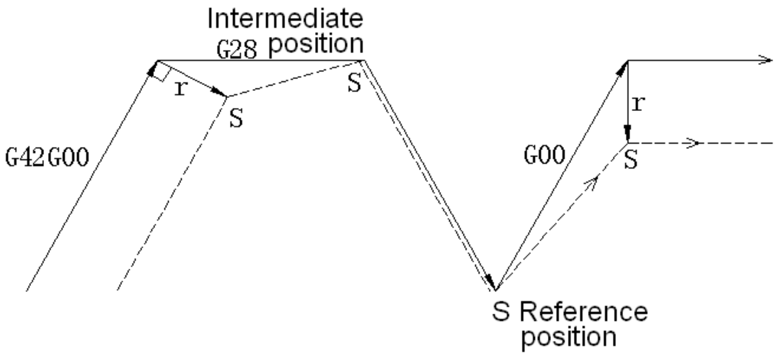


Fig. 3-5-56

(2) Specifying G28 (automatic return from the reference position) in the offset mode

If G29 is commanded in the offset mode, the offset is always cancelled at the intermediate position, and the offset mode will be restored automatically from the subsequent block.

Command G29 immediately after G28

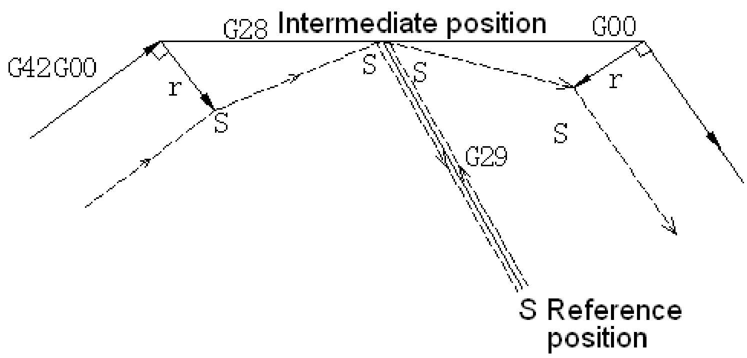


Fig. 3-5-57

Not command G29 immediately after G28

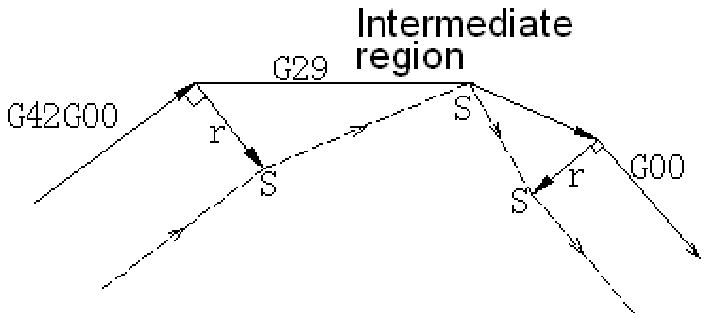


Fig. 3-5-58

Cutter compensation G code in the offset mode

The offset vector on which the tool stops can be set to form a right angle to the end position in the previous block, irrespective of machining inner or outer side, by commanding the cutter compensation G code (G41, G42) in the offset mode, independently. If this code is specified in a circular command, correct circular motion will not be obtained.

When the direction of offset is expected to be changed by command of cutter compensation G code (G41, G42), refer to subsection of *Change in the offset direction in the offset mode.*

Linear→linear

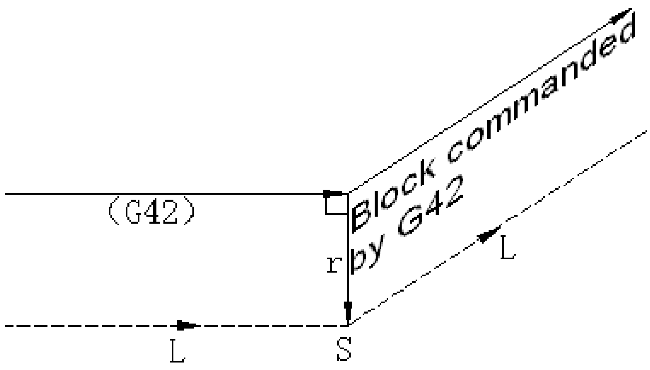


Fig. 3-5-59

Arc→linear

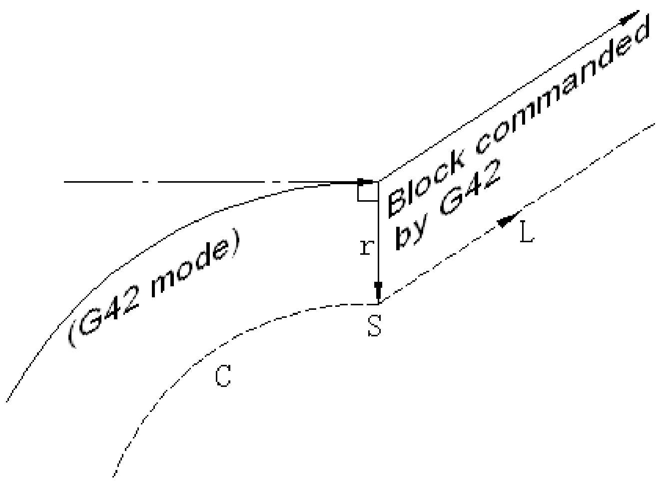


Fig. 3-5-60

G92 specified in offset mode

In offset mode, if G92 (absolute zero point programming) is specified, the machine coordinate remains still, the offset doesn't need to be cancelled, while G92 is processed as the non-movement command.

G52 procedure is same as G92.

Example:

```
G90 G54 G0 X0 Y0
G41 D1 X10 Y10
X20;
G92 X0 Y0 Z0;
X-10 Y10;
```

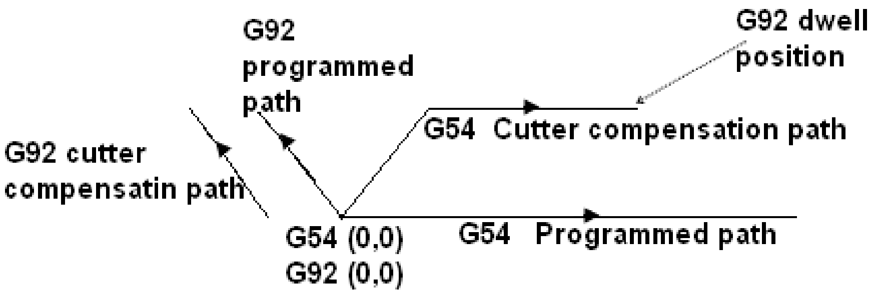


Fig. 3-5-61

Canned cycle command specified in offset mode

In offset mode, if the canned cycle command is specified, the cutter compensation is cancelled. The cutter compensation is not restored until G80 cancels the canned cycle.

A block without tool movement

The following blocks have no tool movement. In these blocks, the tool will not move even if cutter compensation is effected.

M05; ························M code output
S21; ························S code output
G04 X1000; ························Dwell time
G22 X100000; ························Setting machining area
G10 L2 P01 X100; ························Setting offset value
(G17) Z2000; ···Move command not included in the offset plane
G90; ························G code only
G91 X0; ························Move distance is zero

$\Big\}$ Not move

(3) A block without tool movement specified in offset mode

When a single block without tool movement is commanded in the tool offset mode, the vector and tool center path are the same as those when the block is not commanded. The block is executed at the single block stop point.

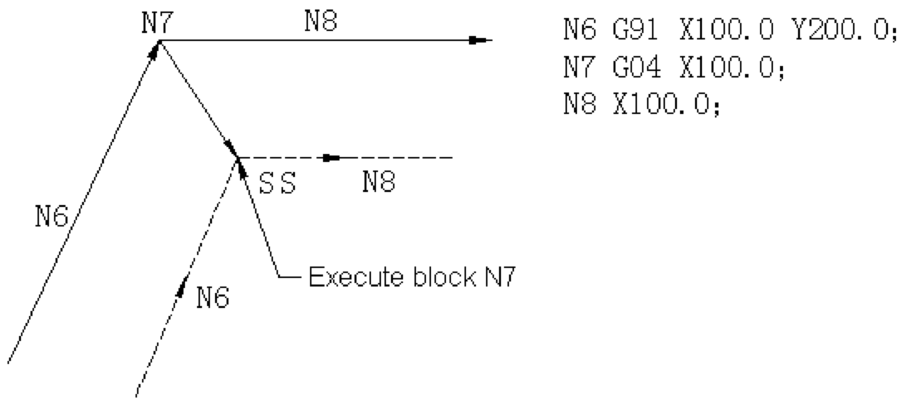


Fig. 3-5-62

However, when the move distance is zero, even if the block is commanded singly, tool motion becomes the same as that when more than one block of without tool movement are commanded, which will be described as follow:

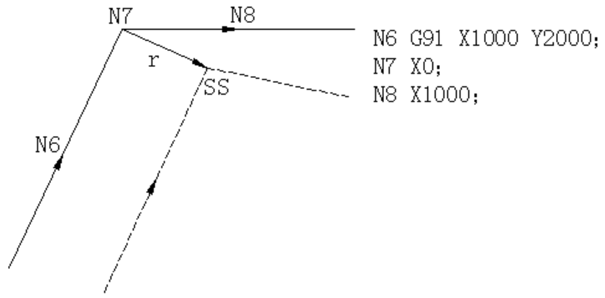


Fig. 3-5-63

When two blocks without tool movement are commanded consecutively, (Note: In the system, maximum eight blocks without tool movement can be commanded consecutively; if it exceeds eight, the alarm No. 054 is displayed.), the vector and tool center path are the same as those when these blocks are commanded. These blocks are executed at the single block stop point.

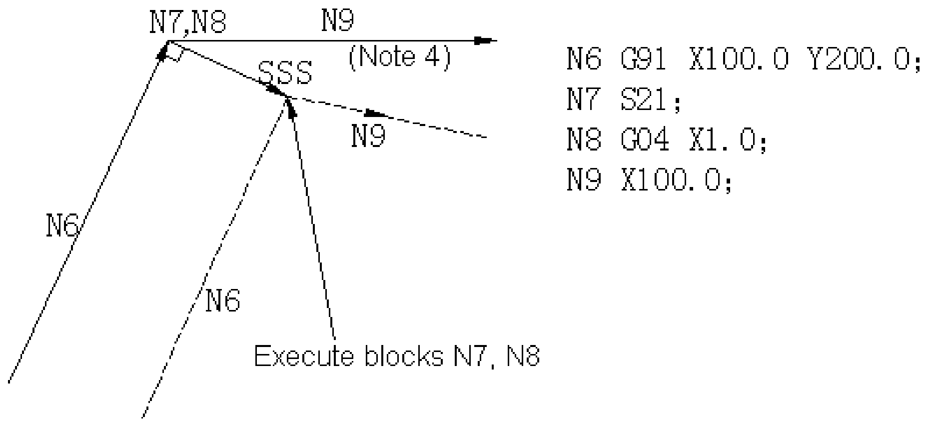


Fig. 3-5-64

(4) G41 or G42 mode before the block with G40 and I_J_K_

If G41 or G42 mode before the block with G40 and I_J_K_ is specified, the system assumes that the direction from the end position of the previous block to the vector（I,J）（I,K）or（J,K）is specified, and the offset direction is same as that of the previous block.

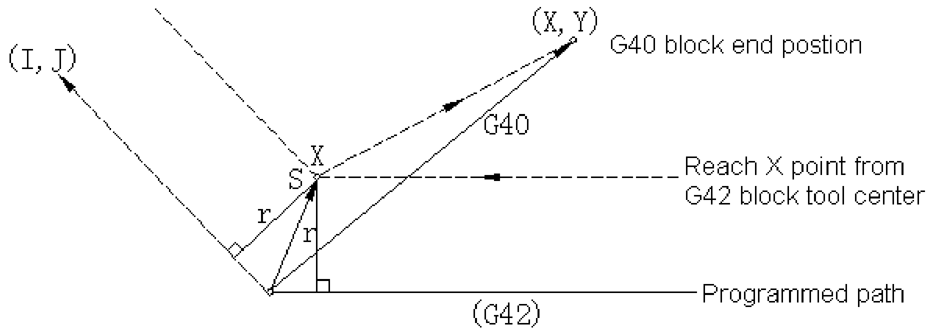


Fig. 3-5-65

(G42 mode)

G40 X__Y __I_ J_
In this situation, CNC can obtain an intersection, irrespective of machining inner or outer side.

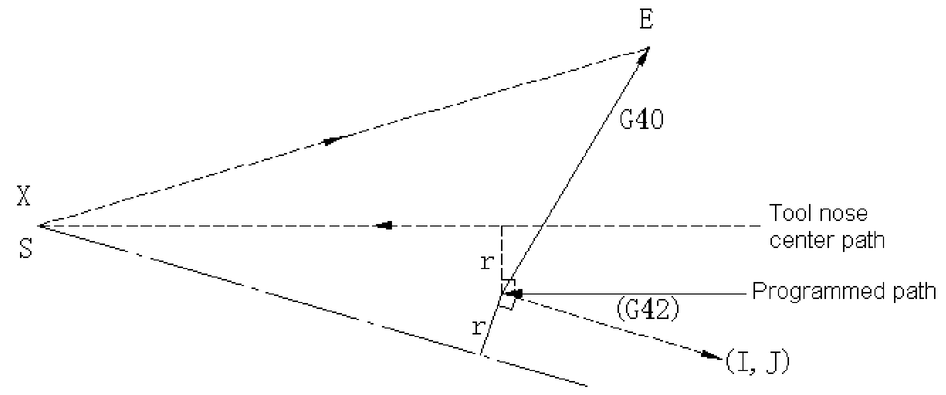


Fig. 3-5-66

If CNC can not obtain an intersection, the tool moves to the end position of the block before G40 and orthogonality is executed.

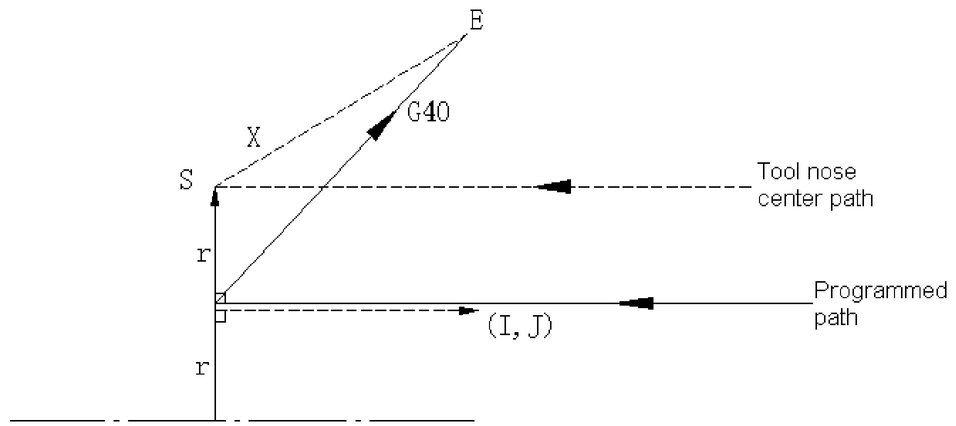


Fig. 3-5-67

A circle with more than one circumference:

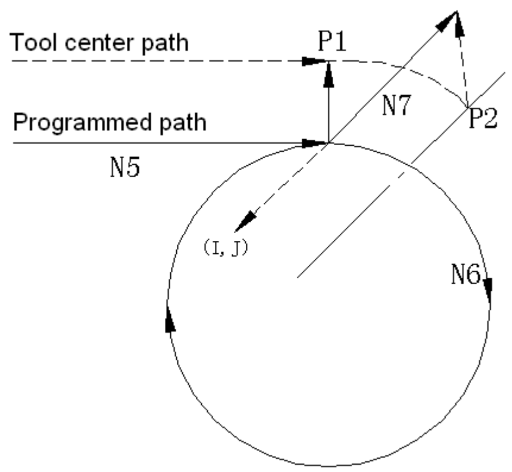


Fig. 3-5-68

（G41）
N5 G01 G91 X80；
N6 G02 J－88；
N7 G40 G01 X35 Y35 ；

In this case of the figure, the tool center path doesn't't move with more than one circle circumference, an arc is formed $P_1 \sim P_2$ as shown. The system may alarm due to the interference check. If the tool should move with more than one circle circumference, the circle must be specified in segments.

(5) Corner movement

When two or more vectors are produced at the end of a block, the tool moves linearly from one vector to the other. This movement is called the corner movement.

If these vectors almost coincide with each other, the corner movement isn't't performed and the latter vector is ignored.

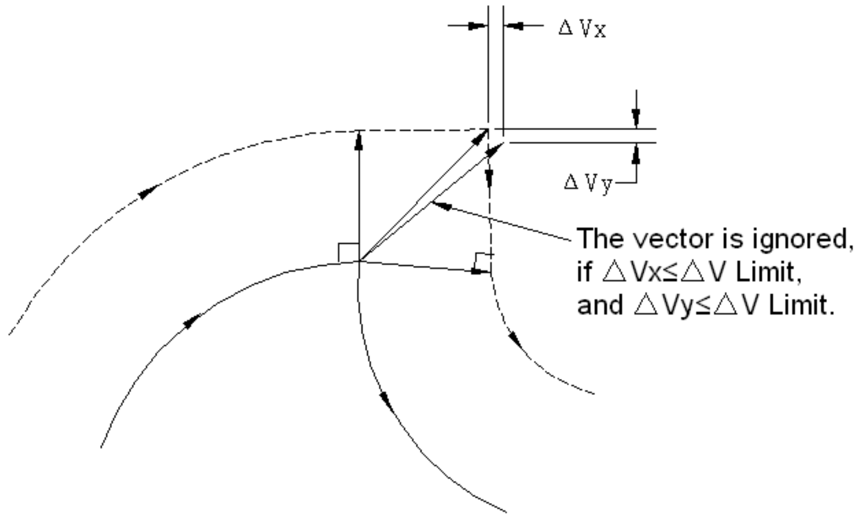


Fig. 3-5-69

If $\triangle V_X < \triangle V$ limit and $\triangle V_Y < \triangle V$ limit, the latter vector is ignored. The $\triangle V$ limit is set in advance by parameter 2610. If these vectors do not coincide, a move is generated to turn around the corner. And the corner move before the single block stop point belongs to the previous block; otherwise, it belongs to the next one.

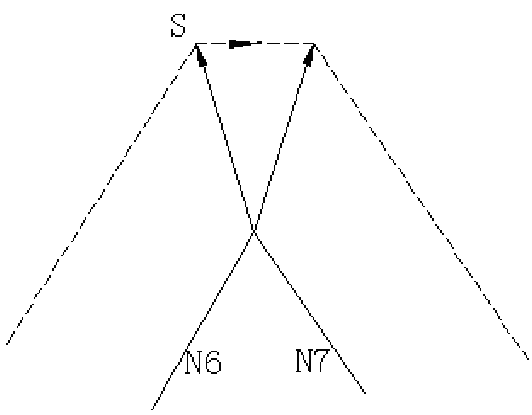


Fig. 3-5-70

However, if the path of the next block is semicircular or more, the above function is not performed. The reason for this is as follows:

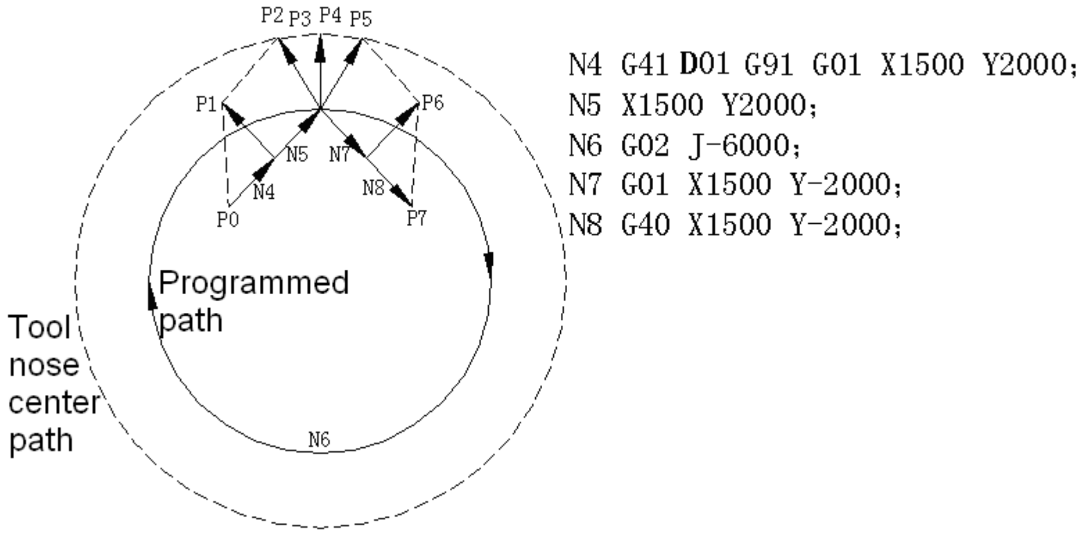


Fig. 3-5-71

If the vector is not ignored, the tool path is as follows:

$P_0 \to P_1 \to P_2 \to P_3$ (Circle) $\to P_4 \to P_5 \to P_6 \to P_7$

3.5.3.6 Preventing Over-cutting by Tool Radius Compensation

(1) Processing the inner round angle that is smaller than tool radius

When radius of the specified round angle is smaller than tool radius, system alarms for over cutting caused by tool radius inner offset. Alarm NO.41 occurs at the beginning of the block and the system stops. Tool path is as the following figure.

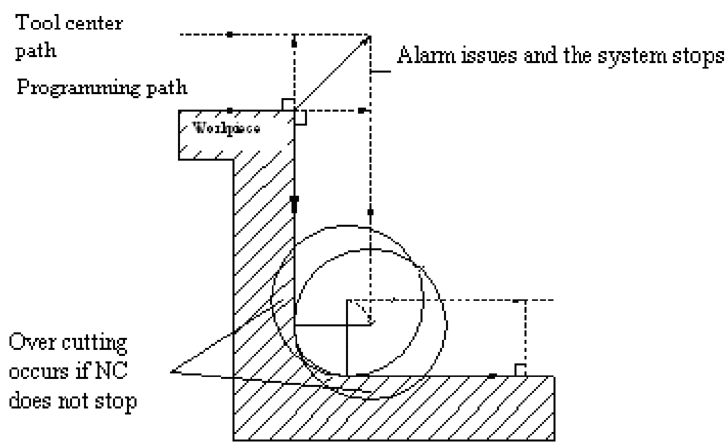


Fig. 3-5-72

(2) Processing a groove that is smaller than tool radius

Tool radius compensation makes the direction of the center path and programming path opposite, which leads over cutting. No.41 alarm issues at the beginning of the block and the system stops.

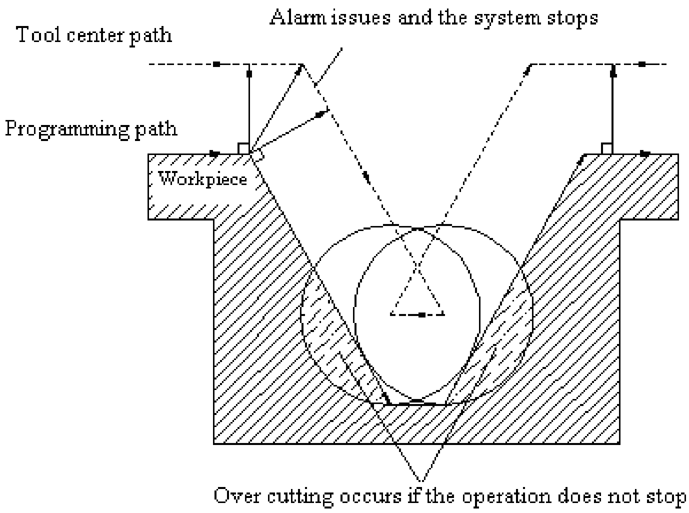


Fig. 3-5-73

（3）Processing a step that is smaller than tool radius

When there is a step in a program, tool center path of offset may opposite to programming path for the process is performed by round cutting command. At this time, the first vector is ignored and tool moves to the second vector. The program stops at the current point in single block mode, or program moves to the next if it is not in the single block mode.

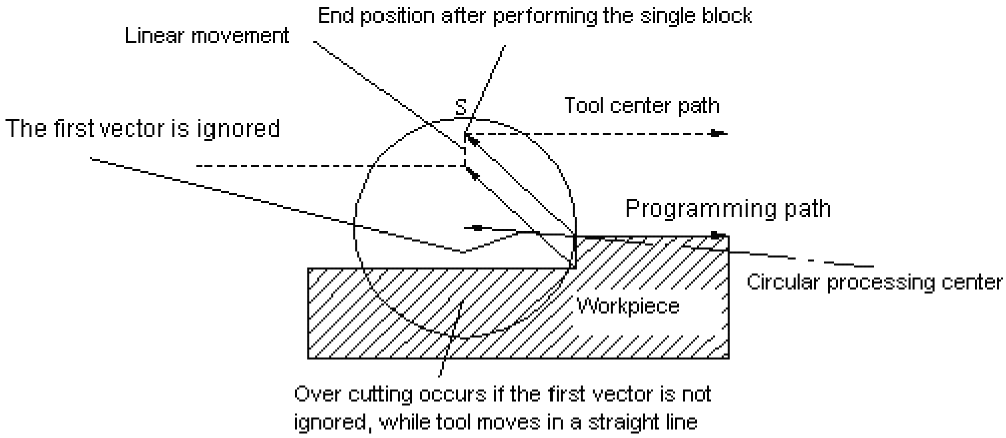


Fig. 3-5-74

（4）Starting compensation and cutting along Z axis

Before starting the cutting operation, set tool compensation previously at a certain distance to the workpiece (generally XY plane), and perform feed operation by the method of tool moves along Z axis. At this time, if Z axis rapid feed is not separate from cutting feed, please pay attention to the following items:

Refer to the following programs:

N1 G91 G00 G41 X500 Y500 D1;
N3 G01 Z-300 F100;
N6 Y100 F200;

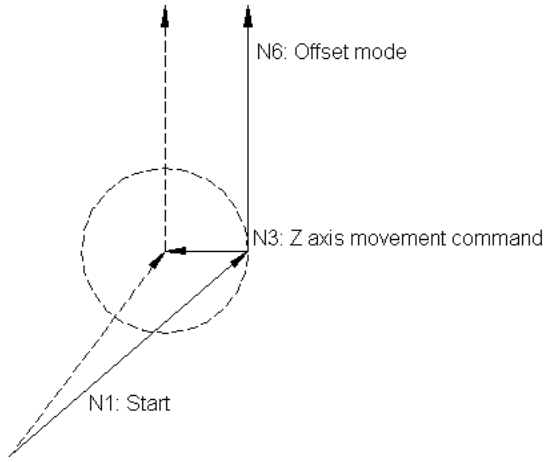


Fig. 3-5-75

In the above example, N3 and N6 are read in buffer register when block N1 is being executed. See the figure below for correct compensation.

Second, if N3 (Z axis movement command) is separated:

N1 G91 G00 G41 X500 Y500 D1;
N3 Z-250;
N5 G01 Z-50 F100;
N6 Y1000 F200;

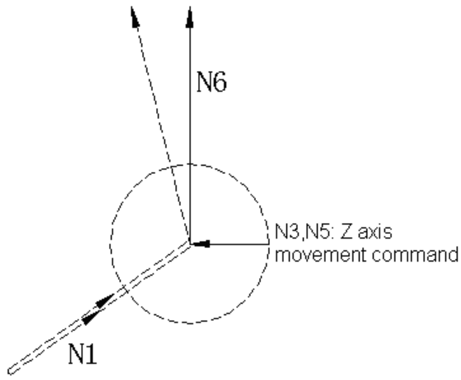


Fig. 3-5-76

Because N3 and N5 blocks are not included in XY plane, N6 can not enter into buffer register when N1 is performing. Tool center path is calculated by information of N1 on the right figure. Tool vector can not be formed, and over cutting occurs as shown on the right figure.

On this occasion, based on the above rule, specify commands with same movement direction in blocks before/behind Z axis feed axis to prevent over cutting.

N1 G91 G00 G41 X500 Y400 D1;
N2 Y100;
N3 Z-250;
N5 G01 Z-50 F100;
N6 Y1000 F200;

Directions of N2 and N6 commands are the same.

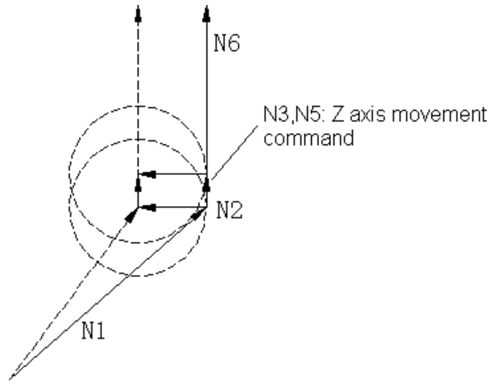


Fig. 3-5-77

Load N2 and N3 to the buffer register when performing N1 program, and perform compensation operation based on the relationship between N1 and N2.

3.5.3.7 Tool Compensation Interference Check

Tool over cutting is called “interference”. Interference check function is used for checking tool over cutting beforehand. However, this function can not check all interferences, namely, interference check is necessary though over cutting does not occur.

1） Judgment conditions for interference check

Condition 1: tool path direction is different from programming path direction, path angle is within 90°～270°.

Condition 2: Except for condition 1, the angle between starting and end point of tool center path is totally different from programming path, and it is bigger than 180°.

Example of condition 1:

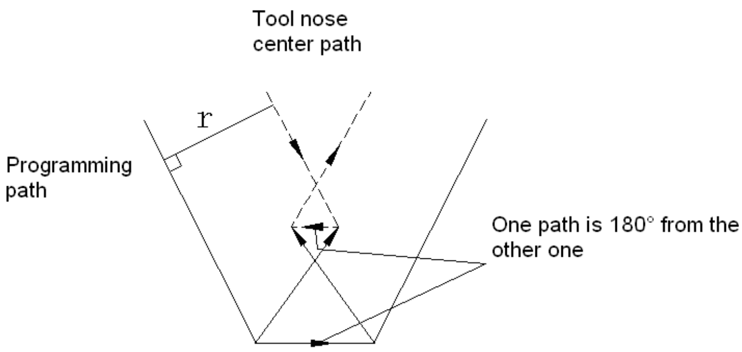


Fig. 3-5-78

Example of condition 2:

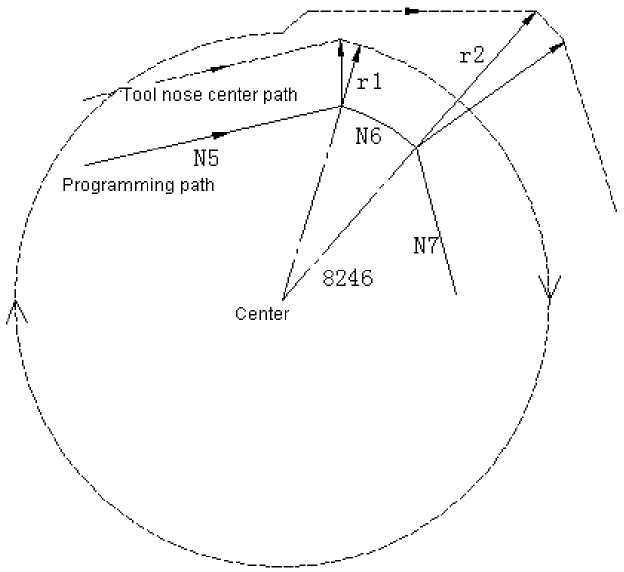


Fig. 3-5-79

（G41）
N5 G01 G91 X80 Y20 D01；
N6 G02 X32 Y-16 I-20 J-80 D02；
N7 G01 X20 Y-50；
（Corresponding offset for D01: r_1 =20）
（Corresponding offset for D02： r_2 =60）
In the above example, alarm occurs at circular N6.

2） Pre-correction for interference
The system does not provide interference correction function temporarily.

3）Correction is performed though interference does not happen. See the examples below:
（a） Concave depth is smaller than compensation value.

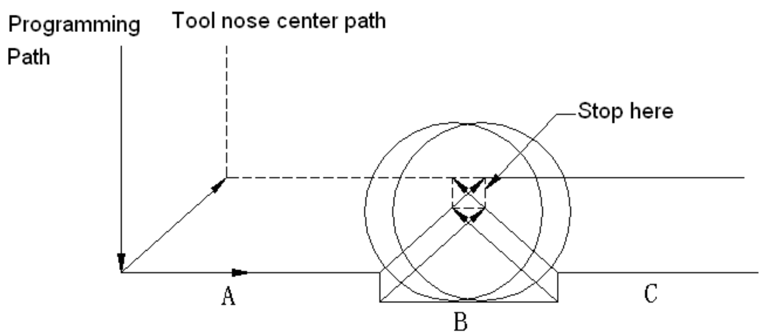


Fig. 3-5-80

Because tool path direction is different from programming path direction, tool may stops for alarm occurs though interference does not happen.

（b） Groove depth is smaller than compensation value

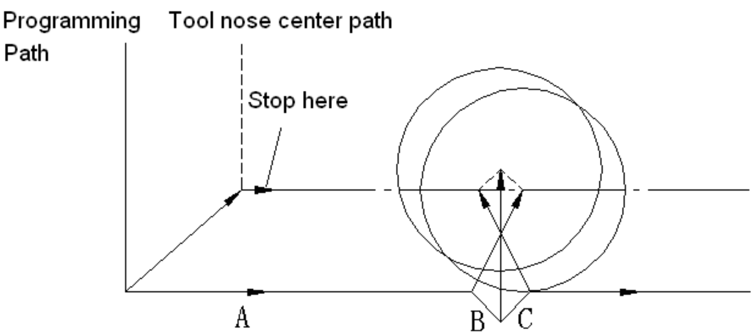


Fig. 3-5-81

It is the same as (a), tool path direction is different from programming path.

Note:
At present, the system does not support tool compensation by 4th, 5th axis.

3.5.4 The Tool Compensation Value and Number Input the Compensation Value by the 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_ ; Wearing compensation value of H code
G10 L13 P_ R_ ; Wearing compensation value of D code
P : The tool compensation number
R : The tool compensation value in the absolute value command （G90）mode
The tool compensation value in the incremental value command（G91）mode, the value adds the one of the specified tool compensation number and the sum is the tool compensation value.

Note: The valid input range of the tool compensation value：

	Metric input	Inch input
The geometric compensation	-999.9999 ㎜～999.9999 ㎜	-999.9999inch ～999.9999inch
The wearing compensation	The range is decided by parameter 2611	

Note: When parameter No.2611 is set to 100, the actual range of the wearing compensation is -100～100.

3.5.5 Automatic Tool Length Measurement (G37)

By issuing G37, the tool starts moving to the measurement position and keeps on moving until the approach end signal from the measurement device is output. Movement of the tool is stopped when the tool tip reaches the measurement position. A difference is determined between a coordinate value obtained when the tool reaches the measurement position and a coordinate value specified by G37. The difference is then added to the tool currently used length offset value.

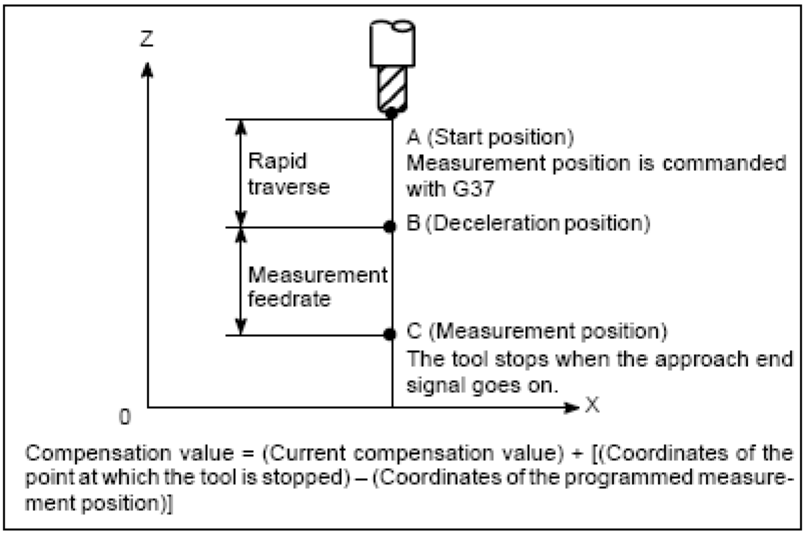


Fig. 3-5-82

Format:

G92 IP_;　Sets the workpiece coordinate system (by G54 to G59 commands).

H00;　Specifies the offset number for tool length offset.

G90 G37 IP_;　Absolute command

G37 is valid only in the block in which it is specified.

IP_ indicates the X_, Y_, Z_, or fourth axis.

Explanations:

Setting the workpiece coordinate system

Set the workpiece coordinate system so that a measurement can be made after moving the tool to the measurement position. The coordinate system must be the same as the workpiece coordinate system for programming.

Specifying G37

Specify the absolute coordinates of the correct measurement position. Execution of this command moves the tool at the rapid traverse rate toward the measurement position, reduce the feedrate halfway, then continuous to move it until the approach end signal from the measuring instrument is issued. When the tool tip reaches the measurement position, the measuring instrument sends an approach end signal to the CNC which stops the tool.

Change the offset value

The difference between the coordinates of the position at which the tool reaches for measurement and the coordinates specified by G37 is added to the current tool length offset value.

Offset value= (current compensation value)+ [（coordinate of the position at which the tool reaches for measurement）-（coordinate specified by G37）]

Alarm

When automatic tool length measurement is executed, the tool moves as shown in figure below. If the approach end signal goes on while the tool is traveling from point B to point C, an alarm occurs. Unless the approach end signal goes on before the tool reaches point F, the same alarm occurs.

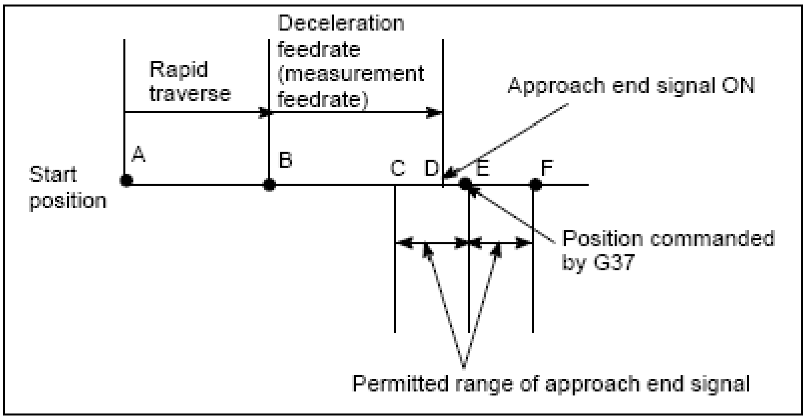


Fig. 3-5-83

Note

1. When an H code is specified in the same block as G37, an alarm is generated. Specify H code before the G37 block.
2. Inset a manual movement to measure speed in the process of movement, after insertion, restart it after returning the tool to the position before insertion.
3. The measurement speed (parameter No. 2651), deceleration position (parameter No. 2625), and permitted range of the approach end signal (parameter No. 2653) are specified by the machine tool builder.
4. Change the tool wear compensation value of H code.
   the offset value is changed when tool offset A is used.
   the tool wear compensation value is changed when tool offset B is used.
   the wear compensation value of H code is changed when tool offset C is used.
5. The approach end signal is monitored usually every 2 ms. The following measuring error is generated:
   ERR.max:Fm×1/60×Ts/1000
   Where :
   Ts::sampling period, for usual 2（ms）
   ERR max: maximum measuring error (mm)
   Fm: measurement feedrate（mm/min）
   For example, when Fm=1000 mm/min, ERR max=0.003 mm
6. The tool stops a maximum of 16 ms after the approach end signal is detected. But the value of the position at which at approach end signal was detected (note the value when the tool stopped) is used to determine the offset amount. The overrun for 16 ms is:
   Qmax= Fm×1/60×16/1000
   Qmax : maximum overrun（mm）
   Fm: measurement feedrate（mm/min）

Example:

G92 Z760.0 X1100.0; Setting a workpiece coordinate system with respect to the programmed absolute zero point.

G00 G90 X850.0; Moving the tool to X850.0. that is the tool is moved to a position that is a specified distance from the measurement position along the Z-axis.

H01; Specifying offset number 1.

G37 Z200.0; Moving the tool to the measurement position.

G00 Z204.0; Retracting the tool a small distance along the Z-axis.

For example, if the tool reaches the measurement position with Z198.0;, the compensation value must be corrected. Because the correct measurement position is at a distance of 200mm, the compensation value is lessened to 2.00mm (198.0-200.0=-2.0).

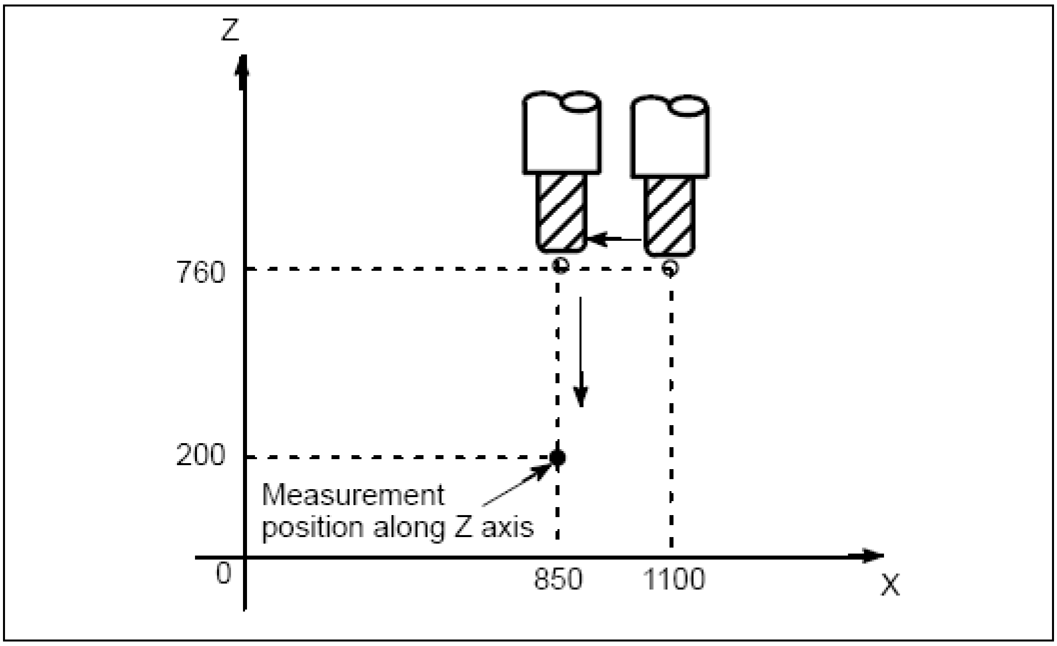


Fig. 3-5-84

3.5.6 Tool Position Offset (G45-G48)

The programmed travel distance of the tool can be increased or decreased by a specified tool offset value or by twice the offset value. The tool offset function can also be applied to an additional axis.

Format:

G45 IP_D_; increase the travel distance by the tool offset value
G46 IP_D_; decrease the travel distance by the tool offset value
G47 IP_D_; increase the travel distance by twice the tool offset value
G48 IP_D_; decrease the travel distance by twice the tool offset value

G45～G48: Non-modal G code for increasing or decreasing the travel distance
IP: Command for moving the tool
D: code for specifying the tool offset value

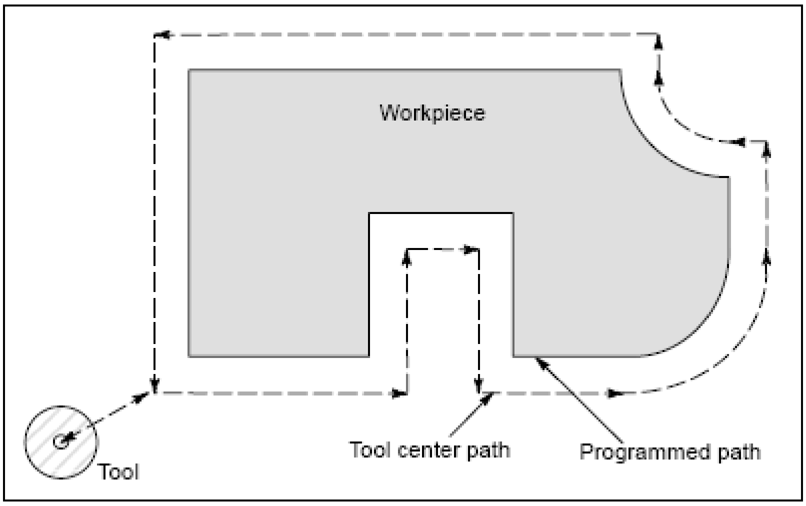


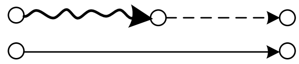
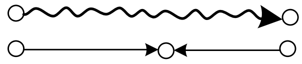
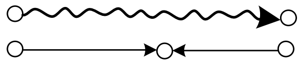
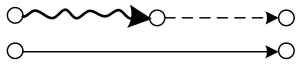
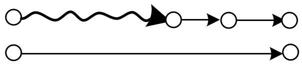
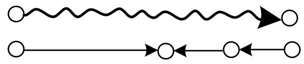
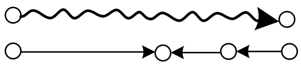
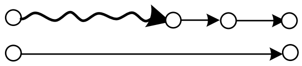
Fig. 3-5-85

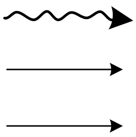
Explanations

1. Increase and decrease

As shown in the table below, the travel distance of the tool is increased or decreased by the specified tool offset value. In the absolute mode, the travel distance is increased or decreased as the tool is moved from the end position of the previous block to the position specified by the block containing G45 to G48.

Increase and decrease of the tool travel distance

G code	When a positive tool offset value is specified	When a negative tool offset value is specified
G45	 Start position End position	 Start position End position
G46	 Start position End position	 Start position End position
G47	 Start position End position	 Start position End position
G48	 Start position End position	 Start position End position



Programming movement distance

Tool offset value

Actual movement position

If a move command with a travel distance of zero is specified in the incremental command (G91) mode, the tool is moved by the distance corresponding to the specified tool offset value.
If a move command with a travel distance of zero is specified in the absolute command (G90) mode, the tool is not moved.

2. Tool offset value

Once selected by D code, the tool offset value remains unchanged until another tool offset value is selected.

Tool offset value can be set at the following range:

Range of tool offset values

	Metric input	Inch input
Tool offset value	-999.9999mm～999.9999 mm	-999.9999 inch ～ 999.9999 inch
	-999.9999 deg～999.9999 deg	-999．9999 deg～999．9999 deg

Note：D0 always indicates a tool offset value of zero.

Example:

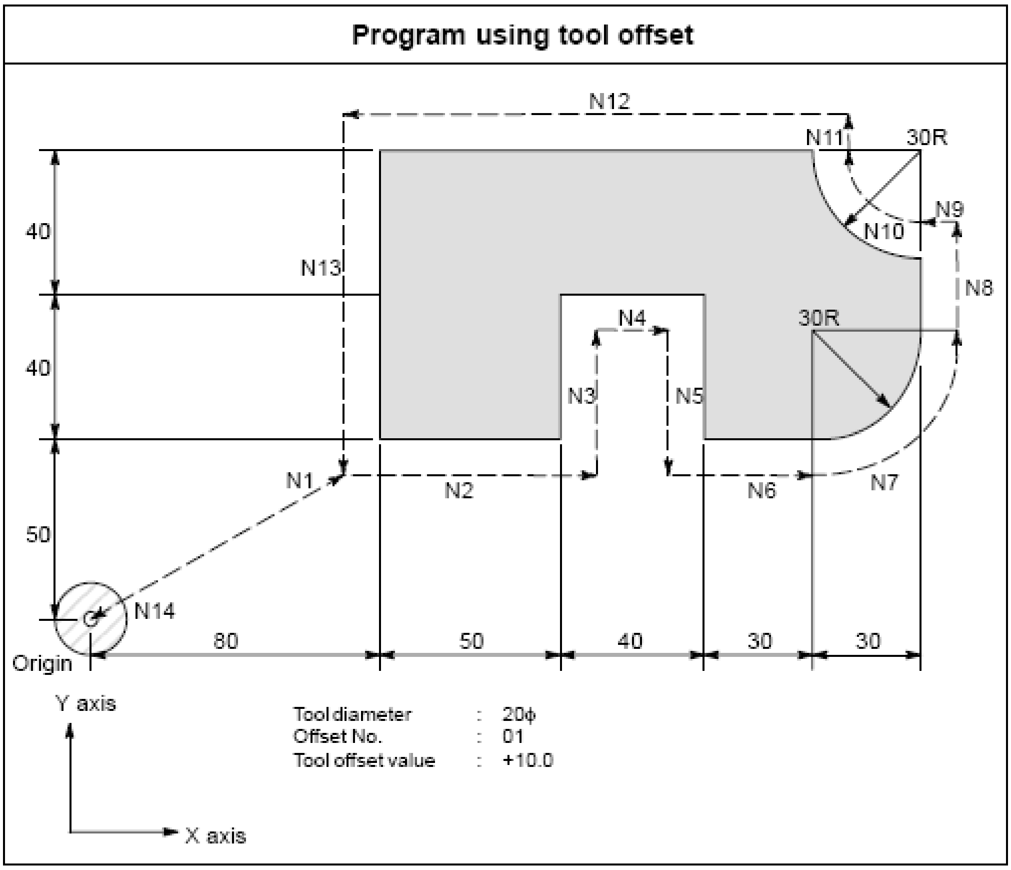


Fig. 3-5-86

Program

N1 G91 G46 G00 X80.0 Y50.0 D01 ;
N2 G47 G01 X50.0 F120.0 ;
N3 Y40.0 ;
N4 G48 X40.0 ;
N5 Y-40.0 ;
N6 G45 X30.0 ;
N7 G45 G03 X30.0 Y30.0 J30.0 ;
N8 G45 G01 Y20.0 ;
N9 G46 X0 ; (decreases toward the positive direction for movement amount “0”. The tool moves along the -X direction by the offset value.)
N10 G46 G02 X-30.0 Y30.0 J30.0 ;
N11 G45 G01 Y0 ; (increase toward the positive direction for movement amount “0”. The tool moves along the +Y direction by the offset value.)
N12 G47 X-120.0 ;
N13 G47 Y-80.0 ;
N14 G46 G00 X80.0 Y-50.0 ;

3.6 The Special Canned Cycle Commands

The special canned cycle and the standard canned cycle are used in combination. Before using the canned cycle, the canned cycle selects G commands and the hole processing data for programming, and the hole processing data are saved. Even after the special canned cycle is executed, the saved standard canned cycle still remains before canceling. If it isn’t in the canned cycle mode, alarm occurs when specifying special canned cycle. It can only be specified at G17 plane, the alarm will occur when it is specified at plane G18, G19.

Based on the different function of the continuous drilling, this chapter mainly introduces the path of the circle, the straight line, the arc, the chess board or the rectangle to call the canned drilling mode cycle for the drilling holes cycle in the consecutive space.

3.6.1 Circumference Hole Cycle (G120)

Format: G98/G99 **G120** G73~G89 X_ Y_ R_ Z_ (Q_ P_) I_ J_ K_ F_;
Or:

G98/G99 G73-G89 X_ Y_ R_ Z_ (Q_ P_) F_;

G120 X_ Y_ I_ J_ K_ ;

Explanations:

X, Y: The center position of the circumference hole cycles affected by G90/G91.

I : The radius r of the circle, the unit is based on the input setting unit and represented by the positive number.

J : The angle of the initial drilling hole position（The position of the decimal position is the degree ）is positive in CCW direction of X axis: CCW direction is positive and CW direction is negative.

K: The number of the drilling holes is n. The specified quantity is 1～9999 rather than 0. When 0 is specified, P221 alarms: the canned hole number is 0.

Take the coordinate specified by X and Y as the center to form the circumference of radius R, and the circumference is divided equally based on X axis and the angle to drill n holes. The drilling in each hole position saves G81 drilling data of the standard canned cycle. The movement in the hole position is processed in G00 mode. Moreover, after G120 command ends, parameters I, J, K specified in the cycle are not saved.

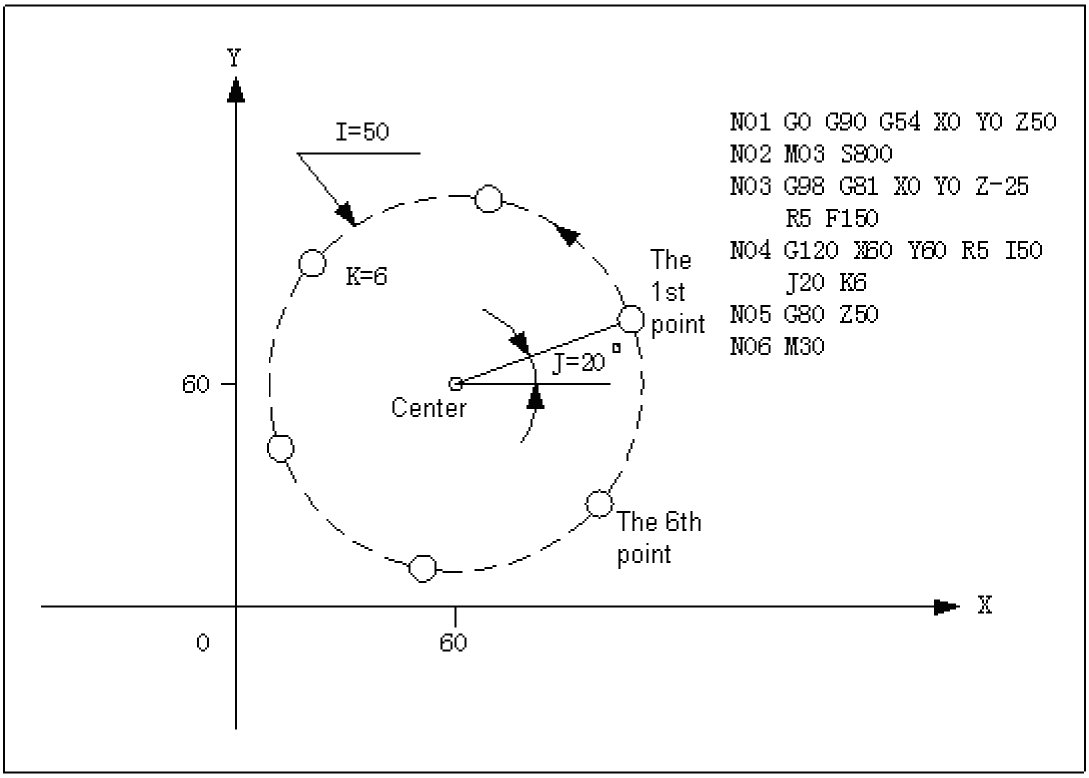


Fig. 3-6-1

Note: Please specify standard canned cycle command when setting G120. Otherwise, alarm will occur.

3.6.2 The Angle Straight Hole Cycle（G121）

Format: G98/G99 **G121** G73~G89 X_ Y_ R_ Z_ (Q_ P_) I_ J_ K_ F_;
Or:

G98/G99 G73~G89 X_ Y_ R_ Z_ (Q_ P_) F_;
G121 X_ Y_ I_ J_ K_;

Explanations:

X, Y: The positioning point that is used only for positioning other than drilling a hole. G90 is for absolute position and G91 is for position of increment corresponds to the current point.

I: Pitch of holes. When it is negative value, the starting position is taken as the center and the hole is drilled in the symmetrical direction.

J: Degree of the angle relative to the X-axis positive direction. The angle is positive in CCW direction and it is negative in CW direction.

K: The quantity of the holes, which isn't include the starting position (XY commands are reference points), its range is 1∽9999. The system alarms when the value is bigger than 0 and smaller than 1.

Definition:

It's assumed that the position specified by X and Y as the starting position, the direction formed by X axis and the angle J is differed by the interval I and divided drilling hole movement of K times. Based on the standard canned cycle G73-G89, the data of the drilling holes should be saved before drilling in each hole. The movement of each hole position is processed in G00 mode. Moreover, G121 is a non-mode command, after it ends, I, J, K parameters specified in the canned cycle are not saved.

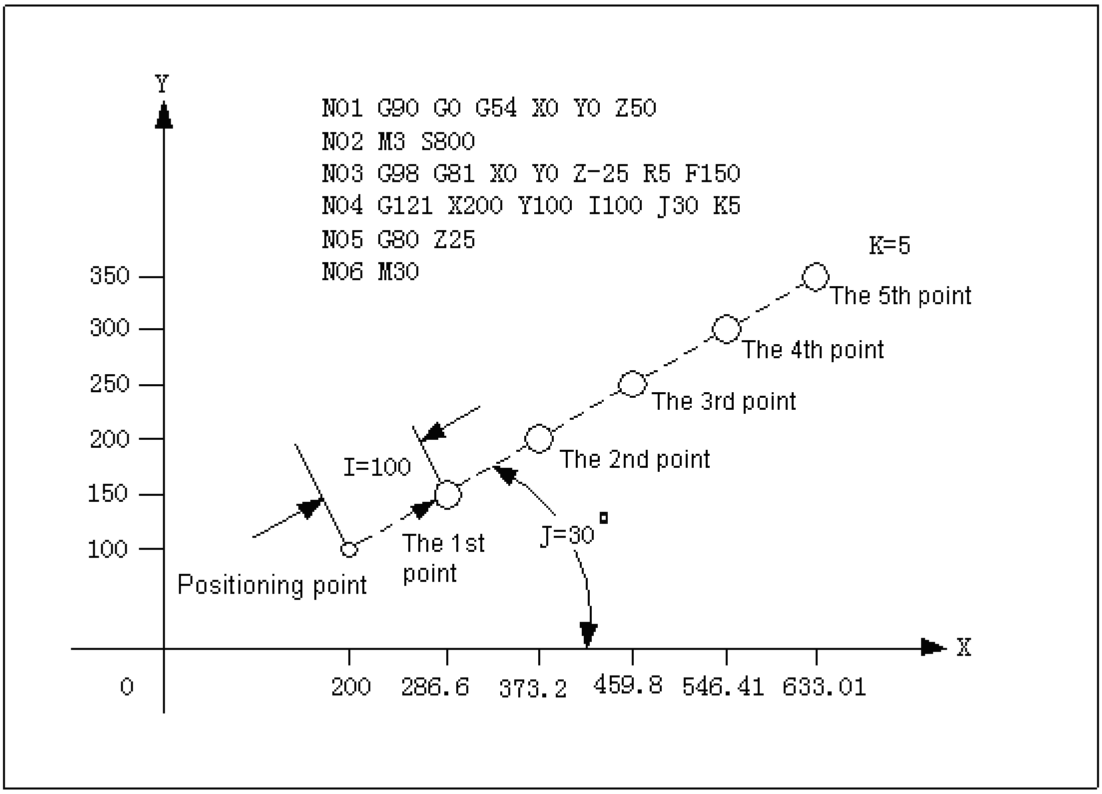


Fig. 3-6-2

Note:

1. When hole number K or it is set to 0, the system will alarm.
2. If G commands of group 0 are with G121 in one block, the following commands are priority.
3. Please specify standard canned cycle command when setting G121, otherwise, alarm occurs.

3.6.3 Arc Hole Cycle（G122）

Format: G98/G99 **G122** G73~G89 X_ Y_ R_ Z_ (Q_ P_) I_ J_ E_ K_ F_;

Or:

G98/G99 G73-G89 X_ Y_ R_ Z_ (Q_ P_) F_;

G122 X_ Y_ I_ J_E_ K_;

Explanations:

X, Y: The center coordinate of the arc is affected by G90/G91.

I: The unit of the arc radius r is based on the setting unit and represented in the positive number.

J: The angle of the initial drilling hole position is positive in CCW direction and is negative in CW position. It is on the +X axis when value 0 is specified.

E: The angle interval, and drill the holes positively in CCW direction and negatively in CW.

K: The quantity of n and the specified range is 1～9999.

Definition:

The coordinate specified by X and Y is taken as the center to form the circumference of radius I, and the point on the angle J is taken as the first drilling hole, k holes are drilled with interval E. Before special canned cycle G122 is performed, standard canned cycle G73～G89 should be specified to set hole drilling mode.

The movement of the hole position is executed in G00 mode. Moreover, G122 is non-mode command, I, J, E parameters specified are not saved.

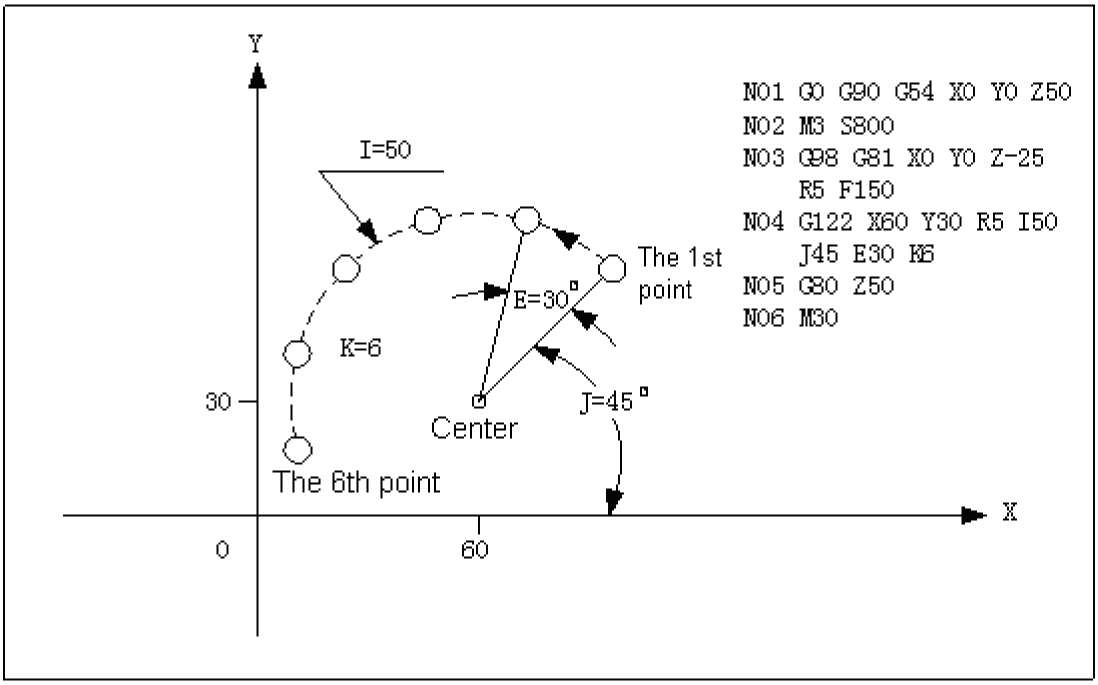


Fig. 3-6-3

Note: Please specify standard canned cycle command when setting G122, otherwise, system alarms.

3.6.4 The Chess Board Hole Cycle（G123）

Format: G98/G99 **G123** G73~G89 X_ Y_ R_ Z_ (Q_ P_) I_ J_ E_ K_ F_;

Or:

G98/G99 G73-G89 X_ Y_ R_ Z_ (Q_ P_) F_;

G123 X_ Y_ I_ J_ E_ K_;

Explanations:

G123 X_ Y_ I_ P_ J_ K_ ;

X, Y: Hole cycle starting point, that is to say, the first hole reference point. G90 is for absolute coordinate and G91 is for incremental coordinate relative to the former point.

I: The interval of X axis. The unit is based on the setting unit. When the value is positive, it is divided in the positive direction from the start position; negative, in the negative from the start position.

E: The quantity of holes in X axis direction and its range is 1～9999.

J: The interval of Y axis. The unit is based on the setting unit. When the value is positive, it is divided in the positive direction from the starting position; negative, in the negative from the starting position.

K: The quantity of holes in Y axis direction.

The position specified by X and Y is taken as the starting position, I is taken as grid interval for drilling E holes in the direction parallel to X axis. The drilling hole data（the hole processing mode and data）should be saved in advance because the drilling in each hole uses the standard canned cycle. The movement in each hole is processed in G00 mode. Moreover, the I, J, K parameters specified in cycle are not saved after G123 command ends.

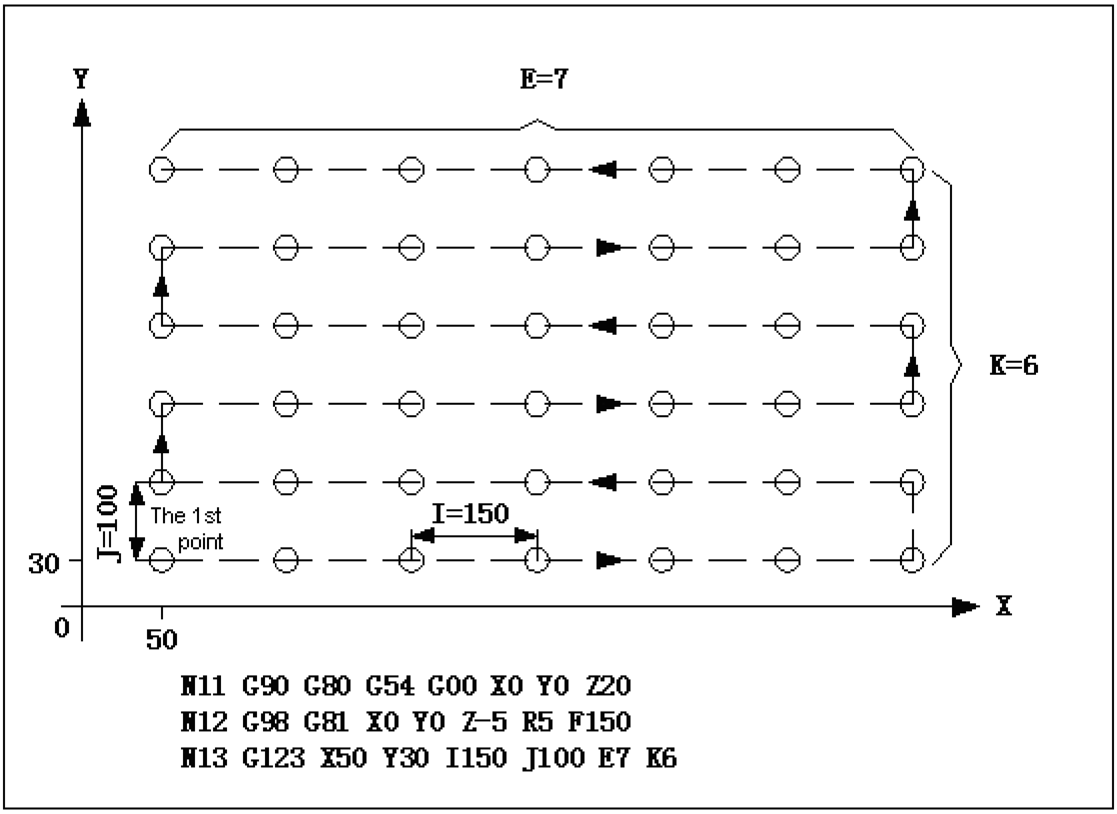


Fig. 3-6-4

Note: Please specify standard canned cycle command when setting G122, otherwise, system alarms.

3.6.5 Continuous Drilling in the Rectangle（G124/G125）

Format:

	G124		
G98/G99		G73-G89	X_ Y_ R_ Z_ (Q_ P_) J_ E_ K_ F_
	G125		

Or: G98/G99 G73-G89 X_ Y_ R_ Z_ (Q_ P_) F_;
G124/G125 X_ Y_ J_E_ K_;

Function:

Based on the number of drilling holes in each side, the holes are drilled continuously in each side of the rectangle.

Explanations:

G124 —drill holes in CW direction

G125 —drill holes in CCW direction

X,Y —The finishing end position coordinate of the 1st rectangular side

R — R Plane position

Z —The hole depth

E—The quantity of the drilling holes on the 1st and the 3rd sides

K —The quantity of the drilling holes on the 2nd and the 4th sides

J —The length of the 2nd and the 4th sides

F —Cutting feed rate

Example:

The drilling holes on the rectangular path, the starting position coordinate of the 1st side is X90, Y40 the length of the 1st side is the distance from the last point (here is point (0, 0)) to the point (90,40) ; the length of the 2nd side is 10mm (perpendicular to the 1st side). It is set by G124 and G125, and G81 drilling mode is used: drill three holes in the 1st and the 3rd sides, drill two holes in the 2nd and the 4th sides; the hole depth is 25mm. The sequence of drilling holes: the first, second, third and forth side.

The programming is shown as below:

G90 G17 G0 X0 Y0 Z25 M03 F1000;

G124 G81 X90 Y40 R5 Z-25 I40 J10 P3 K2 F800;

G80 G0 X100 Y100 M05;

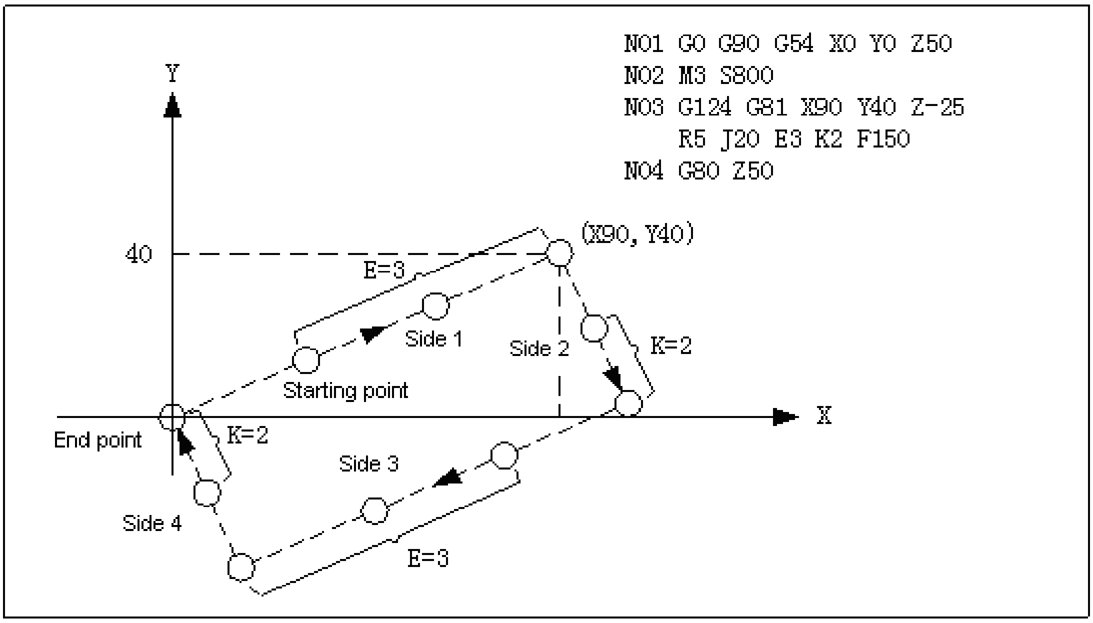


Fig. 3-6-5

Additional explanations:

1) The maximum value of hole number of each side E and K shall be 9999. Default value is 0, and it alarms when the negative value is specified. The decimal munber is rounded off.
2) A rectangular is defined by the current point, end point of the 1st side and the length of the 2nd side. The end point of the 1st side is not specified, which is defaulted to the current point. When the length of the 2nd side is not specified (namely, J is not set), J is 0. It drills hole repeatedly at the same position（total number of holes: (3+2)*2）
3) In the process of continuous hole drilling, return to relative plane based on G98/G99 specified

in block. Words G124, G125, E, K and J are valid in current block. If commands G124 and G125 are specified, while canned cycle (in drilling mode) command is not specified, alarm occurs. If G124 and G125 are not specified, while E, J and K are specified, E and J are ignored; K is the number of repetition.

3.6.6 Milling on the Plane（G126/G127）

Format:

G126

X_ Y_ Z_ R_ I_ J_ L_ F_

G127

Function: The plane is milled based on the specified length and width of each side.

Explanations:

G126 —Milling back and forth

G127 —Milling in one-way

X,Y —The coordinate of the starting position

Z —The cutting down length in Z axis direction

R —R plane position

I — The width in X axis direction. Positive value indicates it is on the positive direction of the starting point. Negative value indicates it is on the negative direction of the starting point.

J — The width in Y axis direction. Positive value indicates it is on the positive direction of the starting point. Negative value indicates it is on the negative direction of the starting point.

L —The cut width increment on X and Y planes should be less than the tool diameter more than 0.

F —Cutting feedrate

Example 1:

Milling back and forth on the plane requires the starting coordinate is（90，40）, the cutting down length in Z axis direction is 5mm, the width in X axis direction is 70mm, the width in Y axis direction is 30mmj, the cut width on the planes of X and Y is 10mm.

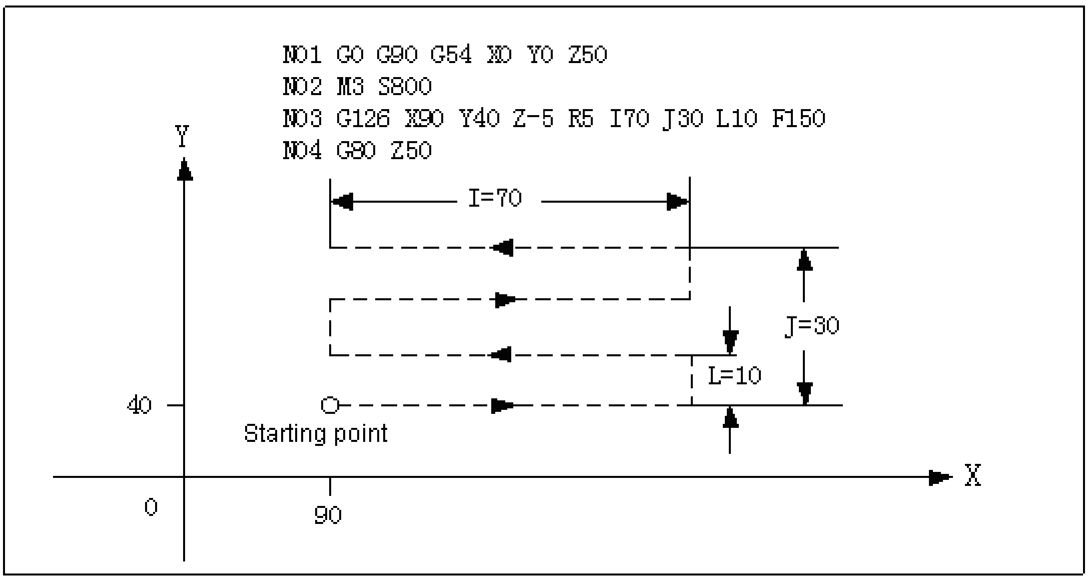


Fig. 3-6-6

Example 2:

Milling in one-way on the plane requires the starting coordinate is （90，40）, the cutting depth in Z axis is 5mm, the width in X axis is 70mm, the width in Y axis is 70mm, the cut width on the planes of X and Y is 10mm.

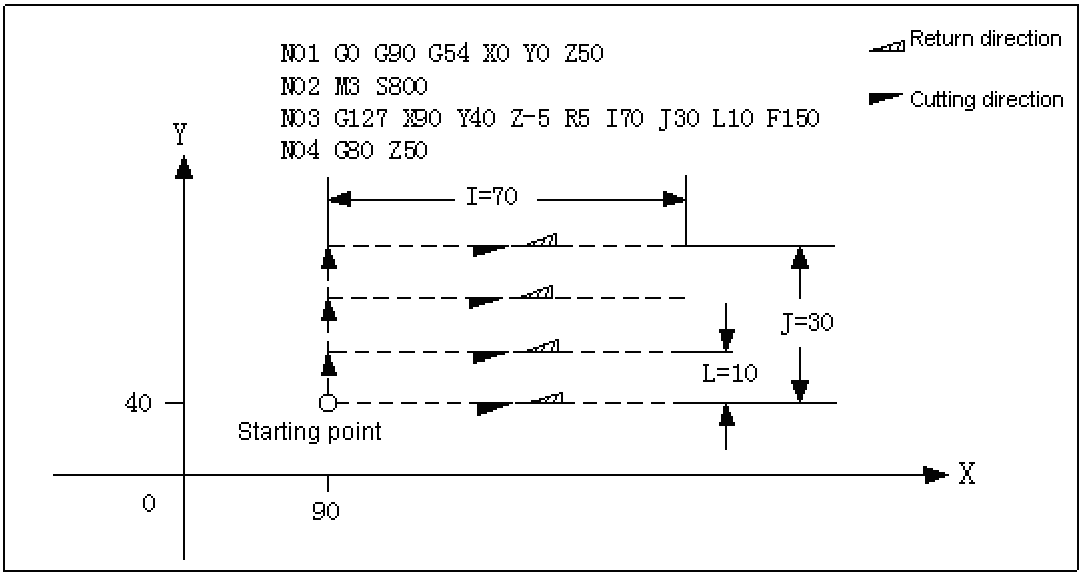


Fig. 3-6-7

Notes:

1. G126/G127 is non-mode command, by which I, J parameters specified are only valid in current block.
2. The current G126, G127 process are ignored when address value R and Z are defaulted. One address of R and Z is defaulted, G126, G127 are performed normally, and default address is treated as modal value.

3.7 Macro Function

3.7.1 The User Macro Program General Introduction

The user saves a group of instructions for a special function to memory as a subprogram in advance, and its function is indicated by an instruction. In the program, the function can be realized by specifying this instruction. The group of the instructions is called body of the user macro program (short for user macro program). Representative instruction is called user macro call instruction (short for macro instruction). Programming personnel only need to remember macro instruction but not to macro program. Macro program consists of variable, calculation command, control command etc.. The macro program is with the capacity and the flexibility which the standard G codes is lack of. Through the combination of the variable commands, commands of calling, various calculation, input and output the data between PLC, control, determining and branch, etc. can be executed in the macro program.

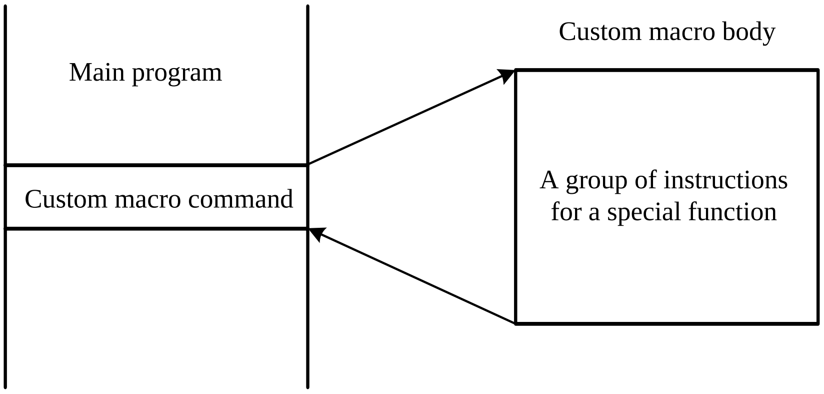


Fig. 3-7-1

3.7.2 The Variable

The common processing program can directly specify G codes and the movement distance through the numerical value; for example, when G01 and X200.0 use the user macro program, the numerical value can be directly specified or through the variable. The variable value can be rewritten through the program or MDI panel.

#1=#2+200； **G01 X#1 F300；**

3.7.2.1 The Variable Formula Representation

The variable is composed by the variable code （#） and its following variable number.

When the variable number is the numerical value;

#i （i=1, 2, 3, 4, 5……）

Example 1: #5

#109

#1005

－#20

The following formula can also be used, and the figure is replaced by the expression formula.

#〔the expression >〕

Example 2: **# [#100]**

 # [#1001-1]

 # [# 4/2]

The variable # i in the manual all are replaced by #〔<the expression formula>〕

3.7.2.2 Quoting the Reference Variable

After the address, the specified variable number can quote its variable value. When the expression formula specifies the variable, the expression formula should be bracketed.

Example:

G01X[#1+#2]F#3;

Put negative sign （-）in front of # to change the sign of the quoted variable value.

For example:

G00 X-#1；

When the variable isn't defined, the variable and the address character all are ignored.

Example:

When the variable value #1 is 0, and the value of the variable #2 is void null, the executing result of G00X#1 Y#2 **is** G00X0.

Note:

In the system, macro variable will be automatically rounded up when it is quoted by addresses S, D, H, P, M, T, L, G (exception for the conditions that macro variable is quoted in special cycles of G116 and G117)

Example:

#1 = 2.365;
H#1; (H is 2)

3.7.2.3 Undefined Variable

Generally, initialize the undefined variable before program is performed.

In the macro variables under 200, the variable value which isn't defined is called as the void null value. #0 is always used in the void variable, which can be read rather than written.

Note: In the system, macro variables above 500 (include 500) are initialized to 0 after power-on.

The undefined variable is with the following characteristics：

（1）Quotation

When one undefined variable is quoted, the address itself is ignored.

#1=<void value >	#1=0
G90 X100 Y#1 ↓ G90 X100	G90 X100 Y#1 ↓ G90 X100 Y0

（2）Calculation

Except <void value vacant> (in expression) is replaced, it is the same as variable value 0:

#1=< void value >	**#1=0**
#2=#1 ↓ **#2=0**	**#2=#1** ↓ **#2=0**
#2=#1*5 ↓ **#2=0**	**#2=#1*5** ↓ **#2=0**
#2=#1+#1 ↓ **#2=0**	**#2=#1+#1** ↓ **#2=0**

（3）Conditional expression

In the cases of E Q and N E, <void value> and 0 are determined as the different values.

#1=<void value >	#1=0
#1EQ#0 ↓ Definable	**#1EQ#0** ↓ Indefinable

#1NE#O ↓ Definable	**#1NE#O** ↓ Indefinable
#1GE#0 ↓ Definable	**#1GE#0** ↓ Definable
#1GT#0 ↓ Indefinable	**#1GT#0** ↓ Indefinable

3.7.2.4 Display and Setting the Variable

The variable value on LCD is blank, it means the variable is void;

The variable value displays as 0111 on LCD, it means overflow when the absolute value of the variable is above 99999999.

Note: In the system, macro variable displays calculation result when macro program is performing.

3.7.2.5 The Solution Range of the Variable

The range of the part variable and the public variable is **-999999.9999**～**+999999.9999**, it alarms if it's out of the range.

3.7.3 Types of the Variables

The variables are classified into the local variable, the common variable and the system variable, the purposes and the characteristics of the variables in each type are different.

3.7.3.1 The Local Variables # 1～# 33

The part variable is used to save data in a macro program (for example intermediate value of calculation, calculation result etc.). It is initialized as void value at power-off. When reset key is pressed or M30 program ends, 6001 #1 decides whether to initialize the variable to void. When the macro program is called, the independent argument variable assigns a value to the part local variable, the part variable which isn't assigned a value is used by the user at random. About the corresponding relation between the part variable and the independent variable, refer to the chapter of calling the macro program.

3.7.3.2 The Common Variables #100～ #199 ，#500～ #999

Different macro programs can share these common variables. When power is off, the variable #100—#199 is initialized as void. When reset key is pressed or M30 program ends, 6001 #2 decides whether to initialize to null value.

The data of the variable #500——#999 are saved, it doesn't get lost even the power is off.

3.7.3.3 System variable ＃1000~

The system variables are the ones of the canned purpose in the system, and it is classified into three types of reading, writing and reading and writing. Read and write the various data of CNC, for example, the current position data and the compensation value of the tool. The system variable is the base of the auto control and the common program development.

（1）The interface signal is from #1000 to #1031 and from #1032 to #1035, from #1100 to #1131 and from #1132 to #1135.

The interface signal is the interchanging one between the programmable machine controller （PLC） and the user macro program.

List 3-8 The system variable of the interface signal

The system variable number	Property	Function
#1000—#1031	Read	The signal in 32 digits is sent from PLC to the user macro program, and the signal is read from the variable #1000 to #1031 based on the bit, the interface input signal is from UI000 to UI031.
#1100—#1131	Read/write	The signal in 32 digits is sent from the user macro program to PLC, and the signal is written from the variable #1100 to #1131 based on the bit, the interface input signal is from UO000 to UO031.
#1032 —#1035	Read	The signal in 32 digits is output from PLC to the variable of the user macro program and the span of the variable value is from为 -999999.9999～+999999.9999.
#1132 —#1135	Read/write	The signal of 32 digits is written into the variable of the user macro program and the variable value range is from -999999.9999～+999999.9999.

（The tool offset value）#2001～#2400

The system variable can read and write the tool compensation value.

List 3-9 The system variable of the tool compensation value

Compensation number	Property	The tool length compensation（H）		The tool radius compensation（D）	
		Appearance geometric compensation	Wearing compensation	Appearance geometric compensation	Appearance wear compensation
1 : 200 : 400	Read/write	#11001 （#2201） : #11200 （#2400） : #11400	#10001 （#2001） : #10200 （#2200） : #10400	#13001 : : : #13400	#12001 : : : #12400

Example: #30=#2005

In the tool offset number, the tool offset value is substituted into the variable #30.

When the offset value is 4, the value of #30 is changed into 4.

#2210=#30

The offset value of the current offset number #10 is written and equal to that of #30 variable.

（3）Macro program alarm #3000

Only the variable is written, when the value of the variable #3000 is 0～200, CNC stops running and alarms. After the expression formula, the alarm information within 26 characters is specified. The alarm number and information display on CRT screen, and the value of which alarm number is #3000 adds 3000.

Example: #3000=1（The tool can't be found.）

"3001 (TOOL NOT FOUND)"displays on the alarm screen.

Suggestion: #3000 macro program alarm should be specified in a single block.

（4）The time information #3001, #3002, #3011, #3012

List 3-10 The system variable list of the time information

VARIABLE NUMBER	PROPERTY	FUNCTION
#3001	Read/write	The variable is a counter, and one millisecond is the unit. When it powers on, the variable value is reset as 0. When it reaches millisecond of 2147483648, the value of the counter returns to 0.
#3002	Read/write	The variable is a counter, and one hour is the unit. Start counting when the auto running starts and the counter also saves the numerical value even it powers off. When it reaches 9544.371767 hours, the value of the counter returns to 0.
#3011	Read/write	The variable is for reading the current date（year/month/day）. The information of year/month/day is switched in the decimal system. For example: 9/28/2008 is expressed as 080928
#3012	Read/write	The variable is to read the current time （hour/minute/second）. The information of hour/minute/second is switched in the decimal system. For example, pm 15:34:56 is represented as 153456.

（5）Prohibition of stopping the single block and waiting the miscellaneous function finish signal#3003

List 3-11 System variable of the auto running control（#3003）

#3003	**PROPERTY**	**SINGLE BLOCK**	**FINISH SIGNAL OF THE MISCELLANEOUS FUNCTION**
0	Read/write	Valid	Wait
1	Read/write	Invalid	Wait
2	Read/write	Valid	Not wait
3	Read/write	Invalid	Not wait

Notes:

·When the power is on, the variable value is 0.

·When stopping the single block is invalid, even the single block switch is ON, the single block doesn't stop.

·When the no waiting miscellaneous function（M, S and T function）is specified, the program executes the next block before the miscellaneous function ends.

Example: The drilling holes in cycle（relative to the incremental programming）is equivalent to G81.

Macro program calling commands

G65 P9081L（repeated times）R（point R）Z（point Z）;

Editing the macro program itself is shown as below：

09081;

#3003=1;

Part Ⅰ Programming

G00 Z#18;
G01 Z#26;
G00Z-[ROUND（#18）+ROUND（#26）];
#3003=0;
M99;
The single block doesn't stop, #18 is relative to R, #26 to Z.

（**6**）Feed hold. The valid and invalid conditions for the feed rate override and exactly stop **#3004.**

#3004	PROPERTY	FEED HOLD	FEED RATE OVERRIDE	EXACT STOP
0	Read/write	Valid	Valid	Valid
1	Read/write	Invalid	Valid	Valid
2	Read/write	Valid	Invalid	Valid
3	Read/write	Invalid	Invalid	Valid
4	Read/write	Valid	Valid	Invalid
5	Read/write	Invalid	Valid	Invalid
6	Read/write	Valid	Invalid	Invalid
7	Read/write	Invalid	Invalid	Invalid

Notes:

·When it powers on, the variable value is 0.

·When the feeding pause hold is invalid：

a. When the feed hold button is pressed, the machine stops in single block stopping mode. However, when the variable #3003 makes the single block mode invalid, the single block doesn't stop.

b. The feeding pause indicator is on when the feeding pause button is released after being pressed. However, the machine doesn't stop; the program continues executing, and the machine stops in the first block which the feeding pause is valid.

·When the feed rate override is invalid, the override is always 100% and it doesn't have any connection with the feed rate override switches on the machine operational panel.

·When exact stop detection is invalid, even the block which doesn't execute the cutting doesn't execute the exact stop detection（position detection）.

Example: The tapping cycle（Relative to the incremental programming）（equivalent to G84）

The macro program calling commands

G65 P9084 L（Repeated times）R（point R）Z（point Z）;

Editing the macro program itself is as below：

09084;
#3003=1;
G00Z#18; Prohibit stopping the single block
#3004=7;
G01Z#26;
M05; The feed hold, the feed rate override and the exact stop checking are invalid.
M04;
Z-#26;
#3004=0;
M05;
M03;
G00Z-#18;
#3003=0;
M99;

(7) Number of parts #3901,#3902

The part number(target number)and the processed part number (processed number)can be read and written.

VARIABLE NUMBER	PROPERTY	FUNCTION
#3901	Read/write	The processed part number (the processed number)
#3902	Read/write	The required part number (target number)

Note: The numerical value can't be negative.

(8) The mode information #4001～#4130

The mode information before the processing block can be read.

SYSTEM VARIABLE	PROPERTY	MODE INFORMATION	GROUPS
#4001	Read	**G00, G01, G02, G03, G6.2**	**01**
#4002	Read	**G17, G18, G19**	**02**
#4003	Read	**G90, G91**	**03**
#4004	Read	**G22, G23**	**04**
#4005	Read	**G94, G95**	**05**
#4006	Read	**G20, G21**	**06**
#4007	Read	**G40, G41, G42**	**07**
#4008	Read	**G43, G44, G49**	**08**
#4009	Read	**G73, G74, G76, G80~G89**	**09**
#4010	Read	**G98, G99**	**10**
#4011	Read	**G50, G51**	**11**
#4012	Read	**G66, G67**	**12**
#4013	Read		**13**
#4014	Read	**G16, G15**	**14**
#4015	Read	**G61~G64**	**15**
#4016	Read	**G68, G69**	**16**
#4017	Read	**G54~G59**	**17**
. . .	Read	. . .	. . .
#4021	Read	**G50.1, G51.1**	**21**
#4107	Read	Codes D	
#4109	Read	Codes F	
#4111	Read	Codes H	
#4113	Read	Codes M	
#4114	Read	serial number N	
#4119	Read	S codes	
#4120	Read	T codes	
#4130	Read	Additional work piece coordinate system number P	

Example: The combined programming of the incremental value/the absolute value, the boring hole cycle (equivalent to G86).

The macro program calling commands

G65 P9086L (Repeated times) R (point R) Z (point Z):

The macro program itself is edited as below:

09086;

#1=#4003; Save G codes in group 03

#3003=1; Prohibit stopping the single block

G00 G91 Z#18;

```
G01   Z#26;
M05;
G00 Z-[#18+#26];
M03;
#3003=0;
G#1 M99;   Restore G codes in group 03
```

（9）The position information #5001～#5105

The position information can be set by the system variables #5001～5105 and its unit is the millimeter or the inch set by the input system.

SYSTEM VARIABLE	PROPERTY	POSITION INFORMATION	COORDINATE SYSTEM	TOOL COMPENSATION VALUE	READ DURING MOVING
#5001 #5002 #5003 #5004 #5005	READ	X The end position of X axis block （ABSIO） Y The end position of Y axis block （ABSIO） Z The end position of Z axis block （ABSIO） The end position of the 4th axis block （ABSIO） The end position of the 5th axis block （ABSIO）	The work piece coordinate system	Exclude	Possible
#5021 #5022 #5023 #5024 #5025	READ	X The current position of X axis （ABSMT） Y The current position of Y axis （ABSMT） Z The current position of Z axis （ABSMT） The current position of the 4th axis （ABSMT） The current position of the 5th axis （ABSMT）	The machine coordinate system	Include	Impossible
#5041 #5042 #5043	READ	X The current position of X axis （ABSMT）			Impossible

#5044 #5045		Y The current position of Y axis（ABSMT） Z The current position of Z axis（ABSMT） The current position of the 4^{th} axis（ABSMT） The current position of the 5^{th} axis（ABSMT）	The work piece coordinate	Include	
#5061 #5062 #5063 #5064 #5065	READ	X Skipping the signal position in X axis（ABSKP） Y Skipping the signal position in Y axis（ABSKP） Z Skipping the signal position in Z axis（ABSKP） Skipping the signal position in the 4^{th} axis（ABSKP） Skipping the signal position in the 5^{th} axis（ABSKP）	The work piece coordinate system	Include	Possible
#5081 **#5082** **#5083** **#5084** **#5085**	READ	The tool length offset value			Impossible
#5101 #5102 #5103 #5104 #5105	READ	X The servo position offset in X axis Y The servo position offset in Y axis Z The servo position offset in Z axis The servo position offset in the 4^{th} axis The servo position offset in the 5^{th} axis			Impossible

Note:

·In G31（jumping function）block, when the jumping signal is connected, the tool position is saved in the variables from #5061 to #5063. When the jumping signals in G31 block isn't connected, the tool position saved in these variables specifies the finishing position value of the block.

·During the traverse, the expected value can't be read due to the buffer function（preread）.

（10） The work piece coordinate system compensation value （the work piece zero position offset value）＃2500～#2906

The work piece zero position offset value can be read and rewritten.

Axes	FUNCTION	PROPERTY	VARIABLE NUMBER
The first axis	The external work piece zero position offset G54 The work piece zero position offset G55 The work piece zero position offset G56 The work piece zero position offset G57 The work piece zero position offset G58 The work piece zero position offset G59 The work piece zero position offset	Read	#2500 #2501 #2502 #2503 #2504 #2505 #2506
The 2nd axis	The external work piece zero position offset G54 The work piece zero position offset G55 The work piece zero position offset G56 The work piece zero position offset G57 The work piece zero position offset G58 The work piece zero position offset G59 The work piece zero position offset	Read	#2600 #2601 #2602 #2603 #2604 #2605 #2606
The 3rd axis	The external work piece zero position offset G54 The work piece zero position offset G55 The work piece zero position offset G56 The work piece zero position offset G57 The work piece zero position offset G58 The work piece zero position offset G59 The work piece zero position offset	Read	#2700 #2701 #2702 #2703 #2704 #2705 #2706
The 4th axis	The external work piece zero position offset G54 The work piece zero position offset G55 The work piece zero position offset G56 The work piece zero position offset G57 The work piece zero position offset G58 The work piece zero position offset G59 The work piece zero position offset	Read	#2800 #2801 #2802 #2803 #2804 #2805 #2806

The 5th axis	The external work piece zero position offset G54 The work piece zero position offset G55 The work piece zero position offset G56 The work piece zero position offset G57 The work piece zero position offset G58 The work piece zero position offset G59 The work piece zero position offset	Read	#2900 #2901 #2902 #2903 #2904 #2905 #2906

3.7.4 The Operational Commands

Various operations can be operated among the variables.

#i=＜Expression ＞

The right ＜expression formula＞ of the operational command is the combination of the constant, the variable, the function and the operator. The constant can replace the right ＜expression formula＞. The constant free of the decimal position in ＜expression formula＞ can be processed as it's with the decimal position at its end.

3.7.4.1 Arithmetic Operation

#i=#j+#k Addition
#i=#j-#k Subtraction
#i=#j*#k Multiplication
#i=#j/#k Division
#i=#jMOD#k Mod

3.7.4.2 Logical Operations

#i=#joR#k Logic sum （for each binary number）

#i=#jXOR#K Anticoincidence X OR/Exclusive OR（for each binary number）

#i=#jAND#k And (for each binary number)

3.7.4.3 Function

#i=SIN[#j] Sine（Unit：degree）
#i=COS[#j] Cosine （Unit：degree）
#i=TAN[#j] Tangent （Unit：degree）
#i=ASIN[#j] Inverse sine（Unit：degree）
#i=ACOS[#j] Arc cosine （Unit：degree）
#i=ATAN[#j] Arc tangent （Unit：degree）
#i=SQRT[#j] Square root
#i=ABS[#j] Absolute value
#i=BIN[#j] Switch from BCD to BIN (Rounded to one decimal place)
#i=BCD[#j] Switch from BIN to BCD (Rounded to one decimal place)
#i=ROUND[#j] Round off (e.g. 0 .001, Rounded to three decimal places)
#i=FIX[#j] Round up the part after the decimal position
#i=FUP[#j] The decimal position part is forward into the integer part.
#i=LN[#j] Natural logarithm
#i=EXP[#j] Exponential function

Note: How to use the function ROUND.

（1）In the function ROUND, the data should be rounded up with 0.001 unit in any case.

Example: G01 X[ROUND #1]

If #1 is 1.4567 and X minimum input increment is 0.001, the block changes into G01 X1.457

Note: In this system, floating point comparison can be used in IF or WHILE conditions.

（2）Arc tangent #i=ATAN[#j]:（one independent variable） （(-90°≦ ATAN[#j] ≦ 90°)）. Namely, it becomes ATAN of the calculator.

3.7.4.4 Combined Calculation

The above operation and the function can be combined. The operational preferential order is: the function, then the multiplication and division, finally the addition and subtraction.

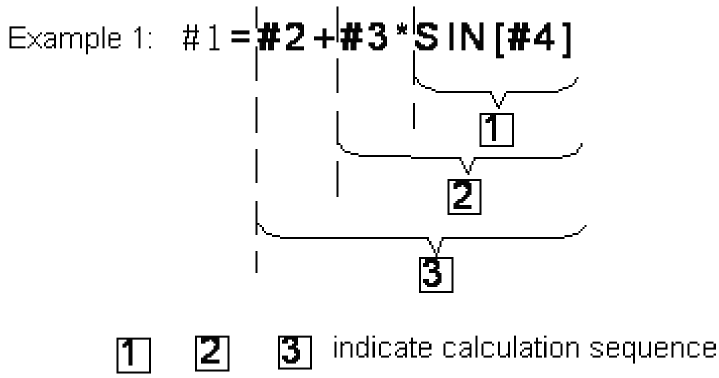


Fig. 3-7-2

3.7.4.5 Changing the Operational Order Through [ ]

[ ] is used to change the order of the calculation. [ ] can be nested for 5 layers including the bracket in the function itself. Alarm occurs when exceeding 5 layers.

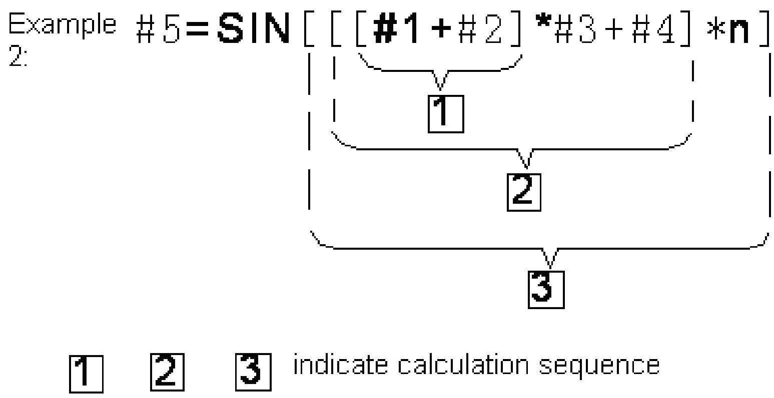


Fig. 3-7-3

3.7.4.6 Precision

Pay attention to the precision during programming through the macro program function.

（1）Operational precision

The operation is executed for one time, the following error exists, and these errors are accumulated after the repeated operation.

OPERATION FORMULA	MEAN ERROR	MAX ERROR	TYPE OF ERRORS
a=b*c	1.55×10^{-10}	4.66×10^{-10}	Relative error

a=b/c	4.66×10^{-10}	1.86×10^{-9}	$\left\lvert \frac{\varepsilon}{a} \right\rvert$
a=$\sqrt{b}$	1.24×10^{-9}	3.73×10^{-9}	
a=b+c a=b-c	2.33×10^{-10}	5.32×10^{-10}	min（$\frac{\varepsilon}{b}$, $\frac{\varepsilon}{c}$）
a=SiNb a=comb	5.0×10^{-9}	1.0×10^{-9}	Absolute error $\lvert\varepsilon\rvert$ degree
a=ATANb/c	1.8×10^{-6}	3.6×10^{-6}	

3.7.5 Macro Statement and NC Statement

Program blocks below are macro statements:

Blocks containing arithmetic command (=), (when the block at the same block with NC statement, they are executed simultaneously.)

Blocks containing control command （e.g. GOTO DO END WHILE IF）, (when the block at the same block with NC statement, alarm occurs)

Blocks contain macro commands (e.g. macro program includes G65, G66, G67 or other macro program contains G code and M code). Any block other than a macro statement is referred to as an NC statement.

3.7.6 Branch and Repetition

In a program, the flow of program can be changed using the GOTO statement and IF statement. Three types of branch and repetition operations are used:

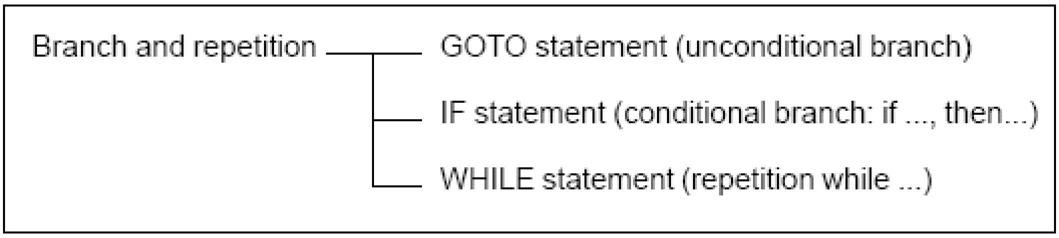


Fig. 3-7-4

Suggestion: Branch and repetition command shall be specified by a single block.

3.7.6.1 Unconditional Branch (GOTO)

A branch transfers to statement n. When a sequence number outside of the range 1~999999, the alarm occurs.

GOTOn: n is sequence number 1~999999

Example: GOTO 1

3.7.6.2 Conditional Branch (IF Statement)

Specify a conditional expression after IF.

IF[＜conditional expression＞] GOTOn

If the specified conditional expression is satisfied, a branch to a sequence number n occurs. Otherwise, the next block is executed.

If the value of variable #1 is greater than 10, a branch to sequence number N2 occurs.
If the condition is not satisfied — IF [#1 GT 10] GOTO 2 ; → Processing — If the condition is satisfied → N2 G00 G91 X10.0 ; :

Fig. 3-7-5

A conditional expression must include an operator inserted between two variables or between a variable and a constant, which must be enclosed in brackets [ ].

An operator consists of two letters and is used to compare two values to determine whether one value is smaller or greater than the other value. Note that inequality sign can not be used.

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(≦)

Fig. 3-7-6

The sample program below finds the total of number 1 to 10.

```
O9500;
 #1=0; . . . . . . . . . . . . . . . . . . . . Initial value of the variable to hold the sum
 #2=1; . . . . . . . . . . . . . . . . . . . . Initial value of the variable as an addend
N1 IF[#2 GT 10] GOTO 2; . . . Branch to N2 when the addend is greater
                               than 10
 #1 =#1+#2; . . . . . . . . . . . . . . . Calculation to find the sum
 #2=#2+1;  . . . . . . . . . . . . . . . . Next addend
 GOTO 1; . . . . . . . . . . . . . . . . . Branch to N1
N2 M30; . . . . . . . . . . . . . . . . . End of program
```

Fig. 3-7-7

Note: Blocks behind block n are executed after performing GOTOn command. The sequence number n should write at the beginning of the clock.

3.7.6.3 Repetition (Sentence **WHILE**)

WHILE[＜the conditional formula＞］＝DOm（m=1,2,3）

ENDm

When <the conditional formula> is satisfied, the blocks from Dom to ENDm are executed after

WHILE execution; When <the conditional formula> is not satisfied, the blocks after ENDm are executed. Conditional formula and comparison symbols are same as that of IF.

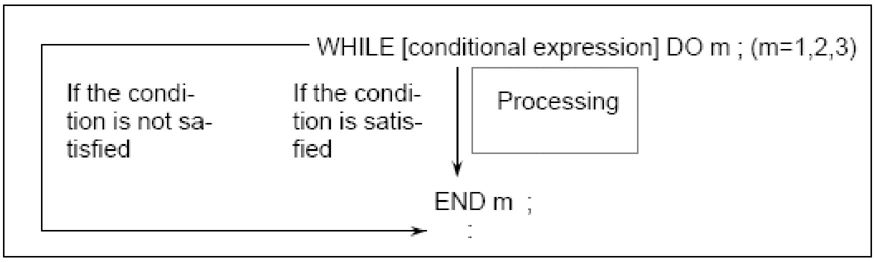


Fig. 3-7-8

A number after DO and END are identification numbers for specifying the range of execution. When a number other than 1, 2 and 3 is used as identification number, alarm occurs.

The identification number (1-3) in DO-END loop can be used as many times as desired.

Note: when a program includes crossing repetition loops, alarm occurs.

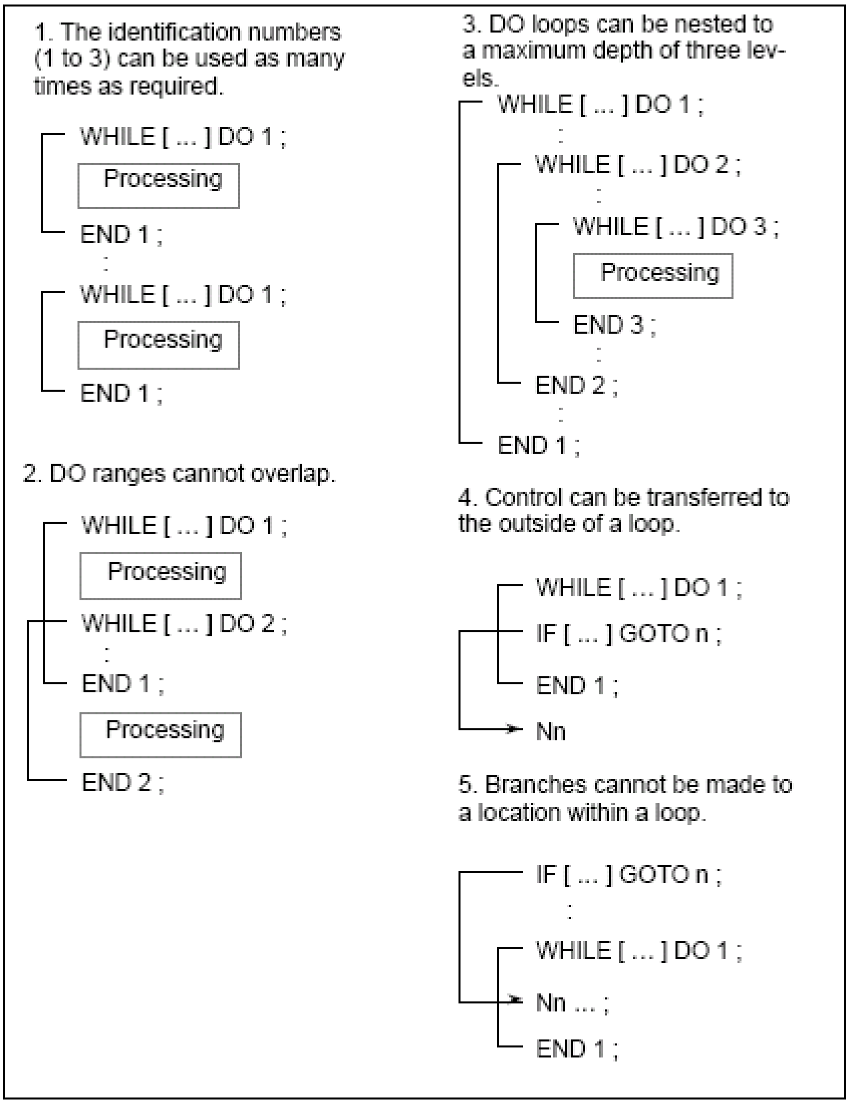


Fig. 3-7-9

When DO is specified without specifying the WHILE sentence, an infinite loop ranging from DO to END.

Example:

```
O0001;
#1=0;
#2=1;
WHILE[#2 LE 10]DO 1;
#1=#1+#2;
#2=#2+1;
END 1;
M30;
```

Fig. 3-7-10

3.7.7 Macro Program Calling Commands

3.7.7.1 Simple Calling

The macro program calling:

Non-mode calling （G65）

Mode calling （G66, G67）

The macro program is called through G codes.

The macro program is called through M codes.

The subprogram is called through M codes.

The subprogram is called through T codes.

The macro program calling （G65） is different with the subprogram calling（G98）, which is introduced as below：

·The independent argument variable (the data are transmitted into the macro program) can be specified through G65 while G98 is lack of the function.

·When M98 block includes the other NC command (such as, G01 X100.0 M98 Pp), the subprogram is called after the commands are executed. Contrarily,

If G65 commands are used, the level of the part local variable is changed. However, if M98 commands are used, the level of the part variable isn't changed.

3.7.7.2 Non-mode Calling（G65）

When G65 is specified, the user macro program specified by address P is called. The data (the independent argument variable) can be sent to the user macro program.

G65 P L <argument–specification> ;

P : Number of the program to call
ℓ : Repetition count (1 by default)
Argument : Data passed to the macro

```
O0001 ;
  :
G65 P9010 L2 A1.0 B2.0 ;
  :
M30 ;
```

```
O9010 ;
#3=#1+#2 ;
IF [#3 GT 360] GOTO 9 ;
G00 G91 X#3 ;
N9 M99 ;
```

Fig. 3-7-11

（1） Specifying the independent argument variable

The independent variable can be specified in two types. The independent variable I uses the letters except for G, L, O, N and P, and each letter specifies for one time. Ⅱ uses A, B, C and I_i, J and K_i (i is 1～10). Based on the used letter, the independent variable type can be auto set.

A) Independent variable I

The corresponding relation between the addresses in the independent variable assignment I and the variable number in the macro program is shown as below:

ADDRESSES OF THE INDEPENDENT VARIABLE	VARIABLES IN THE MACRO PROGRAM
A	#1
B	#2
C	#3
D	#7
E	#8
F	#9
H	#11
I	#4
J	#5
K	#6
M	#13
Q	#17
R	#18
S	#19
T	#20
U	#21
V	#22
W	#23
X	#24
Y	#25
Z	#26

Addresses G, L, N, O and P can not be used in arguments.

Addresses do not need to be specified alphabetically. They confirm to word address format..

Example:

B_A_D_…J_K_ Correct

B_A_D_…J_I_ Not correct

B) Independent variable assignment II

A__ B__ C__ I__J__K__ I__J__K__ _ _ _ _

The independent value can assign the values to A, B and C; moreover, the independent variable of the maximum ten groups can be specified by addresses I, J and K. The corresponding relation between the addresses distributed by the independent variable II and the variable number of the macro program is shown as below:

ADDRESSES OF THE INDEPENDENT VARIABLE ASSIGNMENT II	VARIABLES IN THE MACRO PROGRAM
A	#1
B	#2
C	#3
I1	#4
J1	#5
K1	#6
I2	#7
J2	#8
K2	#9
I3	#10
J3	#11
K3	#12
I4	#13
J4	#14
K4	#15
I5	#16
J5	#17
K5	#18
I6	#19
J6	#20
K6	#21
I7	＃22
J7	＃23
K7	＃24
I8	＃25
J8	＃26
K8	＃27
I9	＃28
J9	＃29
K9	＃30
I10	＃31
J10	＃32
K10	＃33

The suffix of I, J and K is the order of the assigned group.

C) Mixed using the independent variable I and II

It doesn't alarm even the independent variables of the assignment I and II are in the block with command G65. If the independent variables I and II correspond to the same variables, the later specified one is valid.

Example 1:

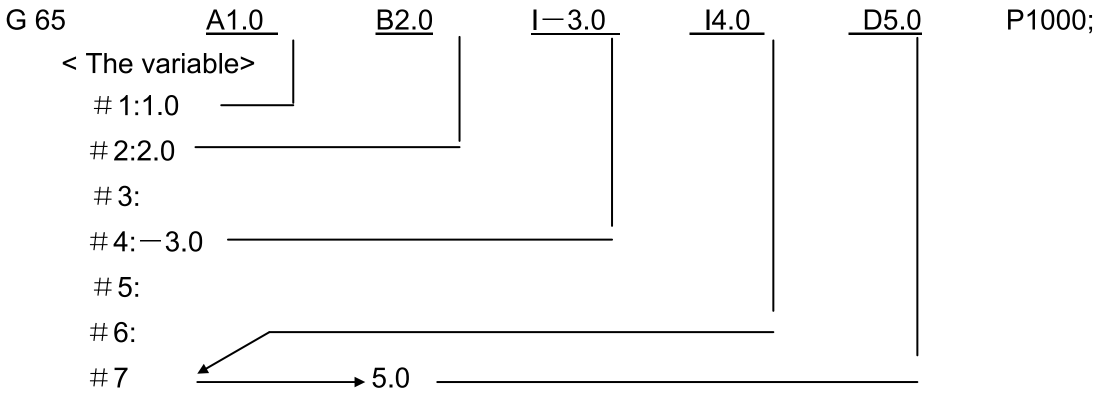


Fig. 3-7-12

In the example, although the independent variables I 4.0 and D 5.0 all are specified in ＃7 variable, the later specified D 5.0 is avid. Calls can be nested to a depth of 4 levels including simple call（G65）. 0-4 levels local variables are provided for the nest, and main program is on the level 0.

Each time a macro is called (with G65), the local variable level is incremented by one. The values of the local variables at the previous level are saved in the CNC. When M99 is executed in a macro program, control returns to the calling program. At that time, the local variable level is decremented by one. The values of the local variables saved when the macro was called are restored.

Relationship between macro program call and local variable are shown as follows:

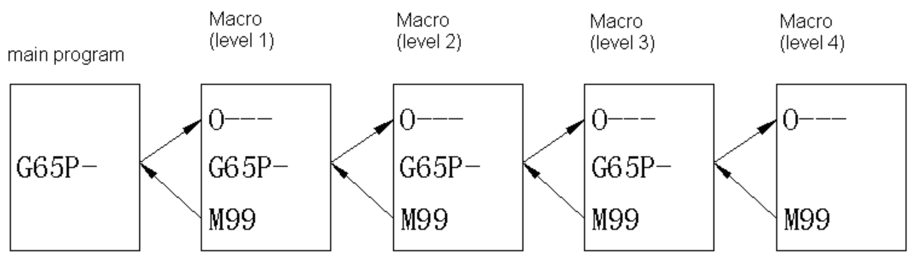


Fig. 3-7-13

Local variables：

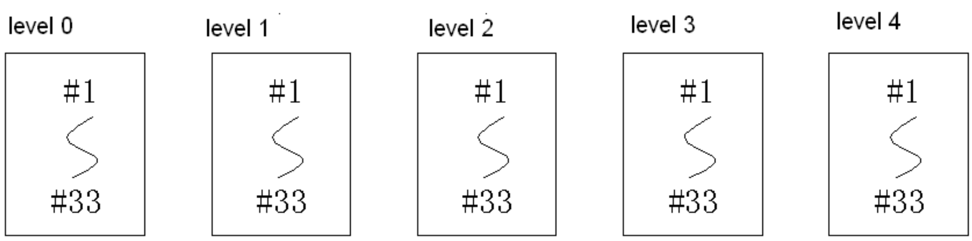


Fig. 3-7-14

Example 2: The tap ring:

The reference position set by the macro program which is set by the datum position is taken as the center of a circle, h holes to be processed is distributed on the ring on the equal interval. The 1st hole is on the straight line of the angle a, refer to the following figure：

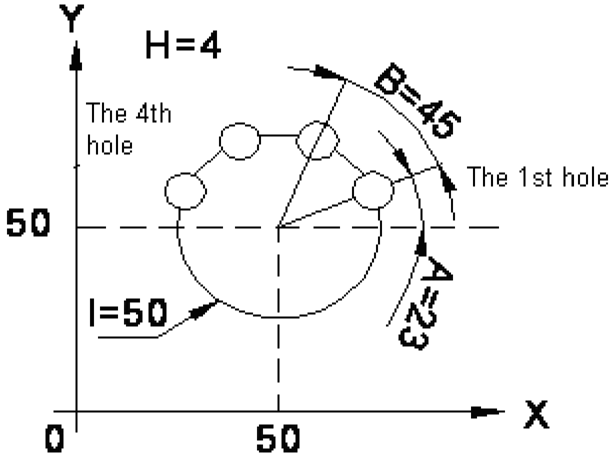


Fig. 3-7-15

Before hole processing is specified, the hole reference point (the circle center) should be set.

X coordinate value of X holes reference point

Y coordinate value of Y holes reference point

Commands of macro program call:

G65 P_ X _Y_Z_R_F_A_B_I_H_;

The following variables should be used:

#1=(A)	The starting angle（the angle with X axis）
#2=(B)	Angle interval between holes
#4=(I)	Radius of circular
#9=(F)	Cutting feed speed
#11=(H)	Hole number_
#18=(R)	Coordinate of R in canned cycle
#24=(X)	X coordinate value of the circle center
#25=(Y)	Y coordinate value of the circle center
#26=(Z)	Hole depth (Z coordinate value)

Commands of macro program call:

S1000 M03;
G54 G90 G00 X0 Y0 Z30;
G65 P200 X50 Y50 Z-10 R5 F200 A23 B45 I20 H4;
M30;

The body of macro program is as follows(under the circumstances of absolute programming):

O200	
#3=1	Set hole order number to 1 (from the 1st hole)
WHILE[#3LE#11]D01	If #3 (hole order number) is smaller than or equal to #11 (hole number), cycle 1 continues.
#5=#1+[#3-1]*#2	Degree of the angle relatives to hole #3
#6=#24+#4*COS[#5]	X coordinate value of the #3 hole center
#7=#25+#4*SIN[#5]	Y coordinate value of the #3 hole center
G98G81X#6Y#7Z#26R#18F#9	Processing hole #3 with mode G81
#3=#3+1	Hole order number #3 increases 1
END1	Cycle 1 ends
G80	Canceling of canned cycle
M99	Macro program ends and returns

Example 3: Unequal interval oblique line

The position set by the macro program which is set by the datum position, is taken as the datum position, and it is arranged in the unequal intervals（1, 12……）in the direction of angle a which is formed by the hole edge and X axis,

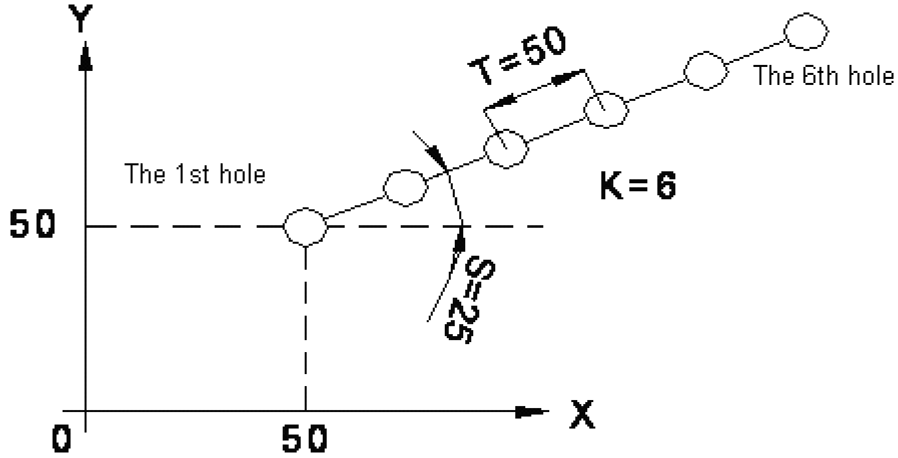


Fig. 3-7-16

The coordinate values of X0 and Y0 reference positions

S angle

T The interval of the holes

K The number of the holes is continuously set by the equal interval

The macro program calling commands

G65 P_X_Y_Z_R_ A_I_K_F_；

When K=1, K can be omitted.

S1000 M03

G54 G90 G00 X0 Y0 Z30

G65 P203 U50 V50 Z-10 R1 F200 S25 T50 K6；

M30

The following variable can be used:

#19(S)	The angle with X axis
#20 (T）	Interval between holes
#6(K)	The number of holes is continuously set by the equal interval
#9(F)	Cutting feed speed
#18(R)	Coordinate of point R in canned cycle
#21=(U)	X coordinate value of the starting point
#22=(V)	Y coordinate value of the starting point
#26=(Z)	Hole depth (Z coordinate value)

The body of macro program is as follows(under the circumstances of absolute programming):

```
0203；
#8=1；
WHILE[#8 LE #6]D02；                 ：    Restrict the hole number to K
#24=#21+ #20*[#8-1]* COS[#1]；       ：    X coordinate of the hole
#25=#22+#20*[#8-1]*SIN[#1]；         ：    X coordinate of the hole
#8=#8+1；                            ：    Hole number increase 1
```

G98G81X#24Y#25Z#26R#18F#9 In example 2 combined holes of 3.7.6.3, this block shall be changed to G65P200)

END2；

G80

M99；

3.7.7.2 Modal Call (G66)

Modal call can be specified when the following command is executed. Call specified macro program each time to execute a movement command when macro call mode is performed.

G66 P（program no.）L（repetition count）＜argument designation＞;

＜argument designation＞is identical with its function in simple call.

G67 Macro call cancellation

Note:

·Program number of modal call is specified by address P after G66.

·When a number of repetition is required, a number from 1 to 999 can be specified at address L.

·Identical with non-modal call (G65), data specified by argument is passed to macro program.

·In modal call G66, subprogram or macro program is not allowed to be called, otherwise, alarm occurs.

·In G66 block, same macro program can not be called with G65, otherwise, alarm occurs.

·Local variable (argument) is only specified in block G66. Note: local variable setting is not required when executing modal call.

Sample 1 Drilling cycle

Drilling cycle is performed at each poisoning point.

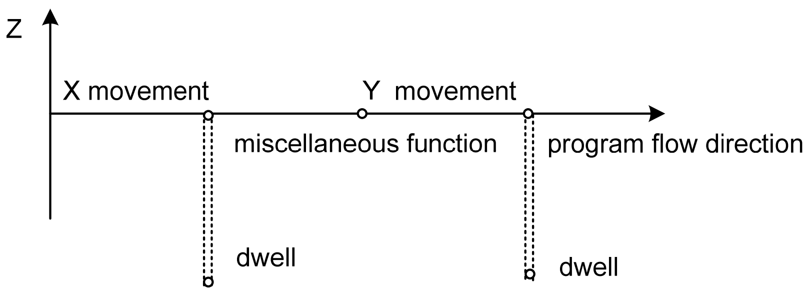


Fig. 3-7-17

G66 P9082 R（point R）X（point Z）X（dwell time）;

X ;
M ;
Y Drilling cycle is performed by a certain move program in this area.
⋮
G67 ;

Macro format is as follows (in incremental programming):

```
G9082;
G00 Z#18;
G01 Z#26;
G04 X#24;
G00 Z-[ROUND[#18]+ROUND[#26]];
M99;
```

3.7.7.5 Macro call using G code

G code used to call macro program is set by parameter. Macro program can be called. The calling method is the same as G65.

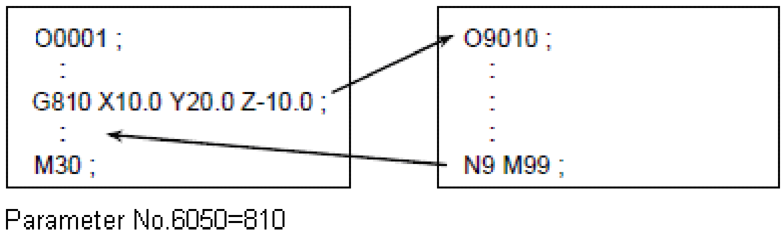


Fig. 3-7-18

Detailed specifications are as follows:

1. Called G code ×× and called program number △△△△ are set in parameter.

2. Except for standard using G functions, at most 10 can be selected between G01 to G999 to call macro program.

Set following parameters:

Program number	Parameter number
O9010	6050
O9011	6051
O9012	6052
O9013	6053
O9014	6054
O9015	6055
O9016	6056
O9017	6057
O9018	6058
O9019	6059

Fig. 3-7-19

Example 1: CW arc machining using G200

G200　I（Radius）D（offset number）;

（1）Set following Parameters

Macro program:9010call G code =200

（2）Record the following macro program.

09010;

```
#1-ABS[#4]-#[2000+#7];
IF[#1 LE 0]GOTO 1;
#2=#1/2;
#3003=3;
G01 X[#1-ROUND[#2]]Y#2;
G17 G02 X#2 Y-#2R-#2;
I-#1;
X-#2 Y-#2 R#2;
G01 X[#-ROUND[#2]]Y#2];
#3003=0;
N1 M99;
```

3.7.7.7 Call Subprogram by M Code

By setting an M code used to call a macro program in parameter, macro program can be called. The calling method is the same as G65.

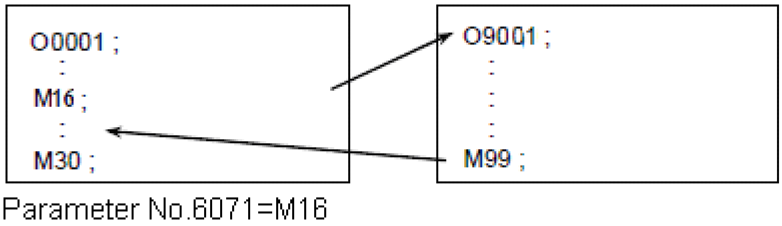


Fig. 3-7-20

Detailed instructions are as follows:

1. Correspondence between M×× code call subprogram and program numbers △△△△ is set by parameter.

The command of N__G__X__Y__……M98P△△△△ can be replaced by following simple command.

Parameter No.	Program No.
6071	O9001
6072	O9002
6073	O9003
6074	O9004
6075	O9005
6076	O9006
6077	O9007
6078	O9008
6079	O9009

Fig. 3-7-21

Example 1:

ATC tool changing by M16

Set the following parameters: No6071=16

Call subprogram O9001 by M16 code.

3.7.7.8 Subprogram Call Using a T Code

Subprogram or macro program can be called with a T code in parameter. Macro program is called once when T code appears one time on machining.

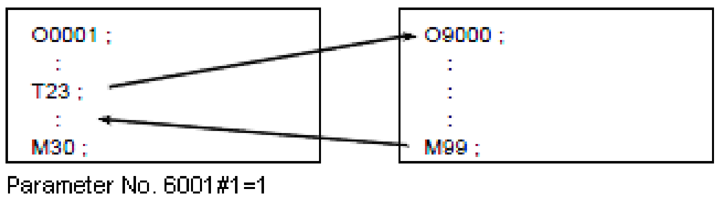


Fig. 3-7-22

Set the following parameters:

No6001#0=1 Subprogram O9000 program is called by T code.

The function of T code executing subprogram call is the same as M98 P9000.

3.7.8 Limitations

（1）Available variables

#0, #1～#33, #100～#199, #500～#999, system variables.

（2）Available variable values

Max.: $\pm 10^{47}$,　Min.:　$\pm 10^{-39}$

（3）< expression formula>using rating data

Max.:±99999999,　Min.: ±0.0000001

decimal: available

（4）Calculation precision　decimal system 8 digits

（5）Macro program call nest degree　Max. 4 levels

（6）Repetition identification sign　1～3

（7）[　] nest　Max. 5 levels

（8）subprogram call nest degree　Max. 4 levels

3.7.9 Example of Customer Macro Call

3.7.9.1 Grooving

Groove canned cycle is performed by customer macro call in the following drawing.

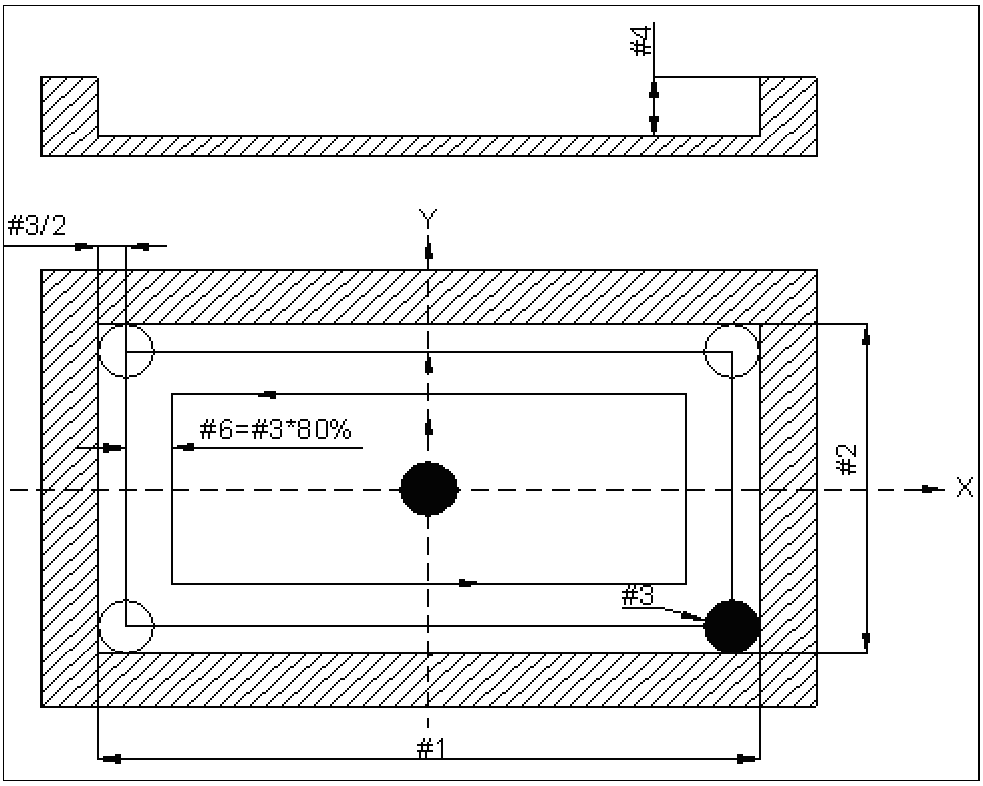


Fig. 3-7-23

```
O220
#1=300
#2=200
#3=10
#4=30
#5=0
#17=2
#6=0.8*#3
#7=#1- #3
#8=#2- #3
S800 M03
G54X0Y0Z30
WHILE[#5LT#4]D01
Z[-#5+1]
G01Z- [#5+ #17]F150
IF[#1GE#2]GOTO1
N1 #9=FIX[#8/#6]
IF[#1GE#2]GOTO3
IF[#1LT#2]GOTO2
N2 #9=FIX[#7/#6]
IF[#1LT#2]GOTO3
N3 #10=FIX[#9/2]
WHILE[#10GE0]D02
#11=#7/2- #10*#6
#12=#8/2- #10*#6
Y#12F1000
X- #11
Y- #12
X#11
Y#12
X0
#10=#10-1
END2
G00Z30
X0Y0
#5=#5+#17
END1
M30
```

3.7.9.2 Plane Processing

See the figure below for plane processing.

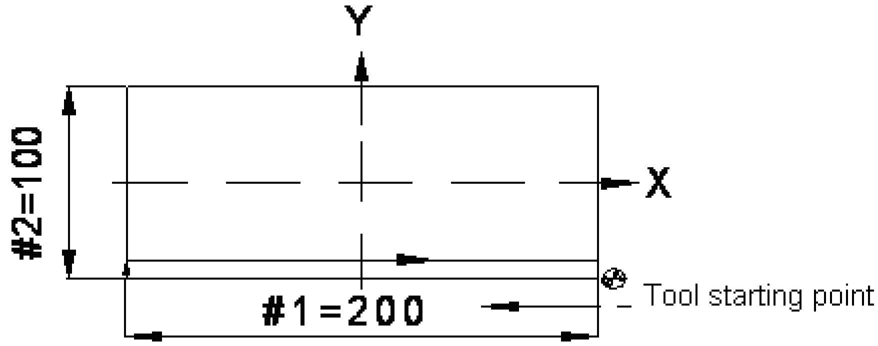


Fig. 3-7-24

```
O0300
#1=200                        Plane length in X direction
#2=100                        Plane length in Y direction
#3=10                        （Flat-end milling cutter）Tool diameter
#4= - #2/2                    Setting Y coordinates as argument, initial value: - #2/2
#14=0.8*#3                    Variable increasing value of each time, namely, step pitch
#9=- #6+1
#5=[#3+#1]/2+6                X coordinate of the starting point
#6=0.5                        Argument of dz, initial value: 0.5
#16= 2                        Processing depth of each layer
#7=20.5                       Required processing depth
G0G90G54X0Y0Z50
S1000 M03
WHILE[#6LE#7]D01              If predefined depth is not reached, continuous circulation 1 remains.
#4=- #2/2                     Initializing value #4 once a layer is processed
G0 X - #5    Y#4              Moving to the start point rapidly
Z#9                           Moving 1mm above Z- #6
G01Z- #6 F150                 Z- #6 cutting of the first layer
WHILE [ #4 LT[-#2/2+ 0.3*#3 ] D02      If the tool does not reach to the upper edge, continuous 2
remains
G01X #5F1500                           G01 moves to the left
#4=#4+ #14                             #4 variable of Y coordinate increases #14
Y#4                                    Y axis moves #14 in positive direction
X- #5                                  Moving to the right
#4=#4+ #14                             #4 variable of Y coordinate increases #14
Y#4                                    Y axis moves #14 in positive direction
END2                                   Cycle 2 statement ends
G0 Z30
#9=- #6+1                              Lift the tool to a safety height
#6=#6+#16                              The depth Z increases #16
END1                                   Cycle 1 statement ends
G80
M30                                    Program ends
```

3.8 Feed G Code

3.8.1 Feed Mode G64/G61/G63

Format:

Exact stop mode G61
Tapping mode G63
Cutting mode G64

Function:

Once G61 specified, this function is valid until G62, G63 or G64 is specified. Tool is decelerated at the end point of a block, and then an in-position check is made. Then the next block is executed.

Once G63 specified, this function is valid until G61, G632 or G64 is specified. Tool is not decelerated at the end point of a block, but the next block is executed. When G63 is specified, feedrate override and feed hold are invalid.

Once G64 specified, this function is valid until G61, G632 or G63 is specified. Tool is not decelerated at the end point of a block, but the next block is executed.

<table>
<tr><th colspan="2">Function name</th><th>G code</th><th>Validity of G code</th><th>Descriptions</th></tr>
<tr><td colspan="2">Exact stop</td><td>G09</td><td>Only specified blocks are valid</td><td>Tool is decelerated at the end point of a block. The next block is executed after in-position check is made.</td></tr>
<tr><td colspan="2">Exact stop mode</td><td>G61</td><td>Once G61 is specified, the function is valid until G62, G63 or G64 is specified.</td><td>Tool is decelerated at the end point of a block. The next block is executed after in-position check is made.</td></tr>
<tr><td colspan="2">Cutting mode</td><td>G64</td><td>Once G64 is specified, the function is valid until G61, G62 or G63 is specified.</td><td>Tool is not decelerated at the end point of a block, but the next block is executed.</td></tr>
<tr><td colspan="2">Tapping mode</td><td>G63</td><td>Once G63 is specified, the function is valid until G61, G62 or G64 is specified.</td><td>When G63 is specified, single block feed dwell is invalid, federate is invalid until G61, G62 or G64 is specified.</td></tr>
<tr><td rowspan="2">Automatic corner override</td><td>Inner corner override</td><td>G62</td><td>Once 62 is specified, the function is valid until G61, G63 or G64 is specified.</td><td>In tool radius compensation, in order to get a polished machined surface, override is used in cutting federate to ensure the cutting value per unit of time will not increase.</td></tr>
<tr><td>Speed change of the inner arc</td><td>_</td><td>It is valid in tool radius compensation mode and is irrelevant with G code.</td><td>Changing the cutting speed of the inner arc.</td></tr>
</table>

Explanations:

No parameter format.

G64 is default mode of the system, program is not decelerated at the end point of a block, but the next block is executed directly.

The purpose of in-position check in exact stop mode is to check that the servo motor has reached within a specified range.

In exact stop mode, move path of cutting mode and tapping mode are different.

For details, please refer to following figure 3-38.

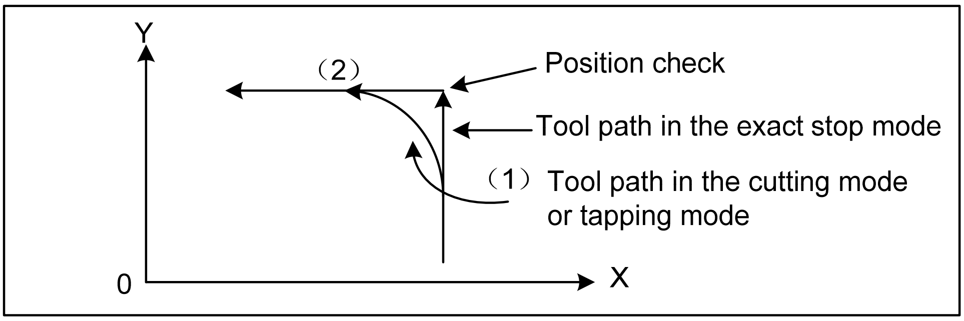


Fig. 3-8-1 Tool paths from block 1 to block 2

3.8.2 Automatic Corner override（G62）

Format: G62

Function: Once G62 specified, this function is valid until G61, G63 or G64 is specified. When tool radius compensation is performed; the movement of the tool is decelerated at an inner corner. This reduces cutting amount at unit time, and produces a smoothly machined surface.

Explanations:

1. When tool radius compensation is performed, the movement of the tool is automatically decelerated at an inner corner and internal circular area. This reduces the load on cutter and produces a smoothly machined surface.

2. When G62 is specified, and the tool radius compensation applied forms an inner corner, the feedrate is automatically adjusted at both ends of the corner. There are 4 types of inner corners (Fig.3-39). In figure: 2°≤θ≤θp≤178°.

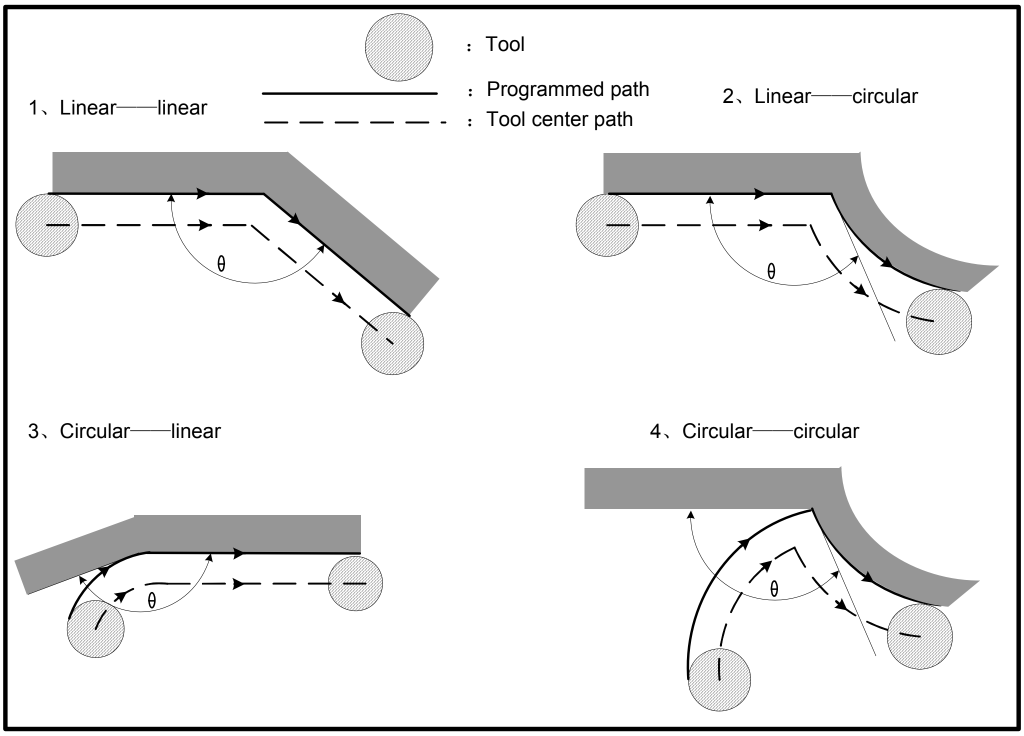


Fig. 3-8-2 4 types of inner corners

3. When a corner is determined to be an inner corner, the feedrate is overridden before and after the inner corner. The distances Ls and Le, where the feedrate is overridden, are distance from points on cutter center path to the corner. Fig. 3-40 Ls+Le≤2mm

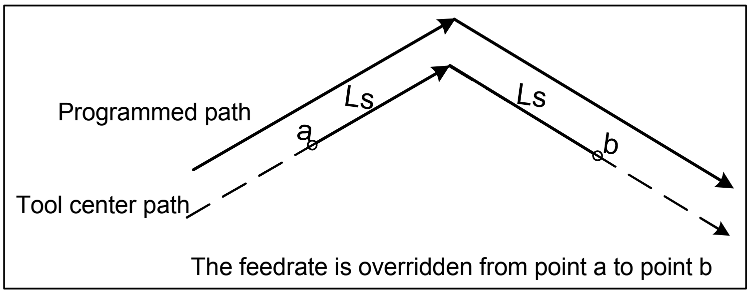


Fig. 3-8-3 straight to straight line

4. When a programmed path consists of two arcs, the feedrate is overridden if the start and end points are in the same quadrant or in adjacent quadrants (Fig. 3-8-4)

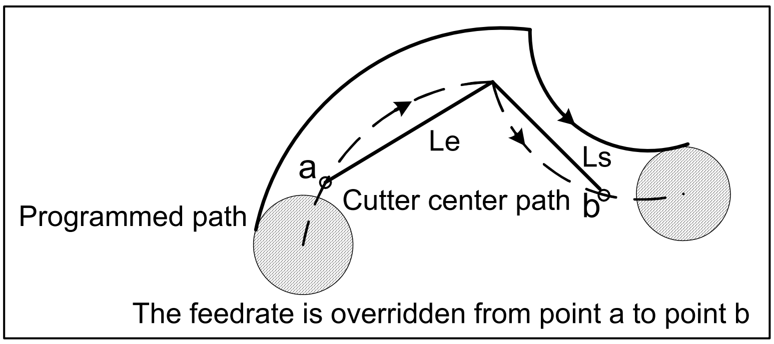


Fig. 3-8-4 arc-arc

5. Regarding program types are straight line-straight line and arc-arc, the feedrate is overridden from point a to point b and from point c to point d. (fig.3-42)

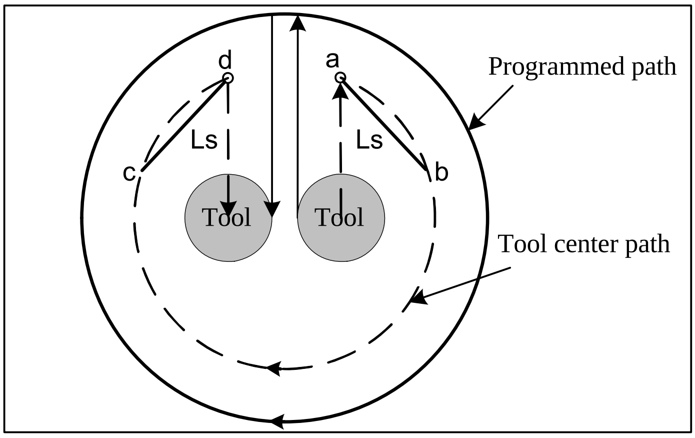


Fig. 3-8-5 straight line-arc, arc- straight line

Restrictions:

1. Override for inner corners is invalid during acceleration/deceleration before interpolation.
2. Override for inner corners is invalid if the corner is preceded by a start-up block or followed by a block including G41 or G42.
3. Override for inner corners is not performed if the offset is zero.

3.9 Introduction of Five Axes Control

3.9.1 Tool Center Point (TCP) Control

TCP control format:

G43.4 IP_α_β_ H ;

IP_α_β_ ;

TCP control cancel format:

G49 IP_α_β_;

IP: In absolute command mode, the end point coordinates
In incremental command mode, the move distance of the TCP

α, β : In absolute command mode, the end point coordinates of rotary axes
In incremental command, the amount of movement of the rotary axes

H: Tool offset number

When the CNC executes rotation interpolation, it controls the control point so that the TCP moves linearly toward the worktable (workpiece). The end point of TCP path is the coordinate in the program coordinate system.

Function:

This function is intended to perform machining on such 5-axis machines which have rotary axes that turn a tool or table as well as three orthogonal axes (X, Y, and Z axes) by accomplishing tool length compensation while changing the position of the tool. Even when the direction (cutter to workpiece) is changed, the TCP still moves along the specified path.

A coordinate system used for programming the TCP control is called the programming coordinate system. The coordinate system that fixed on the worktable is used as programming coordinate system, which makes CAM programming easy.

There are three types of 5-axis machine tool: ①the one that rotates the tool only; ②the one that rotates the table only; ③the one that rotates both the tool and table.

This function is applied in the 5-axis machine tool including X, Y, Z three ortho-axes and cutter rotary axis and worktable rotary axis.

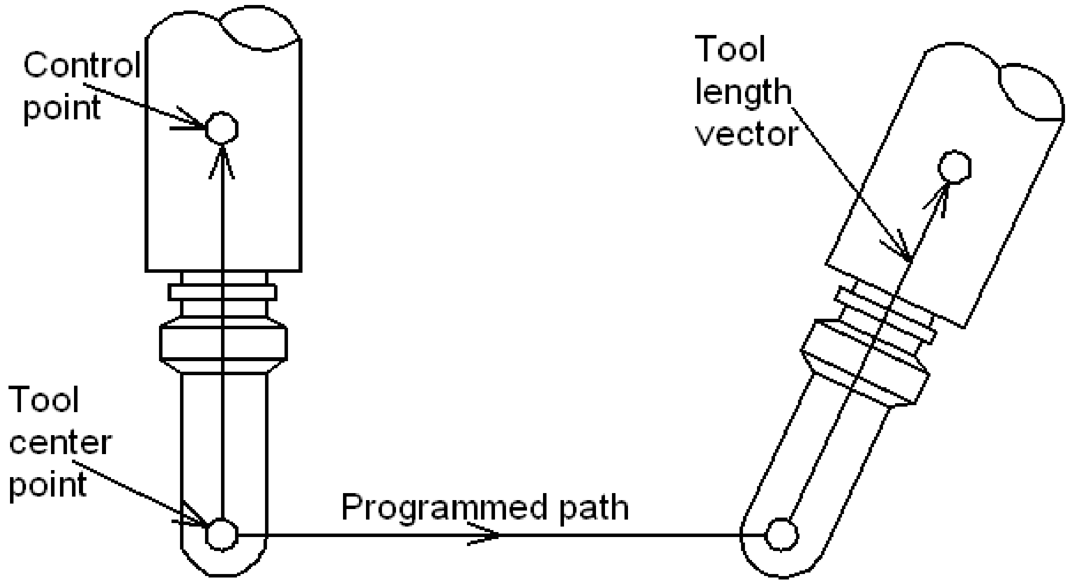


Fig. 3-9-1

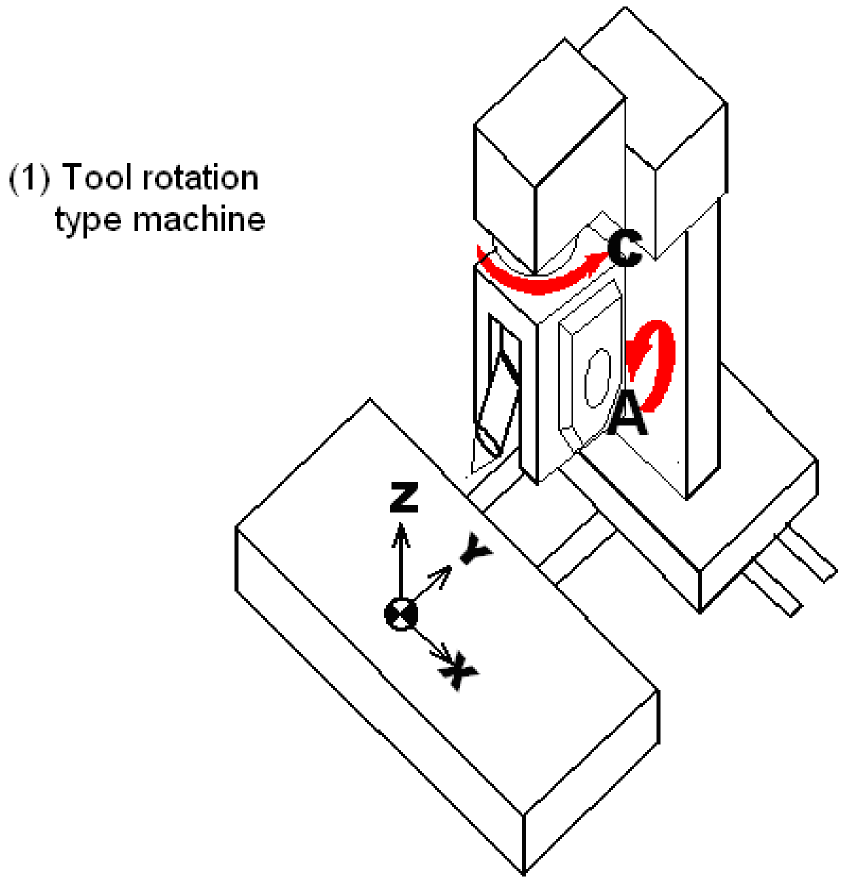


Fig. 3-9-2

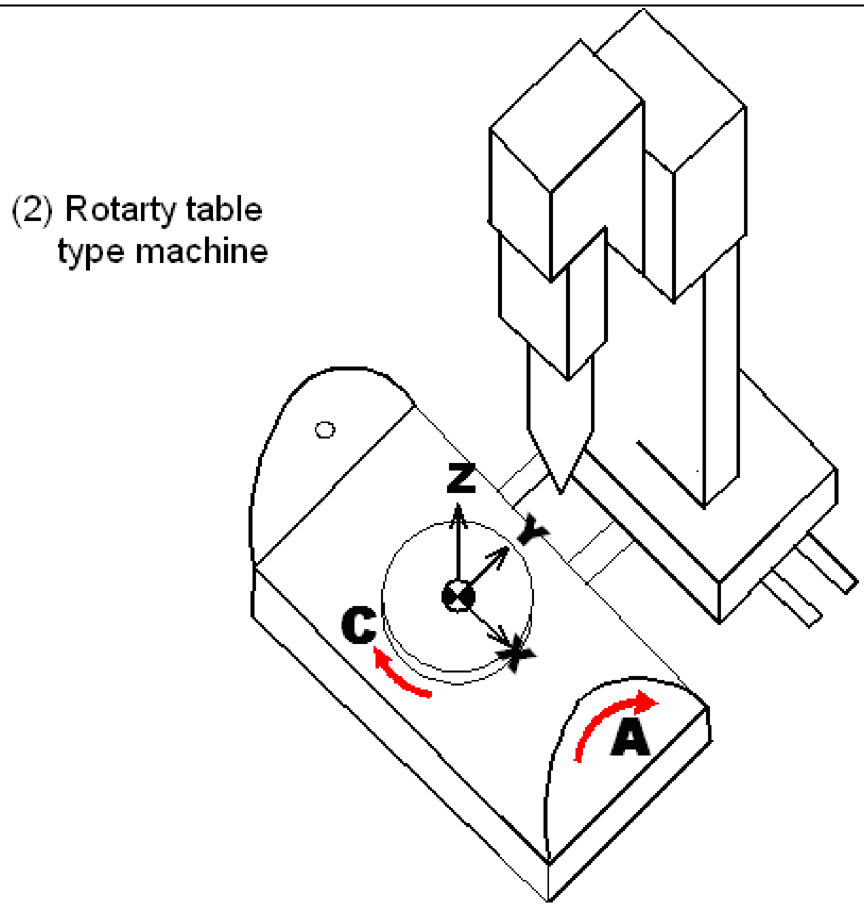


Fig. 3-9-3

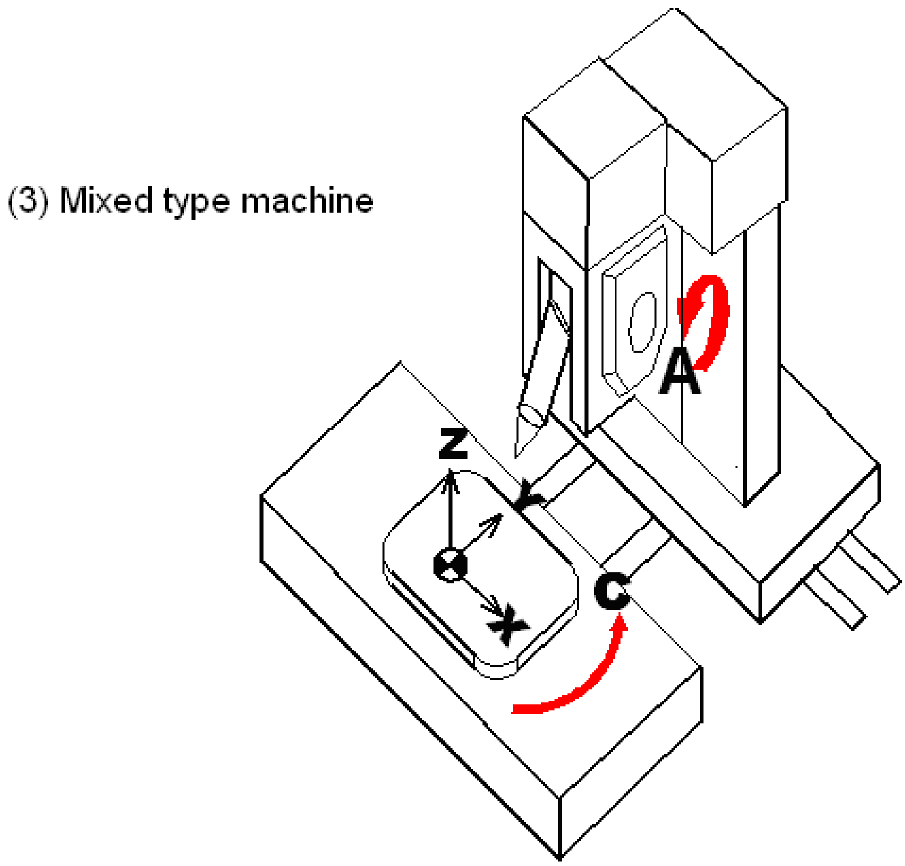


Fig. 3-9-4

When the coordinate system fixed on the worktable is taken as the programming coordinate system, a program can run without considering the rotation worktable, because as the worktable rotates, the position and direction of workpiece are changed at the same time, i.e. when a straight line is specified, the TCP moves along a straight path with respect to the workpiece as instructed.

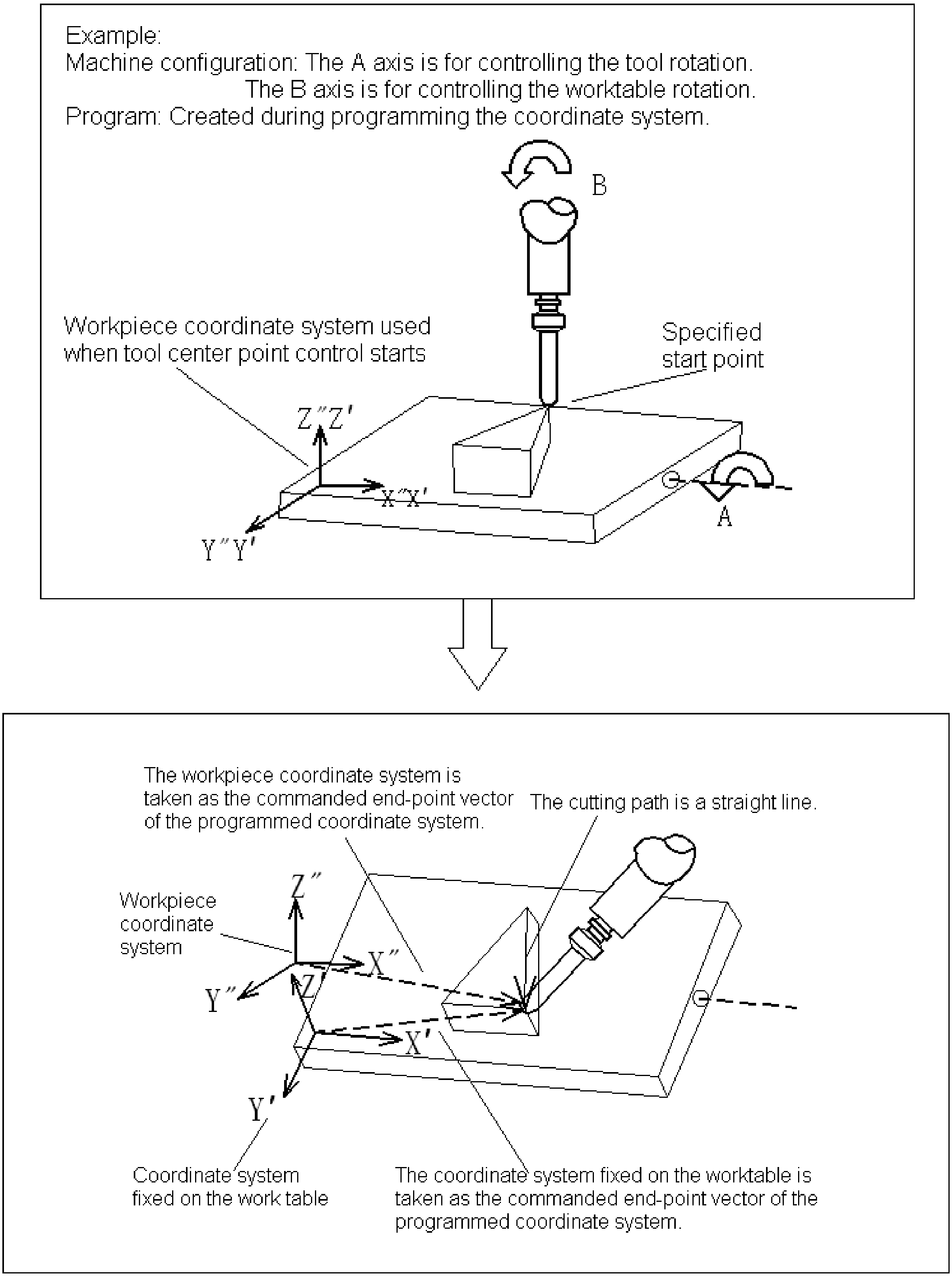


Fig. 3-9-5

Explanations:

- Programmed coordinate system

Program coordinate system is the coordinate system performs TCP control. Command G43.4 specifies the coordinate system fixed on the worktable as the programming coordinate system. When the coordinate value of rotary axis is zero, the program coordinate system and workpiece coordinate system are coincident. Thereafter, the program coordinate system rotates with the worktable rather than the tool. The X, Y, Z commands behind G43.4 are regarded as coordinates within the programming coordinate system. When G43.4 is commanded, the standard status of programming coordinate system of the worktable rotary axis is set by offset commands (G54-G59).

In the following introduction, X’ Y’ Z’ represent the coordinate system fixed on worktable.

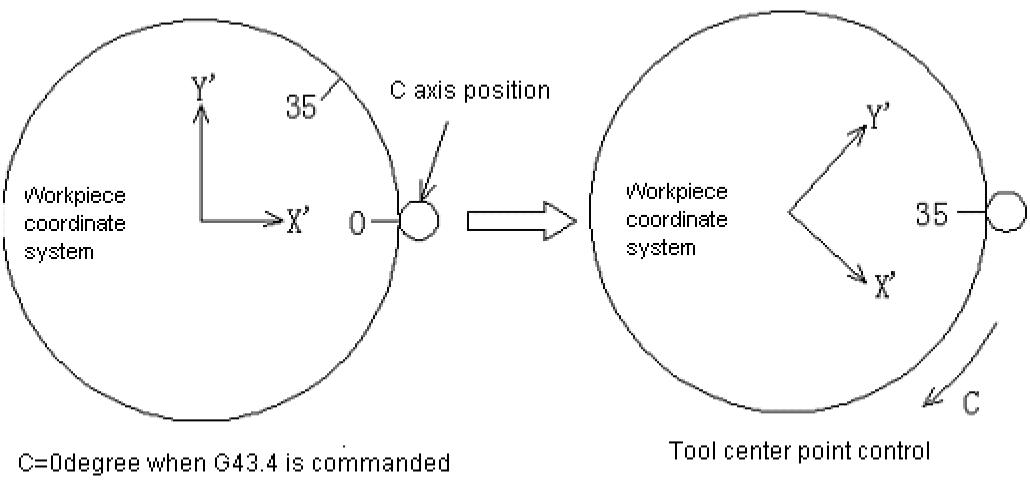


Fig. 3-9-6

- Display the current position during TCP control

The current position of control point in the workpiece coordinate system is displayed in the machine coordinate system during TCP control.

- TCP control command

When TCP control is in use, it specifies the position of TCP at the end of blocks which can be seen in the program coordinate system.
For rotary axis, it specifies coordinate value of blocks ends.
Besides, for feedrate, the F command specifies the tangential speed with respect to workpiece (the relative speed between workpiece and tool).
.

- Commands available during TCP control

Commands available during TCP control are linear interpolation (G01), positioning (G00).

When specifying linear interpolation (G01), a specified speed is performed on TCP through speed control function.

- Reset operation during TCP control

Resetting in G43.4 modal status will disable the status. (the same effect as G49 is executed).

- Mode switching

After the TCP control is enabled, switching the modes will disable the modal status (the same effect as G49 is executed). If TCP control is activated in AUTO and MDI mode, the switching operation after entering into G43.4 status is to cancel the TCP control. If it is needed to switch back to AUTO and

MDI mode, the operation can only be done after re-entering into G43.4 status, otherwise, machining error and danger will occur.

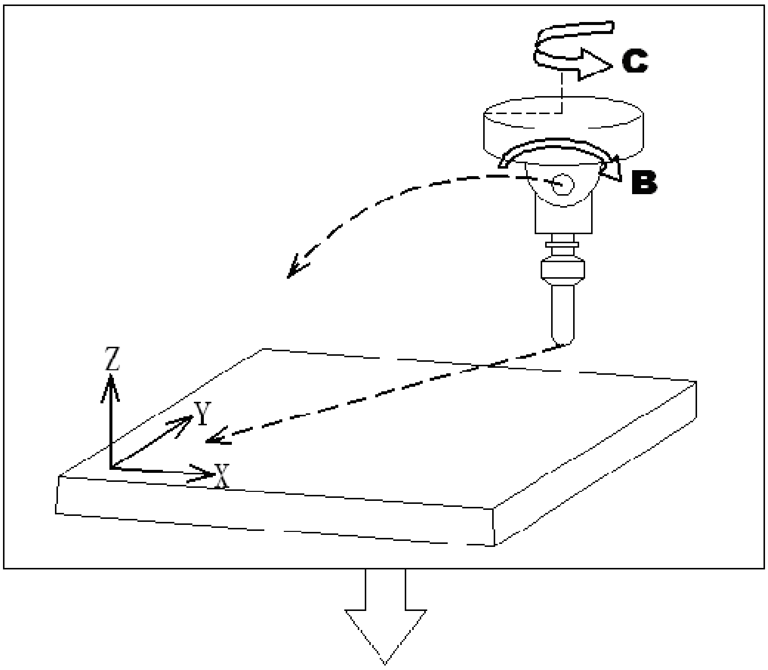
Example:

Tool rotation type machine

When the workpiece coordinate system is taken as programming coordinate system and linear interpolation is specified on X, Y, Z axes, the CNC can control the TCP to move linearly towards the worktable (workpiece) while tool rotation is being performed. Through speed control, the TCP moves towards the worktable (workpiece) at a specified speed.

For this type of machine, when the tool rotation axis rotates, the worktable does not rotate with respect to the workpiece coordinate system, so the programming coordinate system always coincides with the workpiece coordinate system.

N1 G00 G90 B0 C0 ;
N2 G54 ; prepare program coordinate system
N3 G43.4 H01 ; TCP control starts, H01 is the tool compensation number
N4 G00 X200.0 Y150.0 Z20.0 ; move towards end point
N5 G01 X5.0 Y5.0 Z5.0 C60.0 B45.0 F500 ; linear interpolation
N6 G49; cancel TCP control
N7 M30;



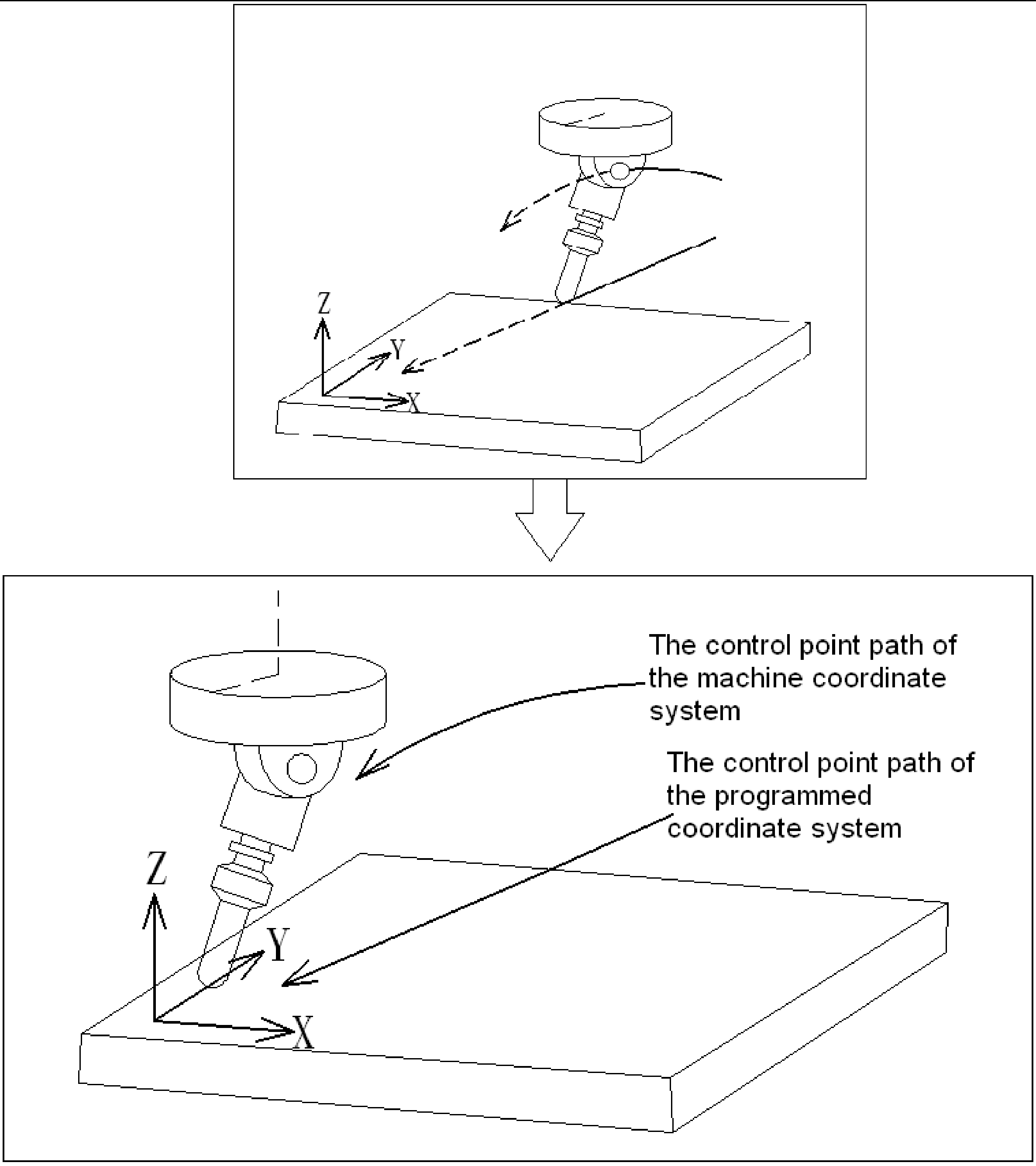


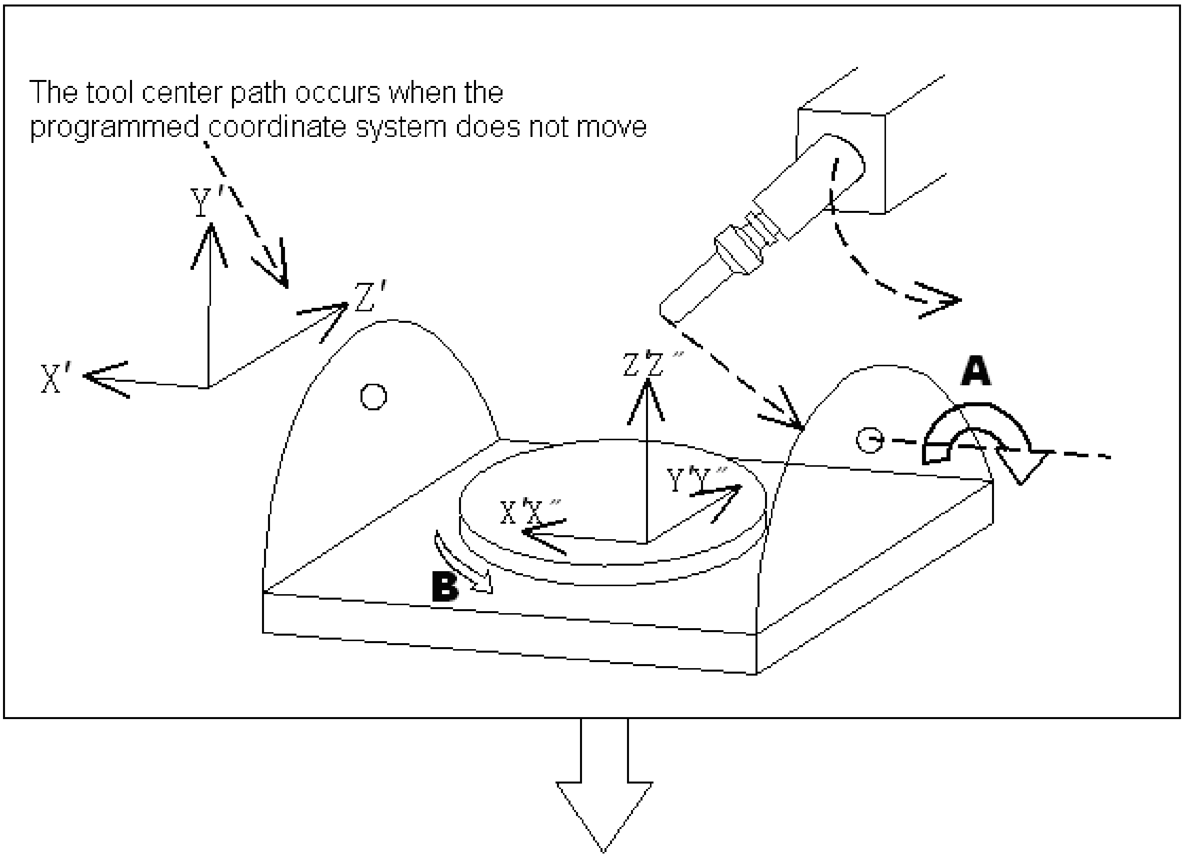
Fig. 3-9-7

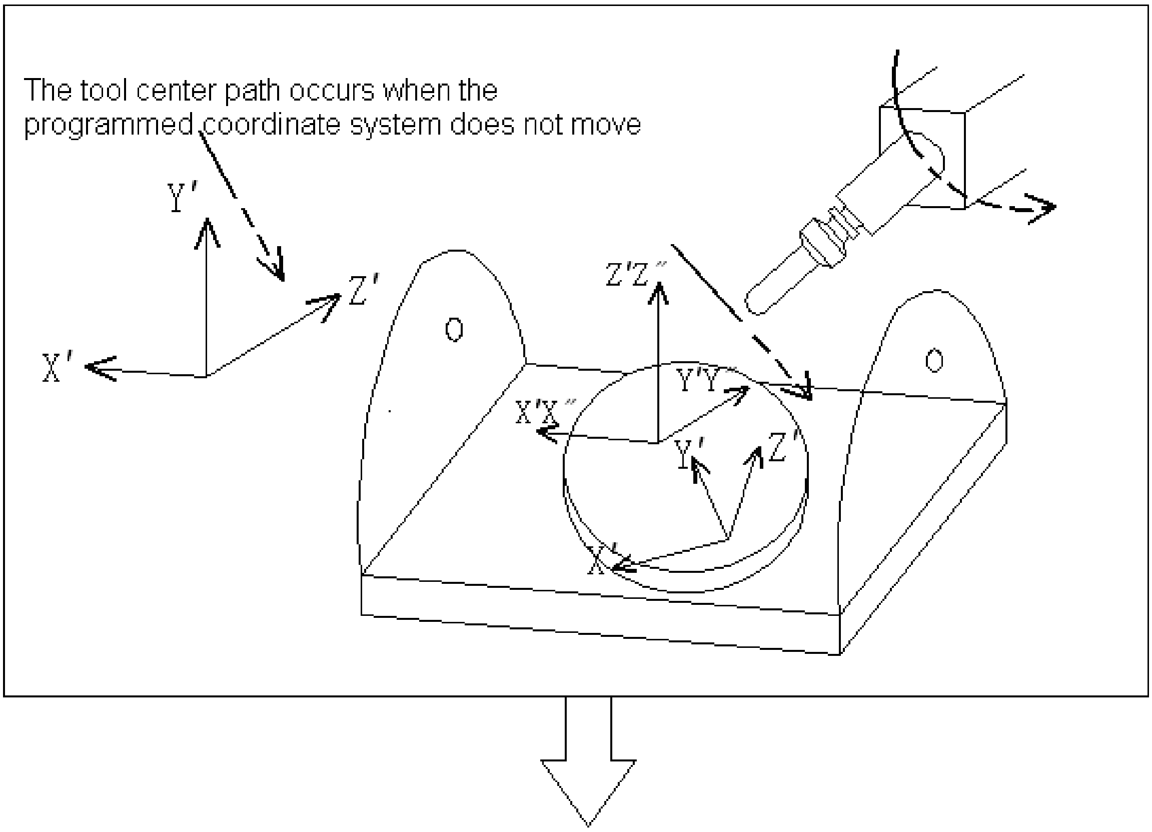
Rotary table type machine

Specify linear interpolation on X, Y, Z axes in the programming coordinate system, the CNC can control the TCP to move linearly towards the worktable (workpiece) while worktable rotation is being performed. The TCP moves towards the worktable (workpiece) at a specified speed.

For this type machine, the rotation of any rotary axis enables the worktable rotation, meanwhile, the workpiece coordinate system does not change, but the programming coordinate system which is fixed on the worktable rotates with it.

N1 G00 G90 A0 B0;
N2 G54; prepare program coordinate system
N3 G43.4 H01; TCP control starts, H01 is the tool compensation number
N4 G00 X20.0 Y100.0 Z0 ; move towards start point
N5 G01 X10.0 Y20.0 Z30.0 A60.0 B45.0 F500 ; linear interpolation
N6 G49; cancel TCP control
N7 M30;


The tool center path occurs when the
programmed coordinate system does not move
Y′
Z′
X′
Z′Z″
A
Y′Y″
X′X″
B


The tool center path occurs when the
programmed coordinate system does not move
Y′
Z′
X′
Z′Z″
Y′Y″
X′X″
Y′
Z′
X′

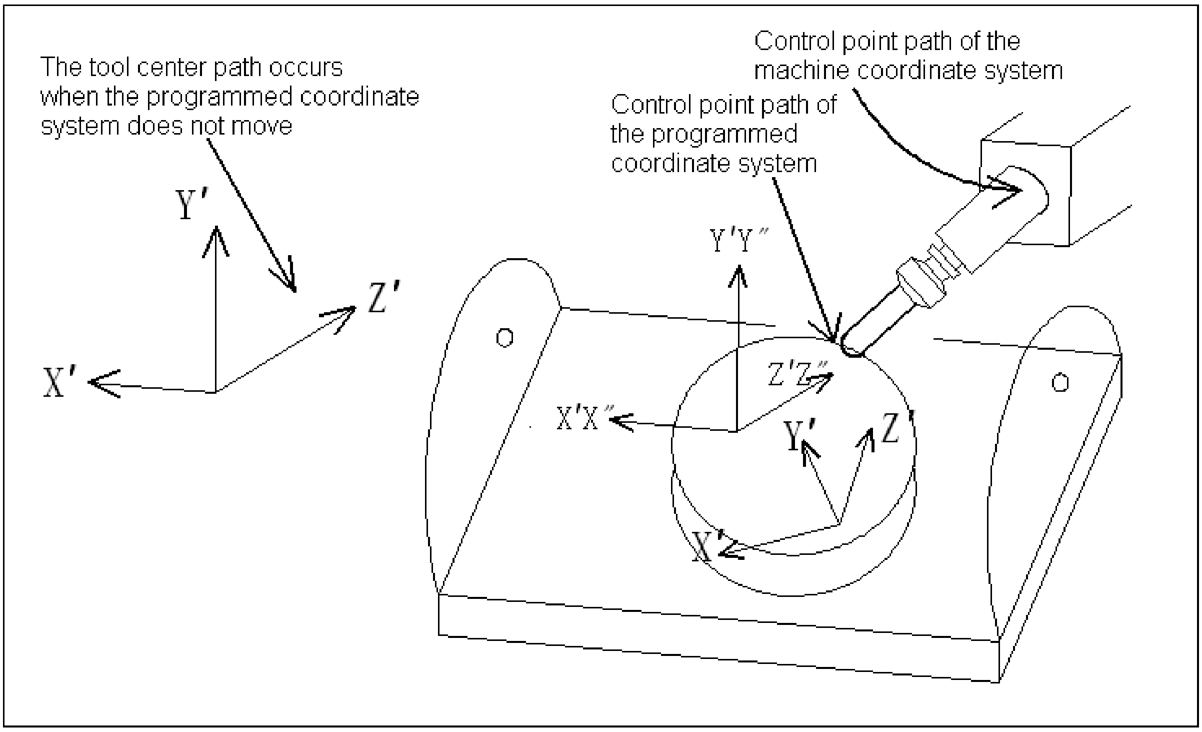


Fig. 3-9-8

Mixed type machine

Specify linear interpolation on X, Y, Z axes in the programming coordinate system, the CNC can control the TCP to move linearly towards the worktable (workpiece) while cutter rotation and worktable rotation is being performed.

For this type of machine, the rotation of worktable rotary axis instead of tool rotary axis enables the rotation of worktable, meanwhile, the workpiece coordinate system does not change, and the programming coordinate system which is fixed on the worktable rotates along with it.

N1 G00 G90 A0 B0;

N2 G54; prepare program coordinate system

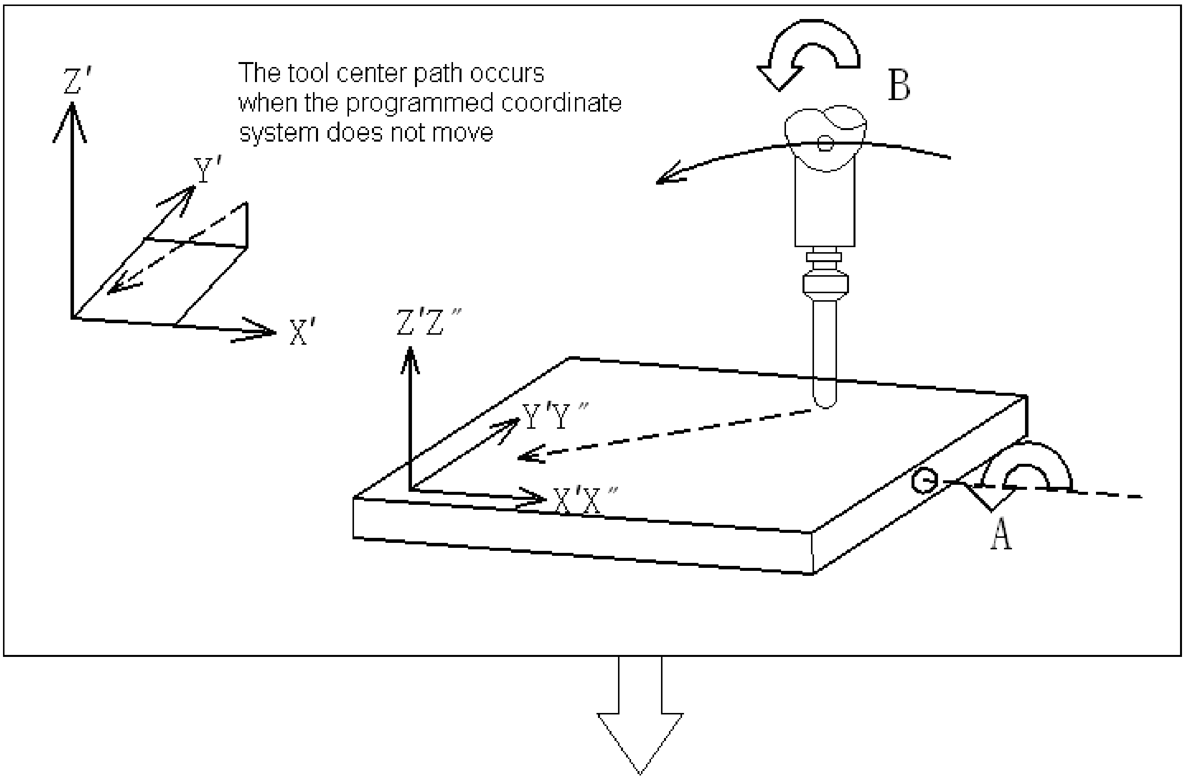
N3 G43.4 H01; TCP control starts, H01 is the tool compensation number

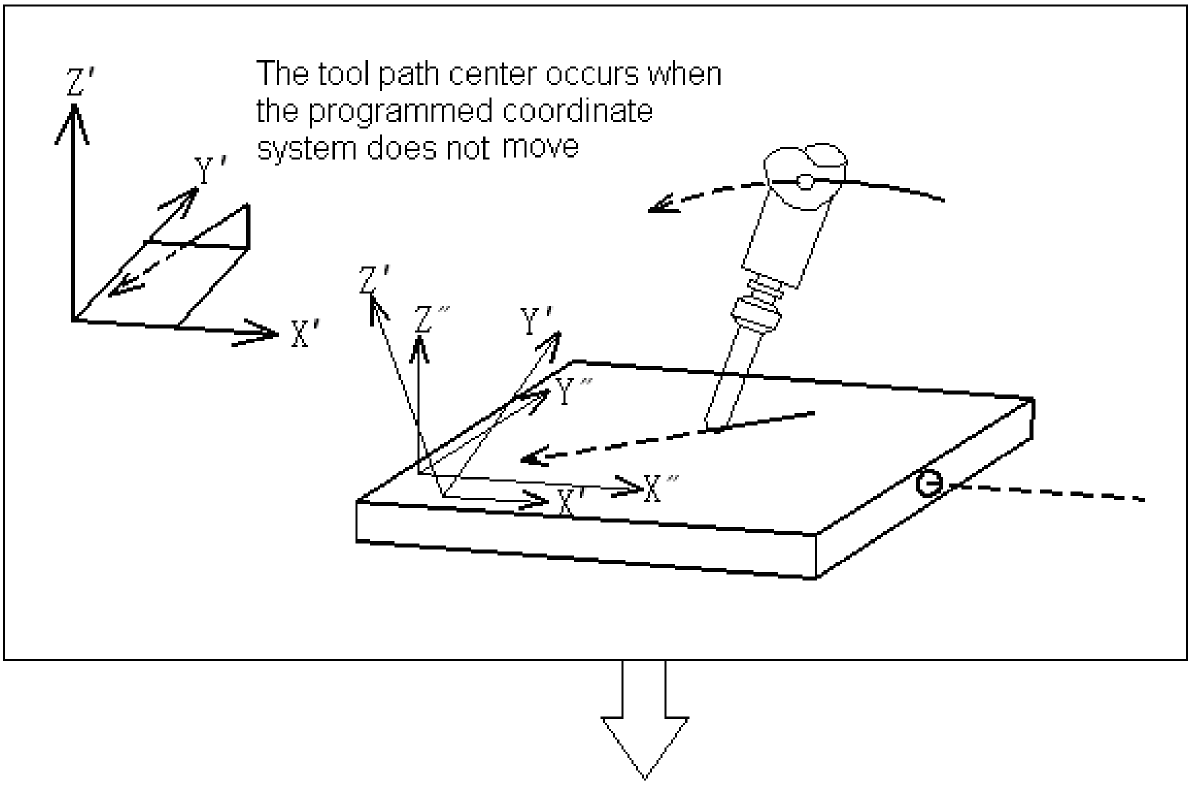
N4 G00 X20.0 Y100.0 Z0; move towards start point

N5 G01 X10.0 Y20.0 Z30.0 A60.0 B45.0 F500; linear interpolation

N6 G49; cancel TCP control

N7 M30;


Z'
The tool center path occurs
when the programmed coordinate
system does not move
B
Y'
X'
Z'Z"
Y'Y"
X'X"
A


Z'
The tool path center occurs when
the programmed coordinate
system does not move
Y'
X'
Z'
Z"
Y'
Y"
X'
X"

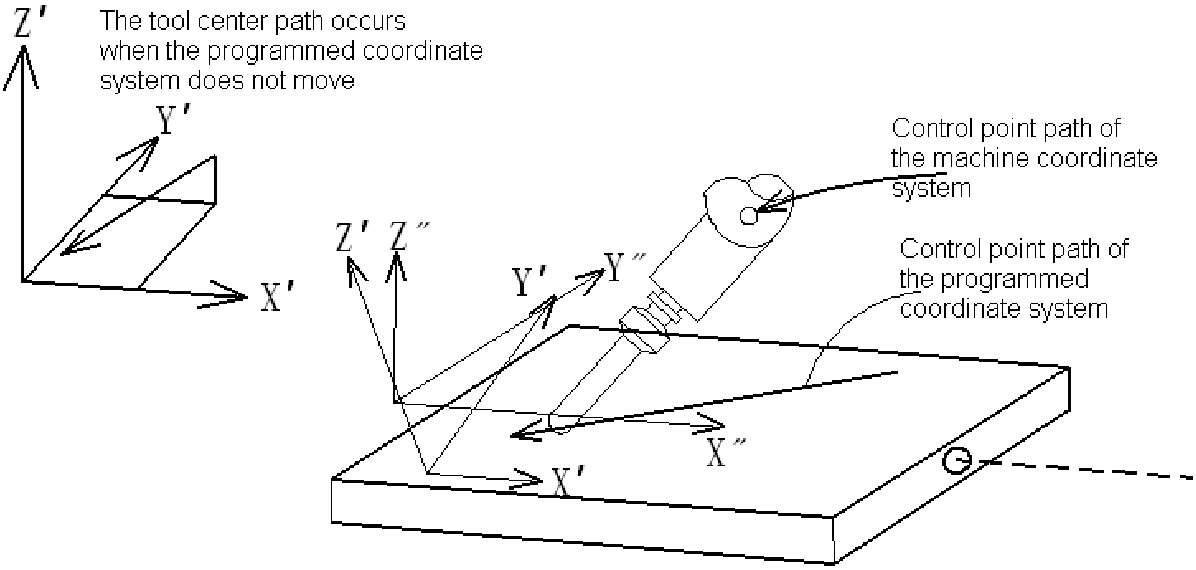


Fig. 3-9-9

Restrictions:

Manual interference

Please do not intervene manually during TCP control.

3.9.2 Tilted Working Plane Command

Programming for creating holes, pockets, and other figures in a datum plane tilted with respect to the workpiece would be easy if commands can be specified in a coordinate system fixed to this plane (called a feature coordinate system). This function enables commands to be specified in the feature coordinate system. the feature coordinate system is defined in the workpiece coordinate system.

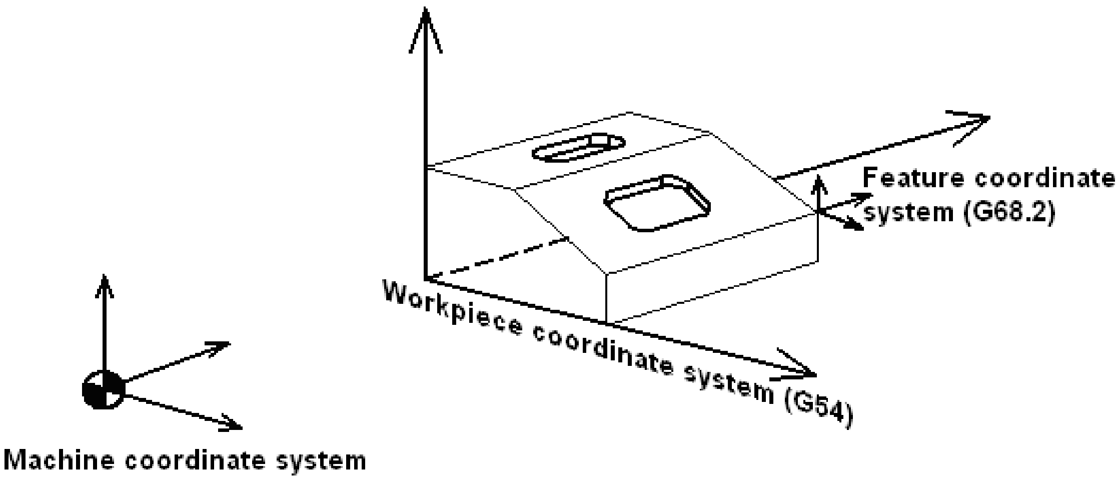


Fig. 3-9-10

Command G68.2 transfers the part machining coordinate system to “feature coordinate system”, and the coordinates in blocks subsequent are regarded as specified in the feature coordinate system, until G69 is commanded. When G68.2 specifies the relationship between feature coordinate system and workpiece coordinate system in advance and does not specify the an angle for rotary axis, command G53.1 will automatically specify the +Z direction of feature coordinate system as direction of the tool axis. The relationship is shown as follows:

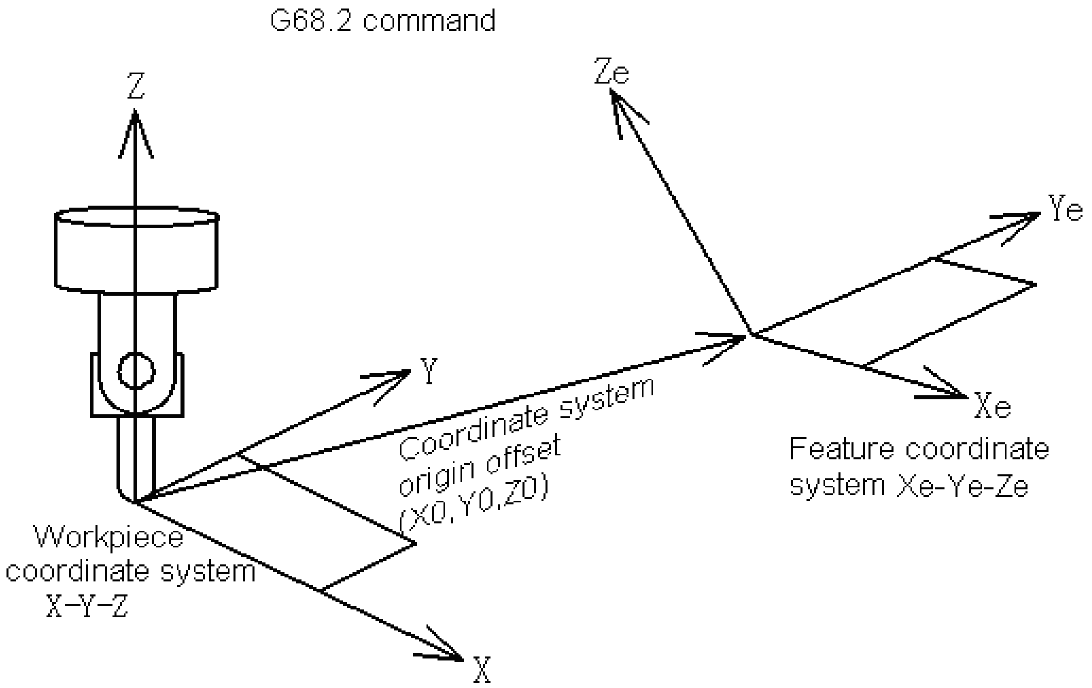


Fig. 3-9-11

The vector of tool axis is directed from tool nose to tool hilt. Shown as follow:

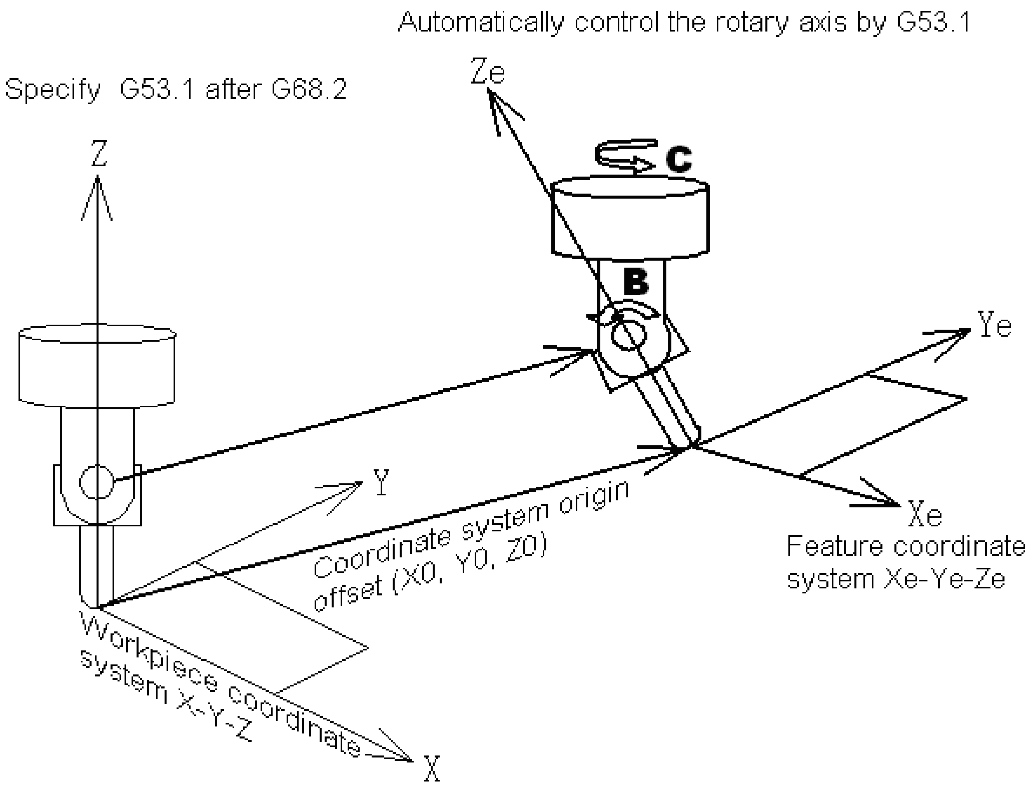


Fig. 3-9-12

Part Ⅰ Programming

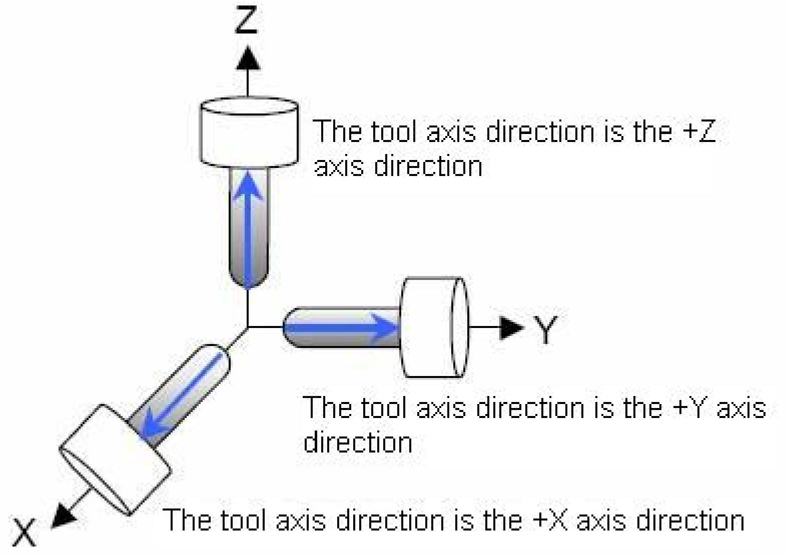


Fig. 3-9-13

This function can be applied in three types of 5-axis machine tools:

1). Tool rotation type machine—the two rotary axes control the tool.

2). Rotary table type machine—the two rotary axes control the worktable.

3). Mixed-type machine—one rotary axis controls the tool while the other rotary axis controls the worktable.

Format:

Set feature coordinate system

G68.2 X x0 Y y0 Z z0 I α J β K γ; (feature coordinate system setting)

Machining commands

G69; (cancel feature coordinate system setting)

X0, Y0, Z0 are the origin points, (absolute coordinates in workpiece coordinate system), I, J, K are Euler angle, used to specify the direction of feature coordinate system.

Tool axis direction control

G 53.1; (tool axis direction control)

Notes:

1. G53.1 should be commanded in a block after the block that contains G68.2. Otherwise, an alarm is generated if G53.1 is specified without G68.2 being specified in a preceding block.

2. G53.1 should be commanded independently.

3. Usually, when G53.1 is commanded, rotary axis moves at the specified cutting feedrate (when cutting feed) or maximum positioning speed (when positioning).

Coordinate conversion using an Euler angle

The conversion of feature coordinate system is performed as it rotates around the origin point of workpiece coordinate system. A rotation of α degree around the Z axis converts the “workpiece coordinate system” to “coordinate system 1”; a rotation of β degree around the X axis converts the “coordinate system 1” to “coordinate system 2”; a rotation (starts from “coordinate system 2”) of γ degree around the Z axis converts the origin point of workpiece coordinate system to (X0, Y0, Z0).

This coordinate system is called “feature coordinate system”. The relationship between “workpiece coordinate system” and “feature coordinate system” is as follows:

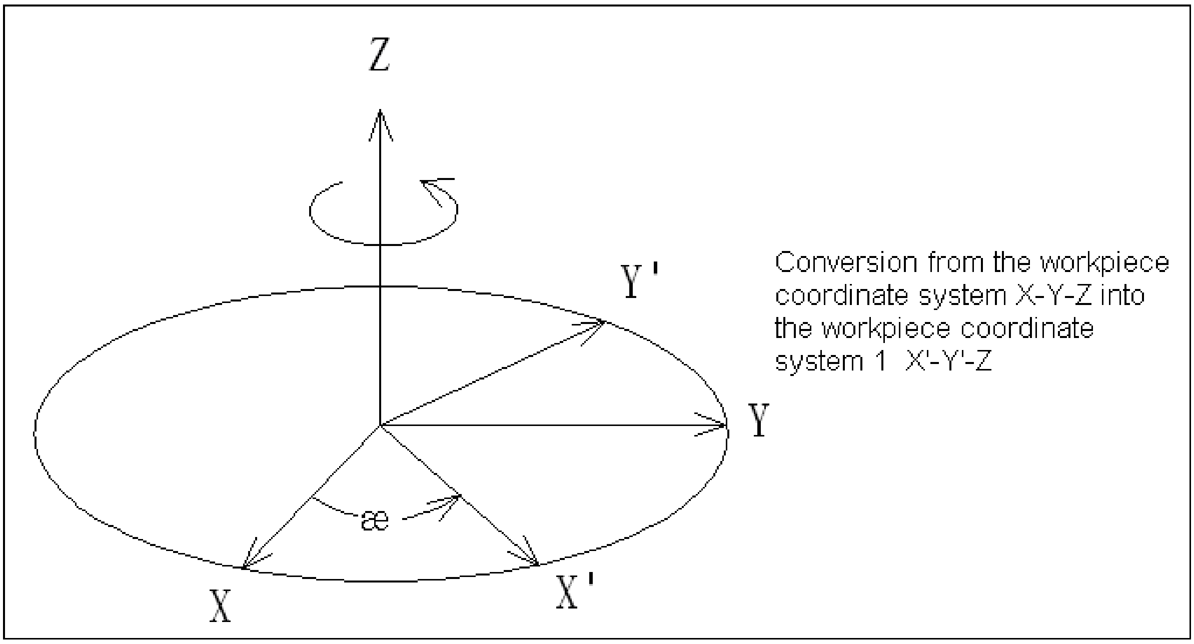


Fig. 3-9-14

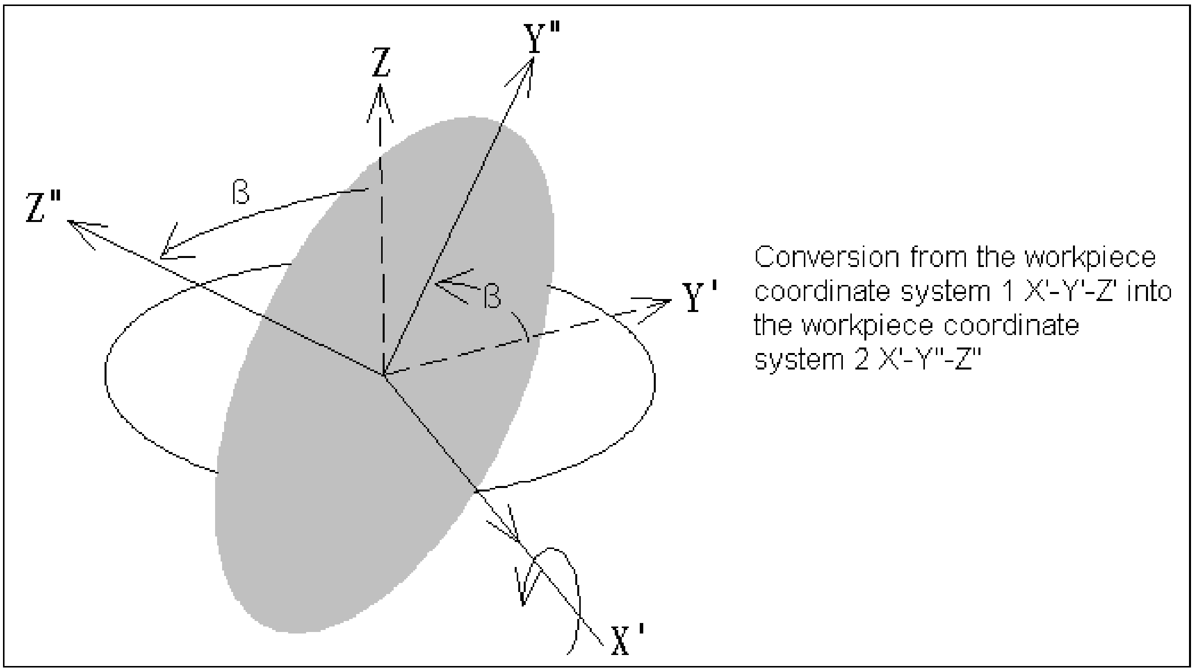


Fig. 3-9-15

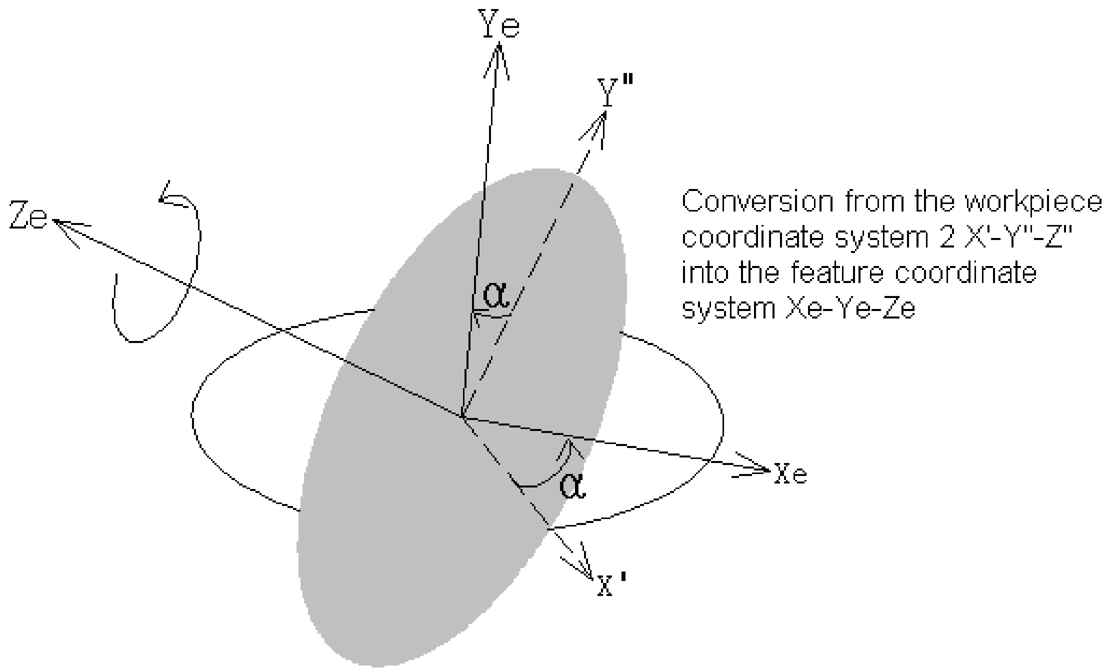


Fig. 3-9-16

Explanations:

Rotary table type machine

When this function is applied in a worktable rotation type machine with two rotary axes, the feature coordinate system Xe-Ye-Ze is set in the workpiece coordinate system based on the coordinate system origin point (X0, Y0, Z0) and the Euler angle. Command G53.1 calculates and controls the motion of the rotary axis, which converts the direction of tool axis to the +Z direction of the feature coordinate system.

Take the A, C-type of the rotary table as an example, the feature coordinate system set by G68.2 is shown as follows (only setting, no motion occurs):

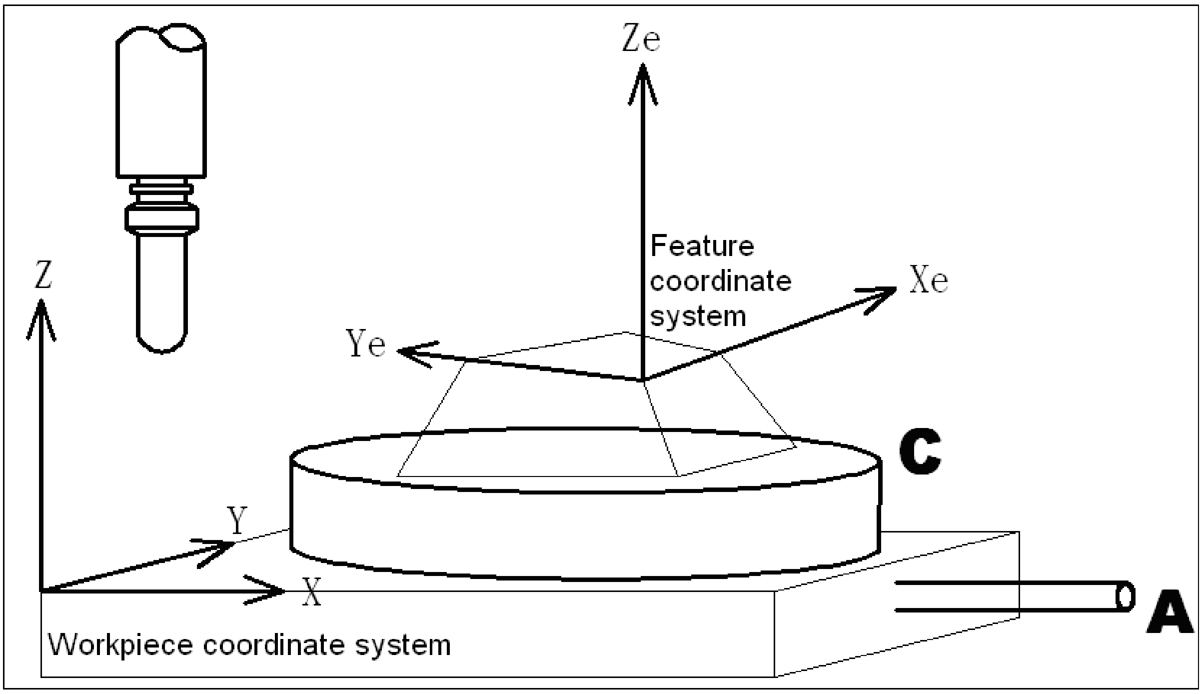


Fig. 3-9-17

After the feature coordinate system (called the first feature coordinate system) is set by command G68.2, when the table rotates by the G53.1 command, the CNC will control the motion of two rotary axis and convert the tool axis direction to the +Z direction of "feature coordinate system". The feature coordinate system that has rotated is called the second feature coordinate system. Once G53.1 is specified, the subsequent machining commands are assumed to be specified in the second feature coordinate system. Shown as follow:

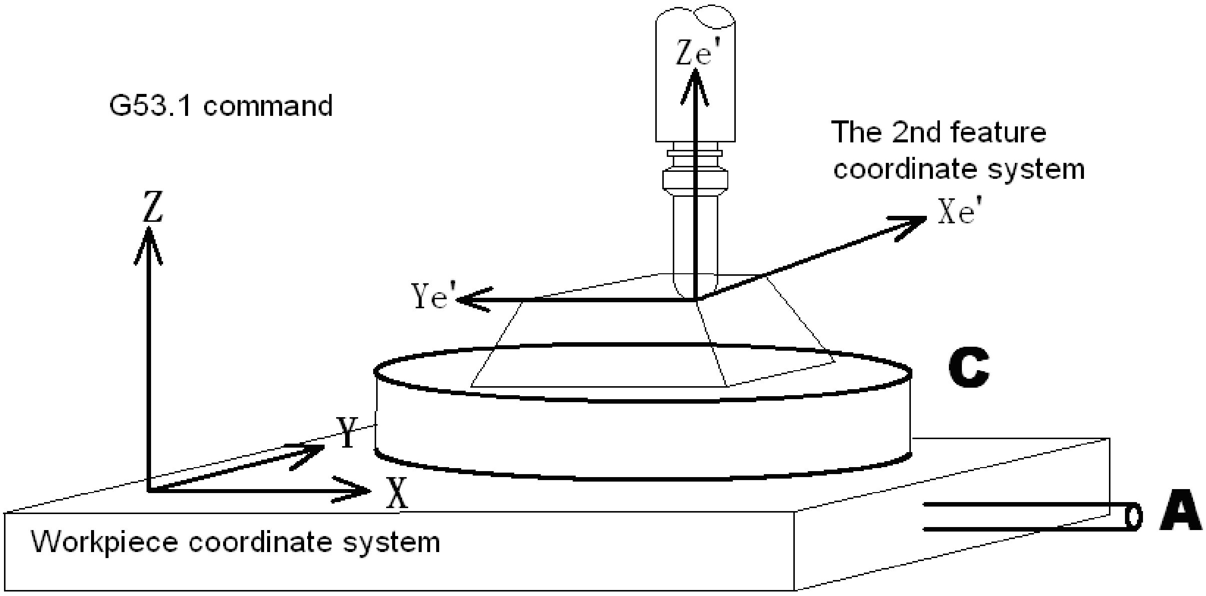


Fig. 3-9-18

Tool rotation type machine

When this function is applied in the tool rotation type machine, the command G68.2 sets the feature coordinate system, G53.1 controls the tool rotary axis in a such a way that the tool axis will be oriented in the +Z direction of feature coordinate system. Tool length compensation can be performed by specifying G43 after the tool rotates, and the control point will be shifted to the tool center point.

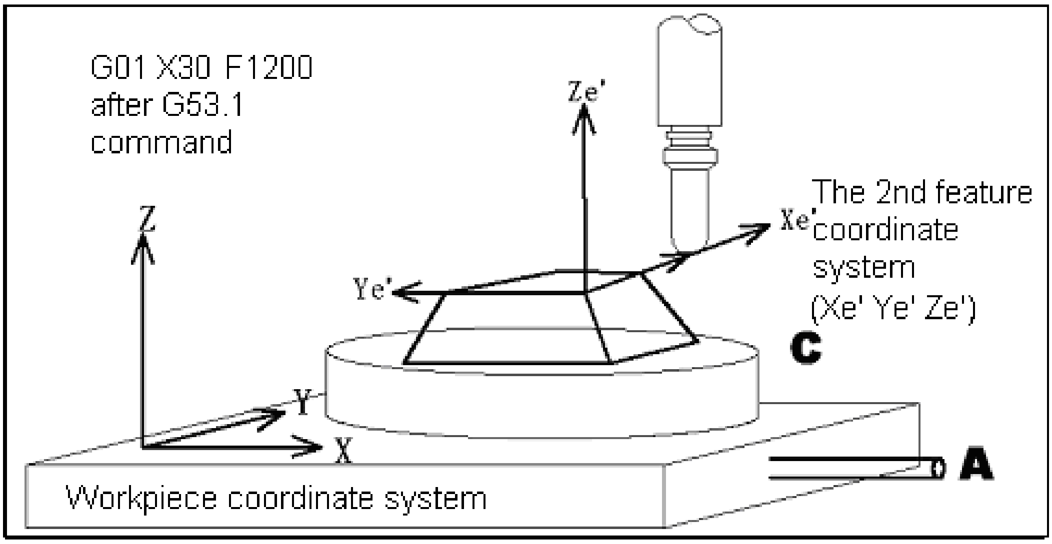


Fig. 3-9-19

Example: (when the tool axis does not cross the rotary axis)

O100
N1 G54 G43 H01;
N2 G90 G01 X0 Y0 Z100.0 F1000;
N3 G68.2 X100.0 Y100.0 Z50.0 I30J15.0 K20.0;
N4 G01 X0 Y0 Z30.0 F1000;

Part Ⅰ Programming

N5 G53.1;
N6 G01 X0 Y0 Z0;

1. Set the feature coordinate system by G68.2. No machine motion occurs.

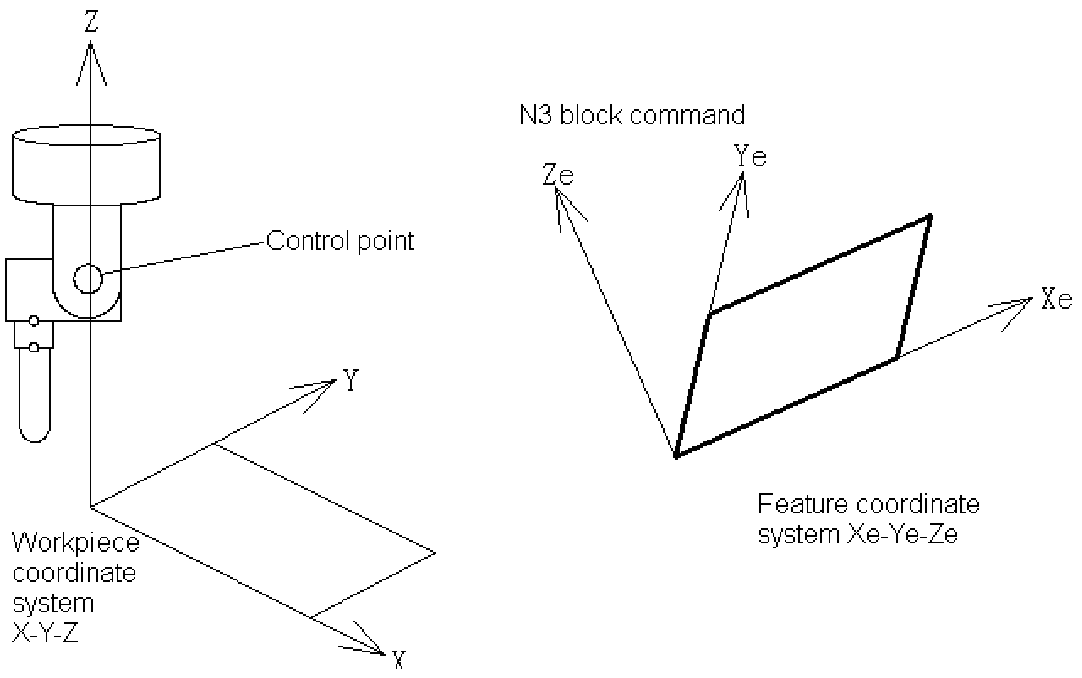


Fig. 3-9-20

2. The commands after G68.2 are specified in the feature coordinate system; therefore, the motions are within the feature coordinate system.

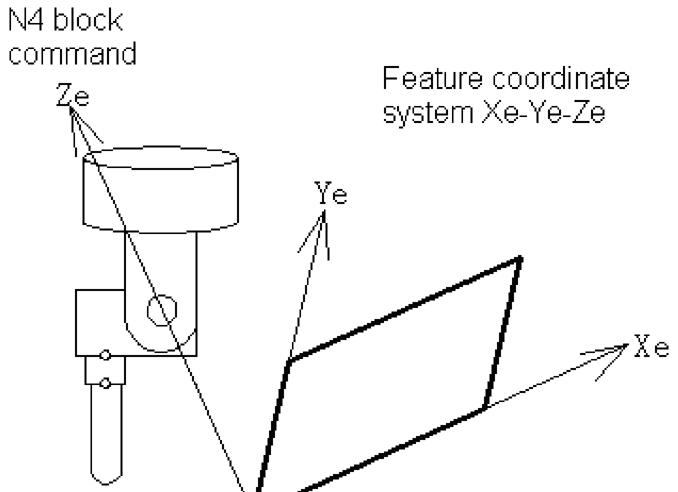


Fig. 3-9-21

3. G53.1 controls the tool rotary axis in a such way that the tool axis will be oriented in the +Z direction of feature coordinate system.

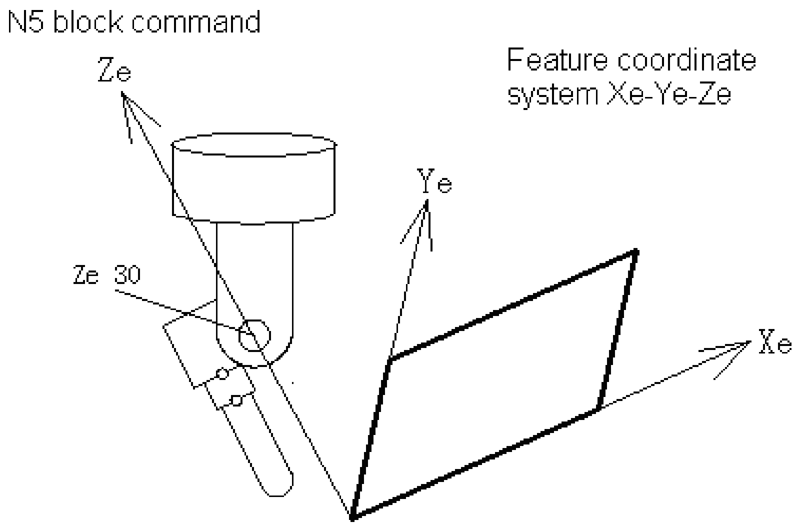


Fig. 3-9-22

4. G43 performs the tool length compensation after considering the tool length and cross offset vector between tool axis and rotary axis. G43 alone does not produce motion. The tool length compensation is valid only when motion commands are specified after G43.

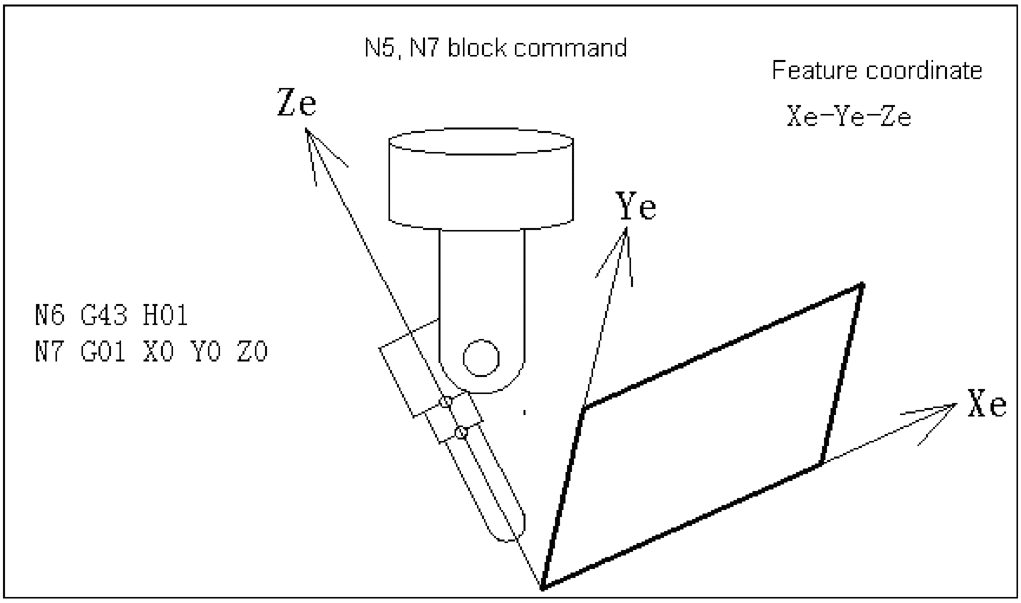


Fig. 3-9-23

(1) The motion when only G53.1 is commanded

When G43 is not commanded, the tool length compensation is not performed. The coordinates in the program are the actual coordinates of the control point.

Example:

O200;

N1 G54;

N2 G90 G01 X0 Y0 Z30.0 F1000;

N3 G68.2 X100.0 Y100.0 Z50.0 I30.0 J15.0 K20.0;

N4 G01 X0 Y0 Z0 F1000;
N5 G53.1;

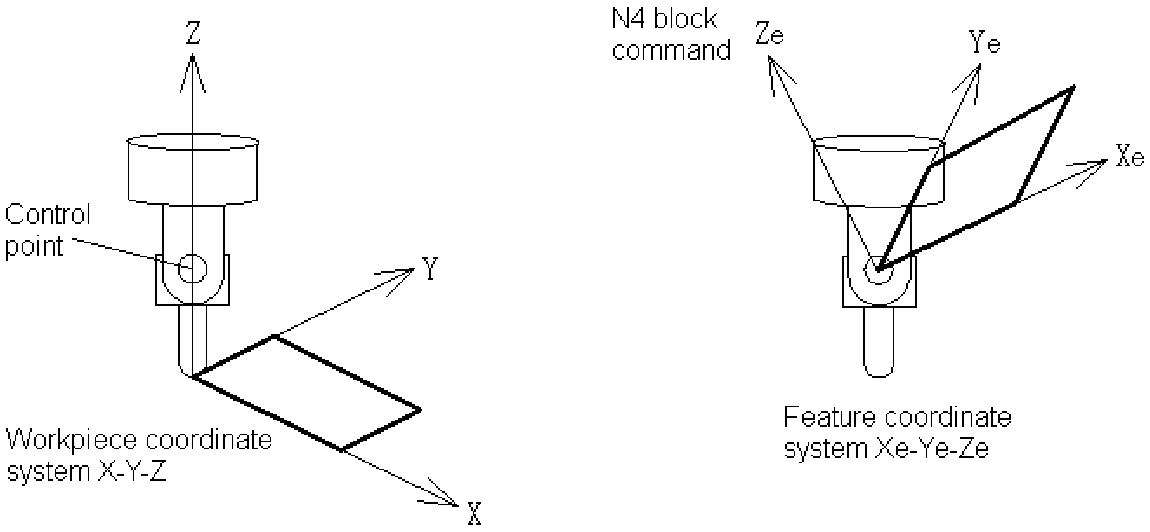


Fig. 3-9-24

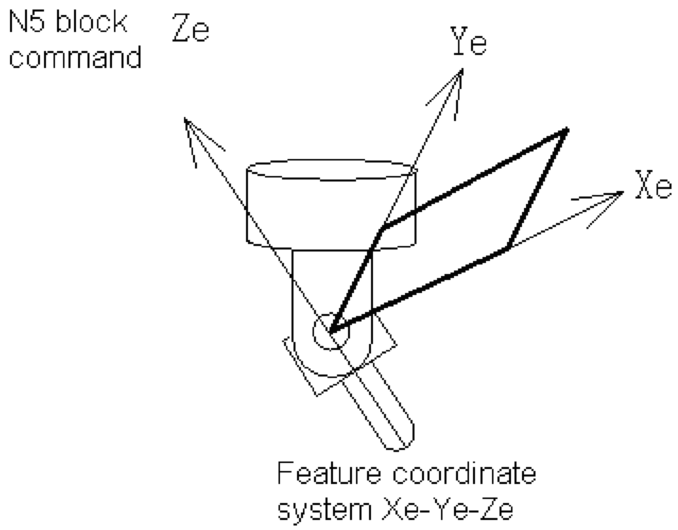


Fig. 3-9-25

(2) The motion when only G43 is commanded

When only G43 is commanded, the tool axis does not rotate. The tool length compensation is performed in the feature coordinate axis after considering the tool length and the cross offset vector between tool and rotary axis. The G43 does not produce motion. The compensation is performed in the motion command after G43.

Example:

N1 G54 G43 H01；
N2 G90 G01 X0 Y0 Z30.0 F1000；
N3 G68.2 X100.0 Y100.0 Z50.0 I30.0 J15.0 K20.0；
N4 G01 X0 Y0 Z0 F1000；
N5 G01 X0 Y0 Z0 F1000；

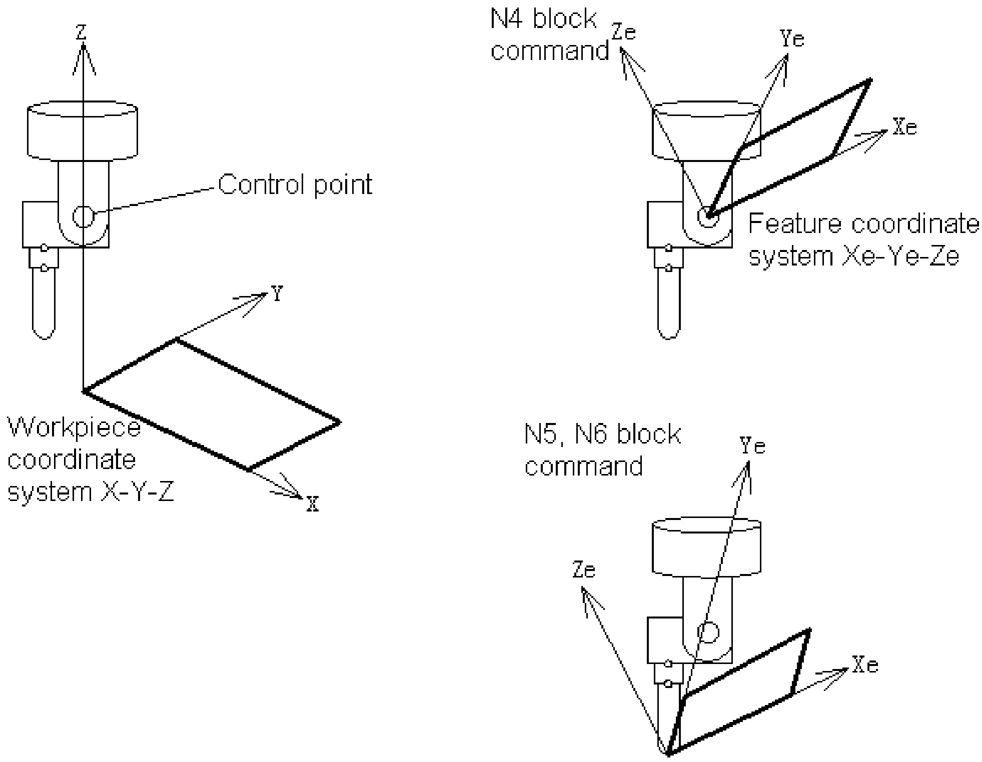


Fig. 3-9-26

(3) Mixed-type machine

This function is also available for a mixed-type machine in which the tool head rotates on the tool rotary axis and the table rotates on the table rotary axis. The feature coordinate system Xe-Ye-Ze is set in the workpiece coordinate system based on the coordinate system origin shift (X0, Y0, Z0) and the Euler's angle. G53.1 controls the tool rotary axis in such a way that tool axis will be oriented in the +Z direction of the feature coordinate system. The worktable rotation will convert the feature coordinate system (called the first feature coordinate system) to a new feature coordinate system (called the second feature coordinate system). The tool axis direction is actually the +Z direction of the "second feature coordinate system). By using G43 after G68.2, the tool length compensation is performed in the feature coordinate axis after considering the tool length and the cross offset vector between tool and rotary axis.

CHAPTER Ⅳ MISCELLANEOUS FUNCTION M CODE

M command consists of an M command address and its following digits, which is a non-axis movement command in machining, used for controlling the program flow or auxiliary functions output to PLC, such as spindle CW reversion/CCW reversion, cooling on/off, tool exchange, table exchange and so on.

Command format:

M □□□□

→ Command value (leading zero can be omitted)

→ Command address

Fig. 4-1-1

M command is used for calling macro program (parameter No.:**6071~6089 specified**) and subprogram (M98, M99). M command of macro program, M98 and M99 is called without execution of PLC.

Correspondence between M command value and function are defined by PLC program which is edited by tool machine builder. Please refer to user manual provided by tool machine builder.

Some closing M codes are described in detail in appendix table 1.8.

4.1 M command for Program Flow Controlling

4.1.1 M00 (Program Stop)

Automatic operation is stopped after M00 is executed. When the program is stopped, all existing modal information remains unchanged. The automatic operation can be restarted by pressing the cycle starting button.

4.1.2 M01 (Optional Stop)

After M01 is executed and optional stop switch on machine operator's panel is pressed, automatic operation is stopped. It realizes the same functions as above mentioned M00. No operation is executed when optional stop switch is off.

4.1.3 End of Program（M30,M02）

This indicates the end of the main program. In automatic operation mode, main program ends and automatic operation stops when M30 and M02 are executed. The system is on a status of reset, The program returns to the beginning by executing M30, while it doesn't return to the beginning when M02 is executed. Note: In this system, the program returns to the beginning when M02 is specified.

4.1.4 Subprogram Call（M98）

This code is used for calling a subprogram. Its format is M98 Pnnnnoooo (nnn is the called times of the program, oooo is the program name). M98 is internal process of NC. M code and strobe signals are not set to PLC.

4.1.5 End of Subprogram or Cycle（M99）

It is used for called subprogram or macro program return controlling to main program or cycle execution program. M98 is internal process of NC. M code and strobe signals are not set to PLC.

Special usage of M99

（1） Specifying the sequence number in main program as returning target

When subprogram is finished, if a sequence number is specified by P, it does not return to the block behind the called one, but it returns to the block which sequence number is specified by P. However, if the main program executes in a mode other than memory mode, P will be omitted.

```
Main program                      Subprogram
O1000                             O1010
N0020 ... ;                       N2020 ... ;
N0030 M98 P1010 ;                 N2030 ... ;
N0040 ... ;                       N2040 ... ;
N0050 ... ;                       N2050 ... ;
N0060 ... ;                       N2060 M99 P0060 ;
```

Fig. 4-1-2

（2） Performing M99 in main program

If M99 is performed in the main program, control returns to the beginning of the main program. For example, put M99 at a proper position of the main program, and the switch of skip any chosen block is set to OFF, M99 will be executed. When executing M99, it returns to the beginning of the main program. Then, blocks are performed repeatedly from the beginning of the main program. When the switch of skip any chosen block is set to OFF, execution is repeated. While the switch is ON, /M99 block is skipped and the next block is executed. If /M99Pn is specified, it does not return to the beginning of the main program but to the block of number n.

```
                   N0100 ... ;
                   N0110 ... ;
                   N0120 ... ;
Program skip       N0130 ... ;
OFF                N0140 ... ;
                   N0150 ... ;
                 / N0160 M99 P0120 ;      Program skip ON
                   N0170 ... ;
                   N0180 ... ;
```

Fig. 4-1-3

（3） Only the subprogram is used

The method of searching the beginning of the subprogram and performing the subprogram are the same as the main program. At this time, if a block with M99 is executed, it returns to the beginning of the subprogram and blocks are executed repeatedly. If the block with M99 Pn is executed, it returns to the block which sequence number is n in the subprogram. If this program needs to be finished, block with /M02 or /M03 should be put at proper position, meanwhile, the switch of any chosen block should be set to OFF. The initial setting of the switch is ON.

```
    N2010 ... ;
    N2020 ... ;
    N2030 ... ;
  / N2040 M30 ;
    N2050 M99 P2020 ;
```

Program skip ON

Fig. 4-1-4

4.2 M Commands Defined by Standard PLC

When machine tool manufacturer uses GSK standard PLC program, meanings of M commands are as follows:

4.2.1 Spindle CW/CCW Rotation and Stop Commands (M03, M04, and M05)

M03: Spindle CW rotation (positive rotation)

M04: Spindle CCW rotation (negative rotation)

M05: Spindle stop

4.2.2 Cooling on/off Commands （M08,M09）

M08: cooling on

M09: cooling off

4.2.3 The 4th Axis Clamp/Release Command (M10, M11)

The 4th axis clamp

The 4th axis release

4.2.4 Spindle Directional Command (M19)

M19: Spindle orientation is to stop the spindle at a specified angle position.

4.2.5 The 5th Axis Clamp/Release Commands (M20, M21)

The 5th axis clamp

The 5th axis release

4.2.6 Rigid Tapping Commands (M29)

M29: The system and the spindle servo are changed into rigid tapping state by this command.

CHAPTER Ⅴ FEED FUNCTION

Feed function is to control the traverse speed of the tool, which includes rapid traverse and cutting feed.

5.1 Rapid Feed (Rapid Traverse)

Rapid traverse is defined by G00, G27, G28, G29, G30 and G60, which is used for tool rapid poisoning.

Rapid traverse speed is not defined in programming and is set by N1226, and each separately sets the rapid traverse speed.

Allowable feedrate range is depended on the specification of machine, which maximum value is limited appropriately.

Range adjustment keys on panel can be operated as follows:
F0,F25％,F50％,F100%

F0: Set by parameter **N1231**

Note: In G00 block, it is valid though feedrate F is defined, G0 is used to position.

5.2 Cutting Feed

In linear interpolation (G01) and circular interpolation (G02, G03), the numbers following F code are used to commend the feedrate of tool. The tool moves at the cutting feedrate complied in programming. Cutting feedrate can be adjusted (range: 0％～200%) by override switch on machine operation panel.

1. Feed per minute (G94): Tool feed per minute is specified by setting a number after F.
2. Feed per revolution (G95): Tool feed per revolution is specified by setting a number after F.

5.2.1 Feed per Minute（G94）

Format: G94 F_
Function: Tool feed per minute. Unit: mm/min or inch/min
Explanations:

1. When G94 (mode of feed per minute) is specified, tool feed per minute is specified by setting a number after F
2. G94 is a modal command, and it is valid until G95 is specified. Feed per minute is default of the system starting.
3. An override from 0% to 200% can be applied with override switch on panel.

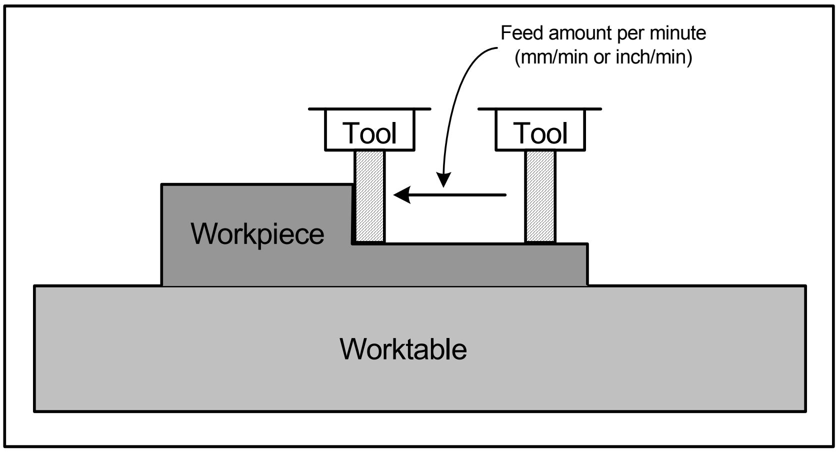


Fig.5-2-1 Feed per minute

Restrictions: The feedrate value F should be specified again when G95/G94 mode switching is performing, otherwise, alarm occurs.

5.2.2 Feed per Revolution（G95）

Format: G95 F_

Function: Tool feed per minute. Unit: mm/min or inch/min

Explanations:

1. When G95 (mode of feed per revolution) is specified, tool feed per minute is specified by setting a number after F.
2. G94 is a modal command, and it is valid until G95 is specified.
3. An override from 0% to 200% can be applied with override switch on panel.
4. Upper limit of feedrate is set by per minute, and the feed per rev is also limited by upper limit of federate. Feed per rev and feed per minute are transferred as follows:

   Fm=Fr×N

   Fm: feed per minute

   Fr: feed per revolution

   N: Spindle speed

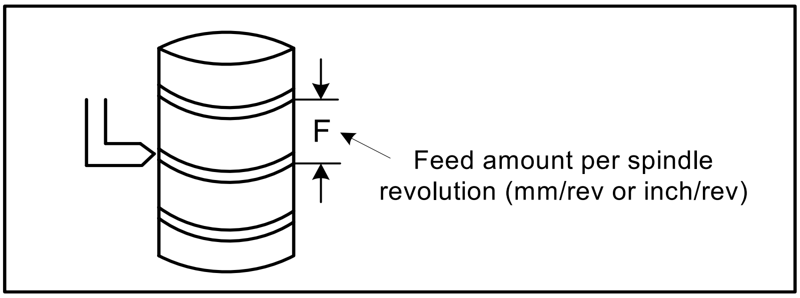


Fig.5-2-2 Feed per revolution

Note: When the speed of spindle is low, feedrate fluctuation may occur. The slower the spindle rotates, the more frequently feedrate fluctuation occurs.

5.3 Tangential Speed Control

Cutting feed controls tangential speed of contour path to reach the feedrate specified by command.

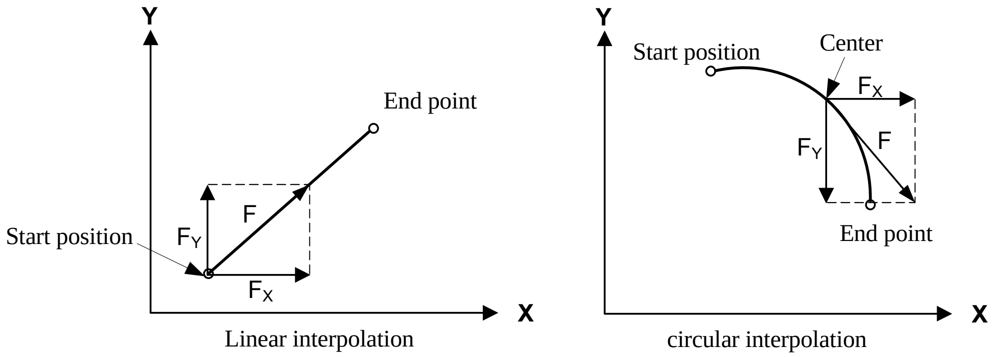


Fig. 5-3-1

F: feedrate on tangent direction $F = \sqrt{F_x^{\ 2} + F_y^{\ 2} + F_z^{\ 2}}$

F_x: feedrate on X axis direction F_y: feedrate on Y axis direction F_z: feedrate on Z axis direction.

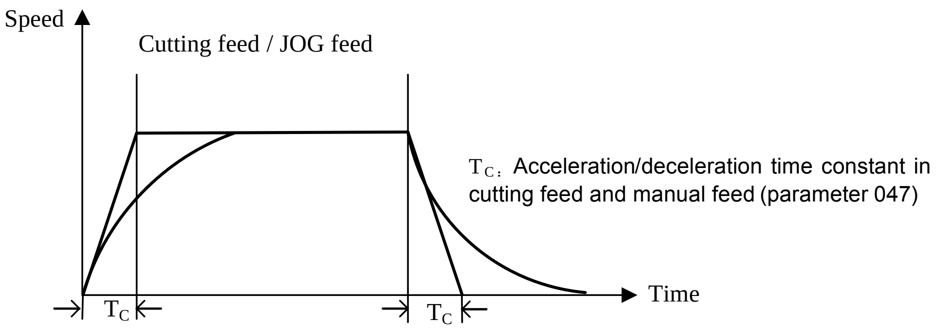


Fig. 5-3-2

5.4 Acceleration/Deceleration Process on the Corner of Program

Example: Only Y moves in the last block, X moves in the next block, X accelerates when Y decelerates, then, the tool path is as follows:

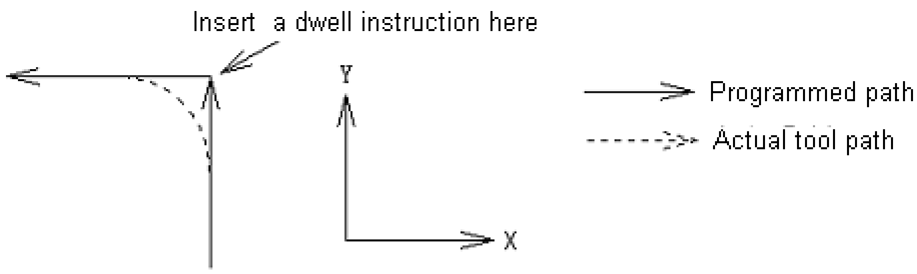


Fig. 5-4-1

If exact stop command is inserted, tool will be move according to full line on above figure. Otherwise, the bigger the cutting federate is, or the longer the acceleration/ deceleration time constant is, the bigger the arc at the corner is. In arc command, arc radius of actual tool path is smaller than the one specified by the program. On allowable range of mechanical system, reduce acceleration/deceleration time constant should be reduced at the corner.

CHAPTER Ⅵ SPINDLE FUNCTION

6.1 Spindle Control

Speed is specified by S code and the number following. After code signal is transferred into analog signal, which is sent to machine used for spindle control.

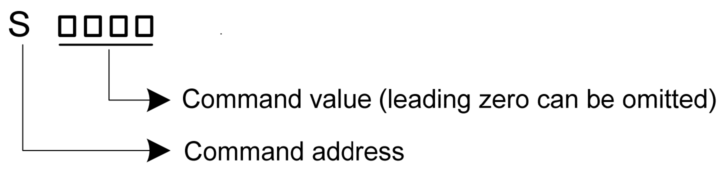


Fig. 6-1-1

Explanations:

1. The spindle speed can be specified directly by address S followed by a max. five-digit value. Its unit is r/min. For example: M3 S300 indicates that spindle revolution speed is 300 rotations per minute.

2. S is an analog value, which is not cleared in reset, but cleared when power off.

3. Necessary requirements of analog spindle rotation are specified rotation directions of M3 and M4 besides speed specified by S code.

4. When movement command and S command are in the same block, both of them are executed at the same time.

CHAPTE Ⅶ TOOL FUNCTION (T FUNCTION)

7.1 Tool Selection Function

Tool selection function consists of address T and the following number, which is used for selecting tools on the machine.

Command format:

T □□□□

→ Command value (leading zero can be omitted)

→ Command address

Fig. 7-1-1

Only one T code can be commanded in a block. For number digits specified the address T and the machine operations corresponding to T, please refer to machine tool builder’s manual.

When a movement command and a T code are specified in the same block, the commands are executed in one of the following two ways:

1. Execute the movement command and T code simultaneously.

2. Execute T code after completion of movement command execution.

The selection of either 1 or 2 refers to the machine tool builder’s operation manual.

PART Ⅱ OPERATION

CHAPTE Ⅰ OPERATION PANEL

1.1 Panel Division

GSK25i is employed an aluminum alloy solid operator panel, which includes LCD (liquid crystal display), edit keypad, menu display and machine operation panel etc.

1.2 Panel Functions

1.2.1 LCD (Liquid Crystal Display)

The system employed color 10.4 inch liquid crystal display, with resolution ratio 800×600.

1.2.2 Edit Keypad

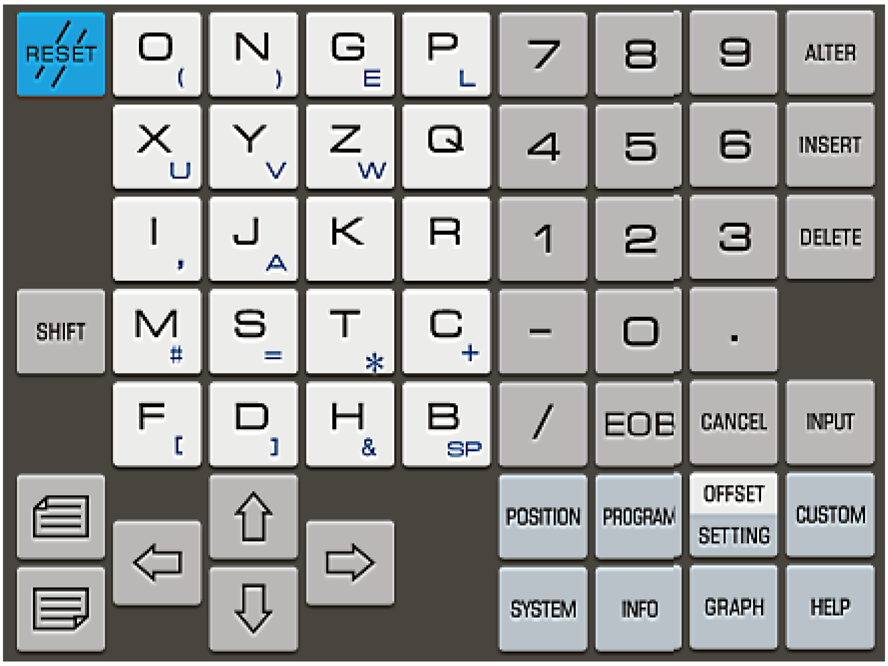


Fig. 1-2-1

Edit keypad is divided into 10 small areas; the function of each area is as follows:

No.	Key name	Function Exlanation
1	Reset key	System reset, feed, output stop
2	Address key	Address input
3	Numerical key	Digt input
4	Cancel key	Delete input characters（unsaved in buffer）
5	Input key	Press this key after number key or address key is pressed, and data is saved in buffer
6	Help	Enter help manual and PMC ladder diagram information
7	Screen operation key	Press any key to enter corresponding interface (details are shown as follows)
8	Page key	For page turning in an interface
9	Cursor move key	Cursor moving up, down, left or right
10	Edit key	For insertion, alteration, deletion of program and word in editing

1.2.3 Introduction of Screen Operation Keys

There are 7 page manual display keys on operation panel, see following figure:

Key Name	Function Exlanation	Remark
Position	To enter position interface	Display current relative pos., absolute pos., integrated pos., monitor display pages through softkey switching.
program	To enter program interface	Display program, MDI, detection, data, file list display pages. Program list is switched by page turning keys.
system	To enter system interface	Switch parameters, diagnosis, PLC through softkey. Check or alter parameters, edit PLC etc.
alarm	To enter alarm interface	Check each alarm information pages through softkey.
graph	To enter graph interface	Display reference graph, graph pages through softkey. Graph center, size, proportion and display interface are set here.
help	To enter help interface	Check corresponding information of the system through softkey.
Offset/ Page set	To enter Offset/ Page set interface	Set tool length compensation, radius compensation and screw-pitch error compensation of each feed axis through softkey switching display. Set coordinate system of work part, macro variables and log-in etc.

Note: By pressing corresponding function keys continuously, above softkey interfaces can be viewed. Please refer to chapter three of this manual for detailed explanation of each page.

1.2.4 Machine Control Panel

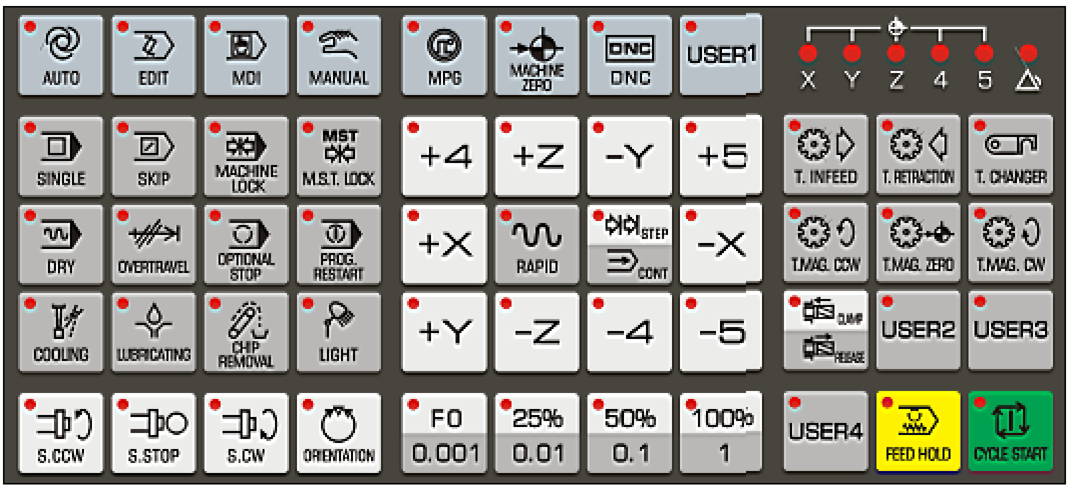


Fig. 1-2-2

Part Ⅱ Operation

Key name	Selection mode	Function explanation	Operation and remarks
Edit	edit mode key	To enter edit mode	Switch to edit mode in automatic operation. The system stops after present block is performed.
Auto mode	Auto mode key	To enter auto mode	The system selects internal memory program in auto mode
MDI	MDI mode key	To enter MDI mode	The switch from auto mode into MID mode is completed after the system has run the current to stop
Machine zero mode key	Machine zero mode	To enter machine zero mode	Being switched to machine zero in auto mode, the system decelerates and stops immediately
Step/continous	Manual step mode key	To enter step mode	This mode is valid only in manual mode
Manual	Manual mode key	To enter manual mode	After being switched from manual mode into auto operation, the system immediately decelerates to stop
MPG	MPG mode	To enter MPG mode	Being switched to MPG mode from auto operation, the system immediately decelerates to stop
DNC	DNC mode	To enter DNC mode	Being switched to DNC mode in auto operation, the system immediately decelerates to stop
Spindle CCW Spindle stop Spindle CW]Spindle exact stop	Spindle control key	Spindle CCW Spindle stop Spindle CW Spindle exact stop	MPG, step, manual
Spindle override	Spindle override key	For spindle speed adjustment (spindle analog control)	Any mode
Tool magazine zero Tool magazine CCW Tool magazine CW Tool magazine	Tool magzine key	Tool magzine on/off	Manual mode

forward Tool magazine backward			
Clamp/release tool	Key of clamp/ release tool	clamp/release tool manually	Manual mode
Manual tool change	Manual tool change	Manual tool change	Auto mode
Block Skip key	Block Skip key	For skipping of block headed with“/”sign, if its switch is set for ON, the indicator lights up	Auto mode, MDI, DNC
Single block	Single block switch	Single block/ continues execution switching. if its switch is set for ON, the indicator lights up	Auto mode, MDI, DNC
Dry run	Dry run switch	If dry run is valid, the Block Skip indicator lights up	Auto mode, MDI, DNC
M.S.T. lock key	M.S.T. lock key	If its M.S.T. lock is set for ON, the indicator lights up. M,S,T function is invalid	Auto mode, MDI, DNC
Machione lock key	Machione lock key	If its machine lock is set for ON, the indicator lights up. Axis operation output is invalid	Auto mode, MDI, machine zero, MPG, step, manual mode, DNC
Work light	Work light switch	Work light switch on/off	Any mode
Lubricating	Lubricating switch	Lubricating switch on/off	Any mode
Cooling switch	Cooling switch	Cooling switch on/off	Any mode
Chip removal	Chip removal key	Chip removal key on/off	Any mode
Feedrate override key	Feedrate override key	Feedrate override adjustment	Auto mode, MDI, Edit mode, machine zero, MPG, step, manual mode, DNC
Rapid traverse	Rapid traverse key	Rapid traverse on/off	Any mode
F0（0．001,0．01，0．1,1）	Selection of rapid override, manual step, MPG override.	Rapid override, manual, step, MPG override selection keys.	Auto mode, MDI, machine zero, MPG, step, manual mode, DNC
+X/-X/+Y/-Y/+Z/-Z/+4/-4/+5/-5	Manual feed key	For positive/negative moving of X, Y, Z, 4, 5 in manual/step operation mode. The positive is the MPG selection axes	Machine zero, step, manual mode, MPG
Overtravel	Key of overtravel	After the machine	Manual mode

release	release	moves to press down the hard limit, it alarms, the overtravel release key is pressed, its indicator lights, and the machine reversely moves till the indicator is OFF.	
Optional stop	Optional stop ON/OFF	Whether the program with M01 is stopped	Auto mode, MDI, DNC
Feed hold	Feed hold key	Auto operation of the system is stopped by pressing this key.	Auto mode, MDI, DNC
Cycle start	Cycle start key	The system is performed automatically by pressing this key	Auto mode, MDI, DNC

CHAPTE Ⅱ SYSTEM POWER ON/OFF and PROTECTION

2.1 System Power on

Before this GSK25i is powered on, confirm that:

1. The machine is in a normal state.
2. The power voltage conforms to the requirement of the machine.
3. The connection is correct and secure.

The liquid crystal display is shown as follows after power on:

Fig. 2-1-1

The current position (Absolute POS) is displayed after the self-check and initiation are finished.

2.2 Power off

Before power is off, confirm that:

1. X,Y Z, 4th and 5th axes of the CNC are stopped;
2. Miscellaneous functions (spindle, pump etc.) are off;
3. The CNC power is cut off prior to machine power is cut off.

Before power is off, check that:

1. LED indicating cycle start of the panel is in a halted state;
2. All movable parts on CNC machine are in a halted state;
3. CNC power is cut off by pressing POWER OFF button.

Emergency Power Off

Under the emergency situations during the machine operation, the machine power should be cut off immediately to avoid the incidents. But it should be noted that there may be an error between the system displayed coordinate and the actual position. So the machine zero and tool setting operation should be performed again.

Note: Please see the machine builder’s manual for the machine power cut-off operation.

2.3 Safety Operation

2.3.1 Reset

Press  key and the system is in the status of reset:

1. All axes movement stop.
2. M, S, T function is invalid.
3. Processing mode is set by altering parameter ＃1801 after reset. 0: System reset; 1: System clear.
4. Set tool length offset reset by altering parameter ＃2601. 0: Not clear; 1: Clear.

Note: Programming can not be performed in Reset mode.

2.3.2 Emergency Stop

Fig. 2-3-1

Under the emergency situations, all axes movement of machine (spindle rotation, cooling) are stopped immediately by pressing emergency stop button. The button is holding on stop position.

Button release modes are different for their different machine tool builders, usually are released by pressing down to turn CW.

Note 1: Cut off machine power by pressing this button.

Note 2: Control unit is on a status of reset.

Note 3: Troubles are removed prior to button is released.

Note 4: After the button is released, return to reference point with manual operation or G28 command.

Note 5: Under the emergency situations, edit operation can not be performed in the system.

General emergency stop signal is normally-closed contact signal. When the contact is disconnected, the system enters emergency stop state, making the machine stop. Circuit connection of emergency stop signal is as follows:

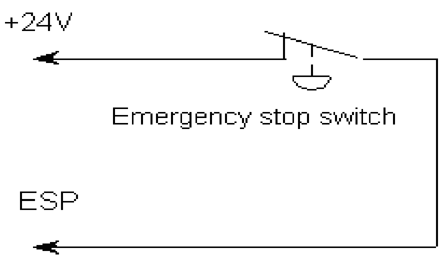


Fig. 2-3-2

2.3.3 Feed Hold

key（or button）can be pressed to make the running pause when the machine is running. Feed hold indicator lights up simultaneously. It calls for special notice that the running pauses after current command is finished when rigid tapping, cycle command or single block command is executing.

2.4 Cycle Start and Feed Hold

Start and feed hold keys on panel are used for program start and pause operation in auto, MDI and DNC mode.

2.5 Overtravel Protection

Overtravel protection measures should be taken to prevent machine damage due to the overtravel of X, Y and Z axes.

2.5.1 Hardware Overtravel Protection

The stroke limit switches are fixed at the positive and negative maximum stroke position of X, Y and Z axes respectively, If the overtravel occurs, running axis decelerates to eventually stop when it contact with limit switch, and the emergency alarm is issued.

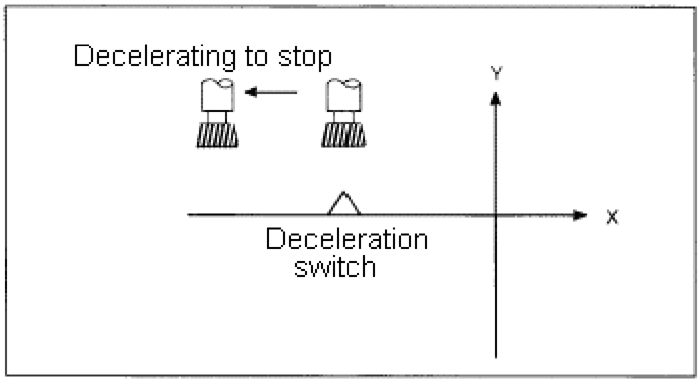


Fig. 2-5-1

Detailed Explanations:

(1) Overtravel in auto mode

In auto mode, all existed axes operation decelerate to stop eventually when tool moves along one axis and contacts with limit switch. Overtravel alarm displays simultaneously. Program stops on current block that overtravel occurs.

(2) Overtravel in manual mode

In manual mode, once any axis of machine contacts with limit switch, all operation of axes decelerate to stop.

(3) Elimination of hardware overtravel

The steps to eliminate overtravel alarm are: In Manual mode, move axes reversely (For example: move out negatively for positive overtravel, vice versa).

After limitation alarm occurs, the axis can not move before eliminating overtravel alarm even though the limitation switch returns to non-operating state.

Because dealing methods are different for different manufacturer, please refer to manual provided by manufacturer for detailed overtravel eliminating method.

2.5.2 Software Overtravel Protection

Software stroke range is set by parameter No: 1080~ No: 1087. Set coordinate value of machine as reference value. If machine position (machine coordinate) exceeds software stroke range, overtravel alarm will occur. When In manual mode, move axis reversely until the axis moves out of the overtravel area and then press reset key to eliminate the alarm.

2.5.3 Stored Stroke Check（G22-G23）

Three areas in which the tool cannot enter can be specified with stored stroke check 1, stored stroke check 2, and stored stroke check 3.

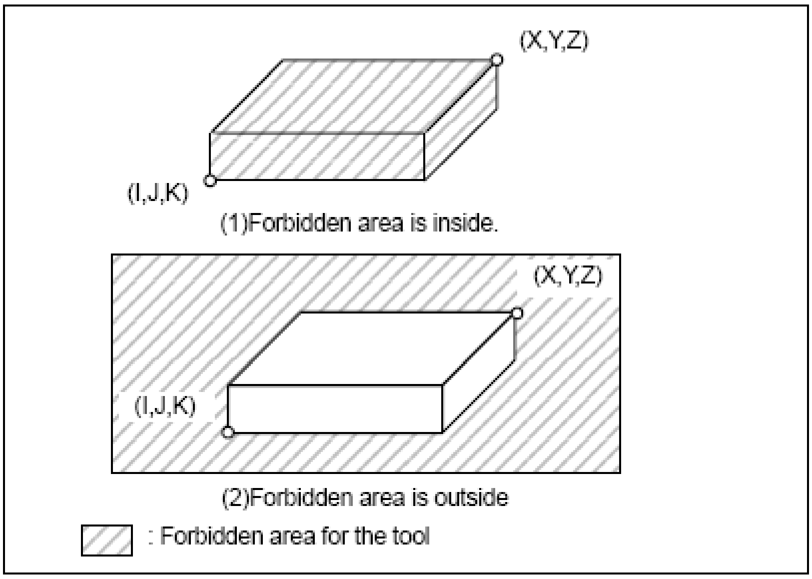


Fig. 2-5-2

Forbidden area that detected by storage stroke check are as follows:

Stroke check 1: outside inhibit

Stroke check 2: inside inhibit and outside inhibit can be switched

Stroke check 3: inside inhibit

When the tool moves into the forbidden area, the tool decelerates and stops. When outside stroke check is performed and the tool enters the forbidden area and an alarm is generated, the tool can be moved in the reverse direction from it came.

Explanations:

1. Stored stroke check 1

Parameters (No. 1080, 1081) set boundary. Outside the area of the set limits is a forbidden area. The machine tool builder usually sets this area as the maximum stroke.

2. Stored stroke check 2

Parameters (No. 1082, 1083) or commands set these boundaries. Inside or outside the area of the limit (G22, G23) can be set as the forbidden area. Parameter OUT (No. 1070#0) selects either inside or outside as the forbidden area.

By setting parameter OT2 (No.1070#2), it is able to set whether perform storage stroke check 2 (0: not perform 1: perform). When the parameter is set to 0 (not perform), command G22 is invalid. When the parameter is set to 1 (perform), G22 is valid.

When a program command is performed, a G22 command forbids the tool to enter the forbidden are, and a G23 command permits the tool to enter the forbidden area. Each of G22 and G23 should be commanded by an independently block.

The commands below are used for creating or changing the forbidden area:

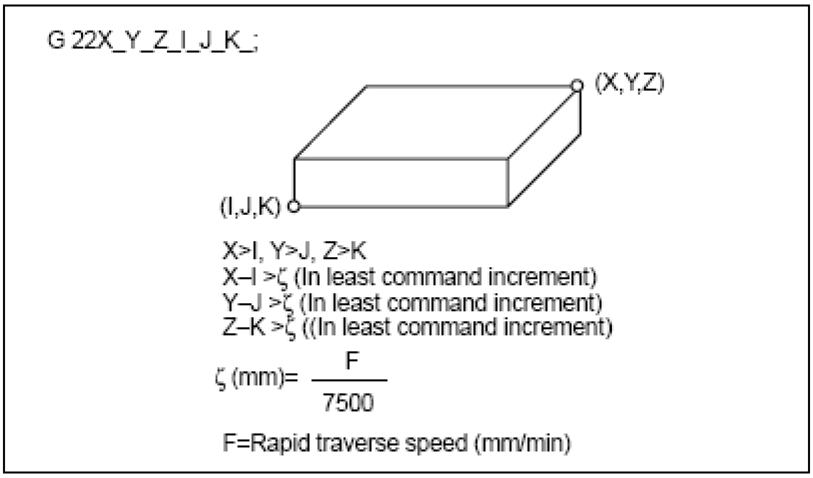


Fig. 2-5-3 Creating or changing the forbidden area by a program

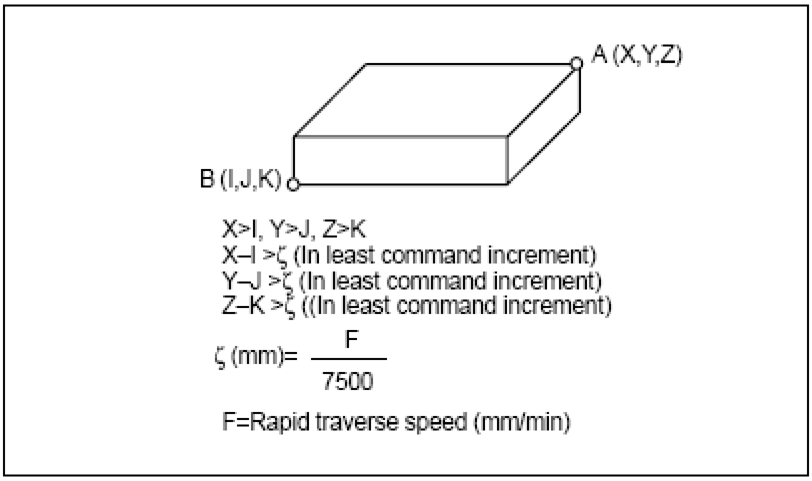


Fig. 2-5-4 Creating or changing the forbidden area by a program

In stored stroke check 2, alarm occurs when I≥X，J>Y，K>Z.

When the forbidden area is set by parameter Nos. 1082, 1083, the data should be specified by the distance from the machine coordinate system in the least command increment. (output increment).

If it is set by a G22 command, specify the data by the distance from the machine coordinate system in the least increment (input increment.) The programmed data are then converted into the numerical values in the least command increment, and the values are set as the parameters.

3. Checkpoint for the forbidden area

Confirm the checking position (the top of the tool or the tool chuck) before programming the forbidden area. If point A (the top of the tool) is checked in the following figure, the distance “a” should be set as the data for the stored stroke limit function. If point B (the tool chuck) is checked, the distance “b” must be set. When checking the tool tip (like point A), and if the tool length varies for each tool, setting the forbidden area for the longest tool requires no re-setting and results in safe operation.

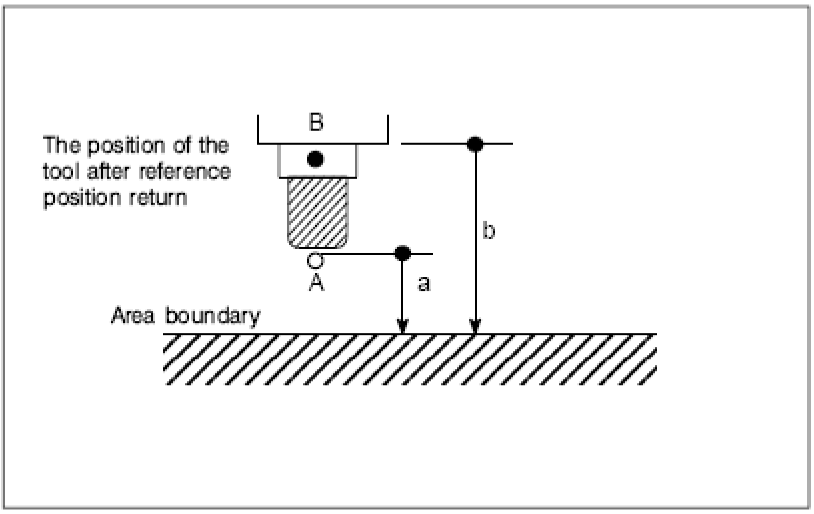


Fig. 2-5-5 Setting the forbidden area

4. Forbidden area overlapping

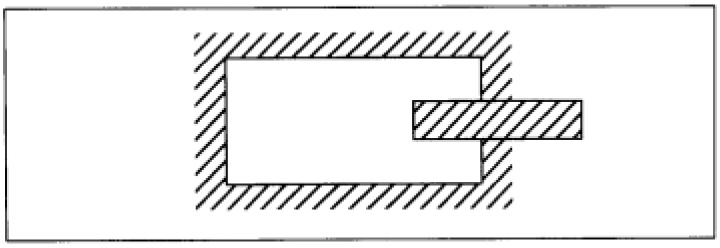


Fig. 2-5-6 Setting the forbidden area overlapping

Unnecessary limits should be set beyond the machine stroke.

5. Overrun amount of stored stroke limit

If the maximum rapid traverse rate if F (mm/min), the maximum overrun amount, L (mm), of the stored stroke limit is obtained from the following expressing:

L (mm) =F/7500

The tool enters the specified inhibited area by up to L (mm).

6. Effective time for a forbidden area

Forbidden area (In G22 mode for stored stroke limit 2) are valid after machine coordinates are established.

When parameter 1070.6=0 before machine coordinate is established, the forbidden area (in G22 mode for stored stroke limit 2) is valid if parameter 1070.6=1.

7. Releasing the alarms

If the enters a forbidden area and an alarm is generated, the tool can be moved only in the backward direction. To cancel the alarm, move the tool backward until it is outside the forbidden area and reset the system. When the alarm is canceled, the tool can be moved both backward and forward. If enters an inside storage stroke forbidden area and an alarm is generated, the tool shall be moved out according to the actual operation conditions, and then reset to eliminate the alarm.

8. Changing from G23 to G22 in forbidden area

In forbidden area, the alarm will generate when G23 is switched to G22.

CHAPTE Ⅲ INTERFACE DISPLAY AND OPERATION

3.1 Position Interface

3.1.1 Five Ways for Interface Display

Press 

to enter Position Interface, there are five modes in this interface such as **【ABS】**, **【REL】**,**【COM】**,**【POSC】**,**【AUTO】**. They can be viewed by corresponding softkey or pressing 

key continuously. Detailed information of each interface is as follows:

1) **Absolute mode:** Press **【ABS】** key to display current tool position on current coordinate system. It is called "absolute coordinate" hereafter (see Fig. 3-1-1)

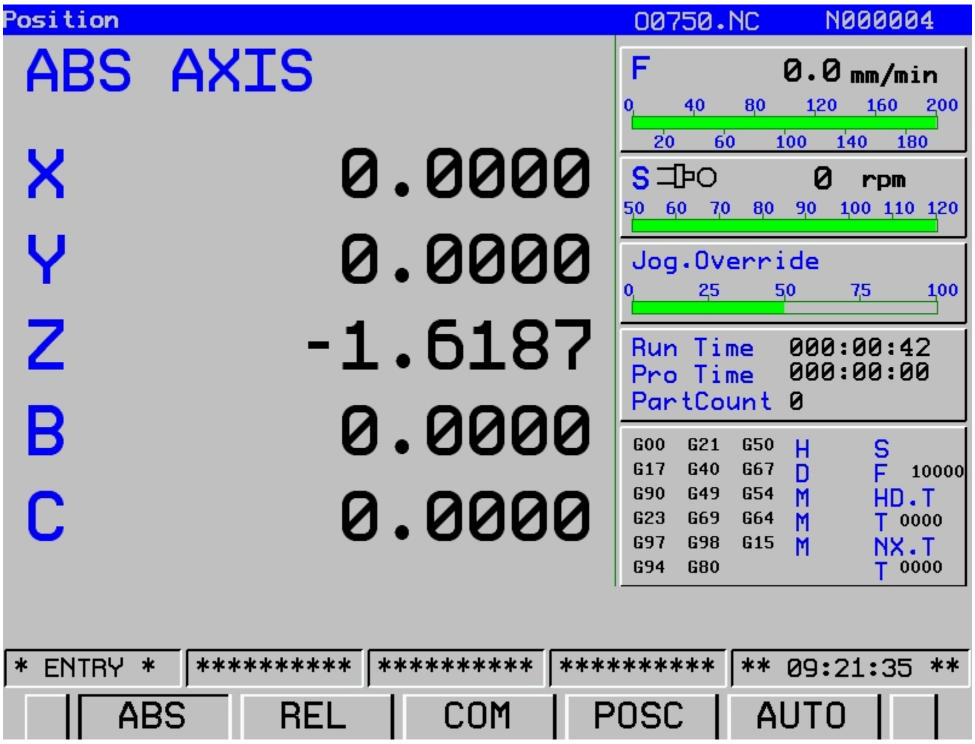


Fig. 3-1-1 Absolute coordinates

In the figure, the values on the left are absolute ones of the coordinate system. The first progress on the right is federate (F), which can be adjusted by feed override button. The progress of the following S and rapid override are also adjusted by override button with different override values.

2) **Relative mode:** relative interface displays current tool position on relative coordinate system. It will be called “Relative coordinate” hereafter. Press softkey **【REL】** to enter sub-interface of relative interface (see Fig. 3-1-2).

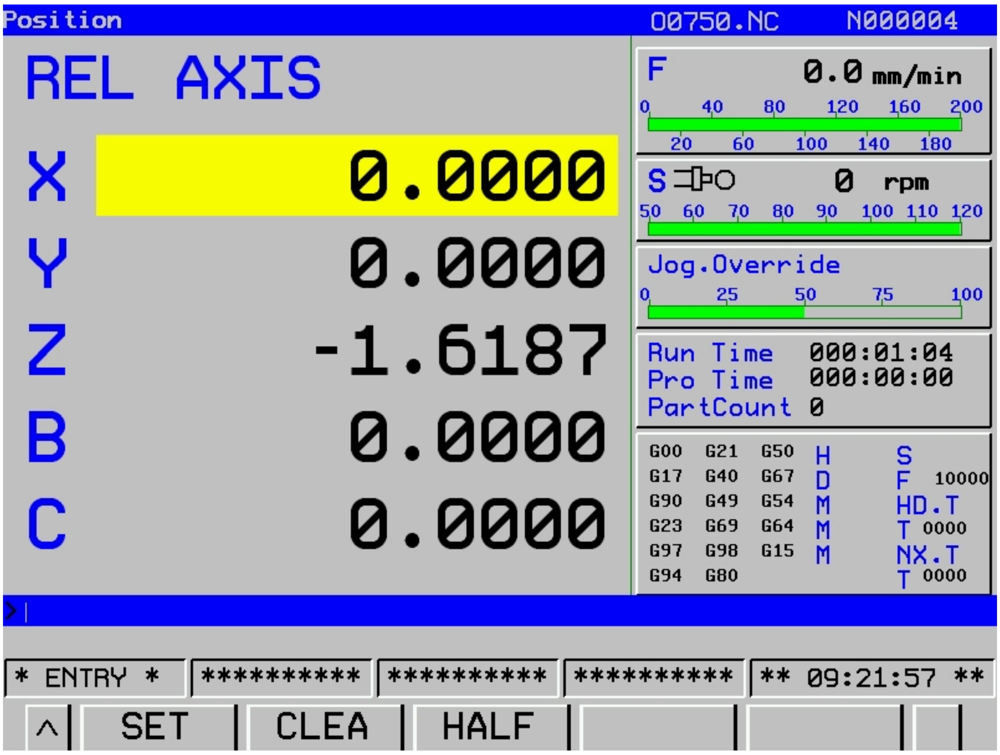


Fig. 3-1-2 Relative coordinates

Preset steps of relative coordinate: chose the axis needs to be altered by up and down direction keys, selected position turns yellow. Input data need to be set to corresponding coordinate by pressing **【SET】** key, and cursor will skip to the next line.

The clearing steps of relative coordinates system: Select the axis by up and down direction keys, then press softkey **【CLEA】** to clear X coordinates.

Half steps of the relative coordinates system: half operations are similar to the above ones.

3) Integrated interface

Press**【COM】**softkey to enter this interface. The following coordinate position values are displayed:

(A) Position on relative coordinate system;

(B) Position on absolute coordinate system;

(C) Position on machine coordinate system;

(D) Range-to-go

There are other information including speed, operation time, parts counting, current mode and so on. Detailed display page is as follows (Fig.3-1-3):

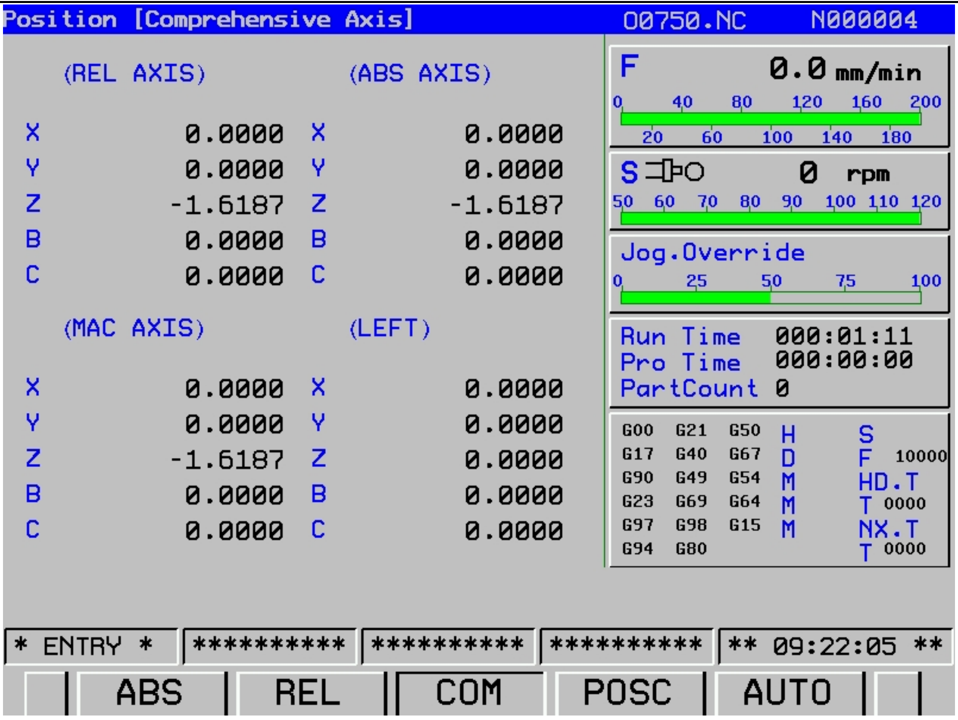


Fig. 3-1-3 Detailed display page

4）Monitoring Mode

Press 【POSC】softkey to enter【POSC】interface（see Fig. 3-1-4）

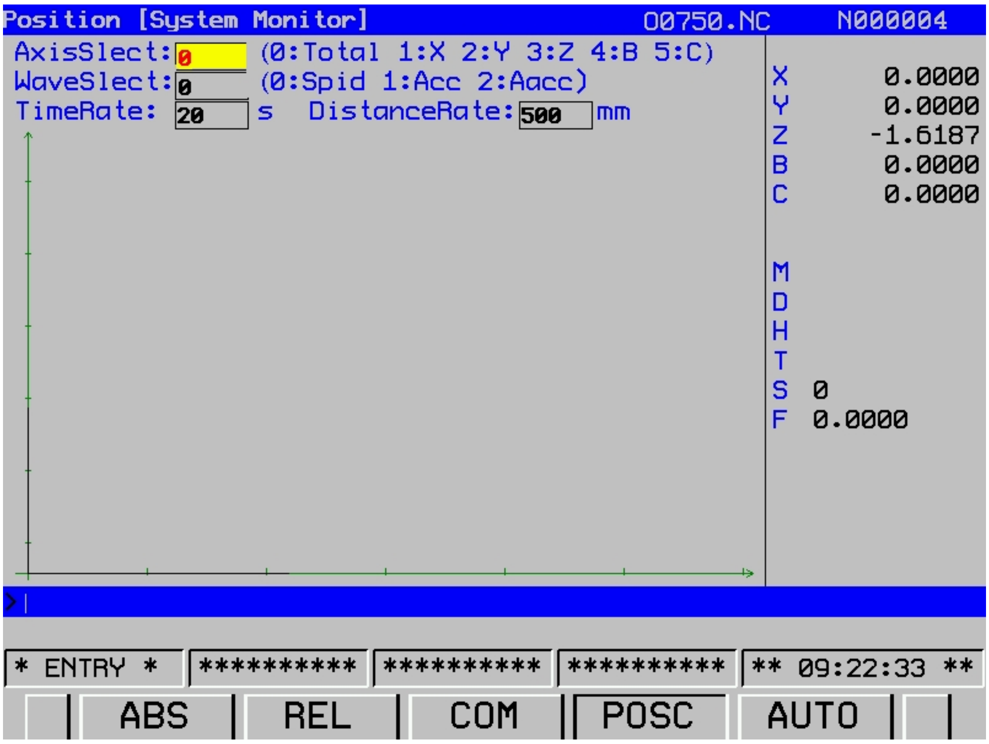


Fig. 3-1-4 Monitoring interface

Each axis perform state can be viewed by altering axis selection parameters, and wave speed, acceleration, jerk are viewed by altering wave shape selection. Wave shape display proportion is changed by altering the proportion of two axes. Among them, cross axis indicates time proportion axis. Each lattice indicates input time block. Vertical axis for distance proportion axis, and each lattice is input distance.

Absolute coordinate value of current operation and some simple modal information are displayed in monitoring interface.

5) Auto-check

Press【**AUTO**】softkey to enter this interface (see Fig. 3-1-5).

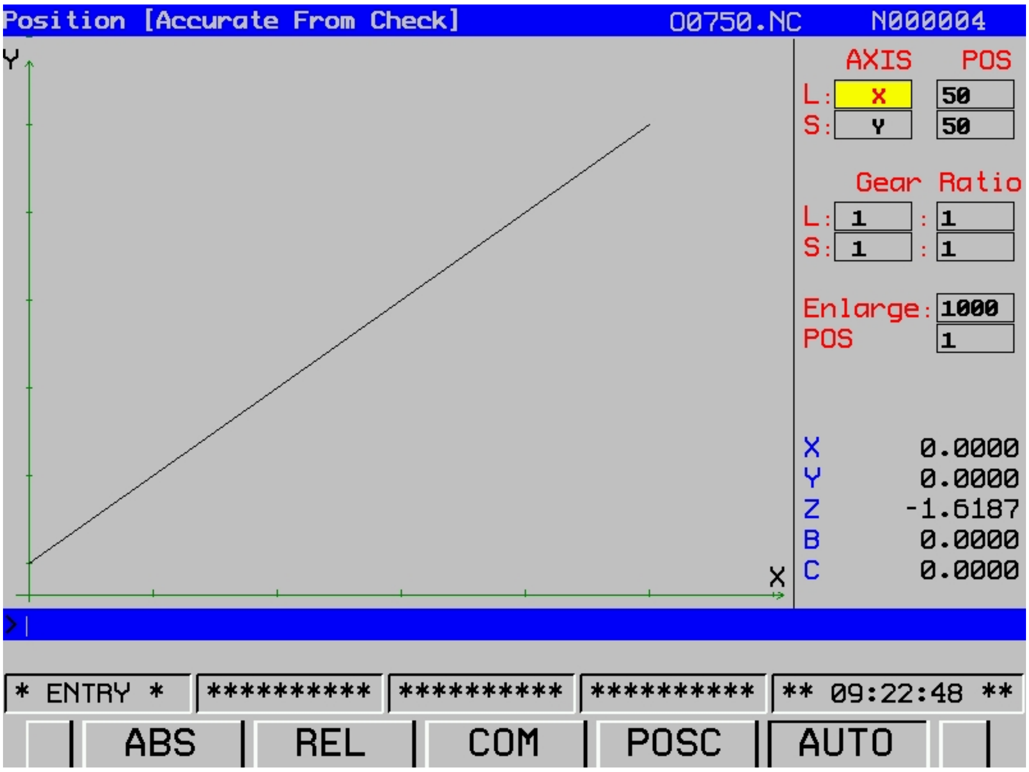


Fig. 3-1-5 Auto-check interface

3.2 Program Interface

Press 

key on panel to enter program page. This page includes five modes such as **【PROG】**, **【DETECT】**, **【DATA】**, **【FILE】** and **【OPT】**. When operation mode is MDI, **【DETECT】** interface changes into【MDI】, and each interface can be viewed and modified by corresponding softkey. Detailed information is as follows:

3.2.1 Program Display

Press **【PROG】** key to enter program display interface. Current program block in memory unit is displayed in this interface (see Fig. 3-2-1)

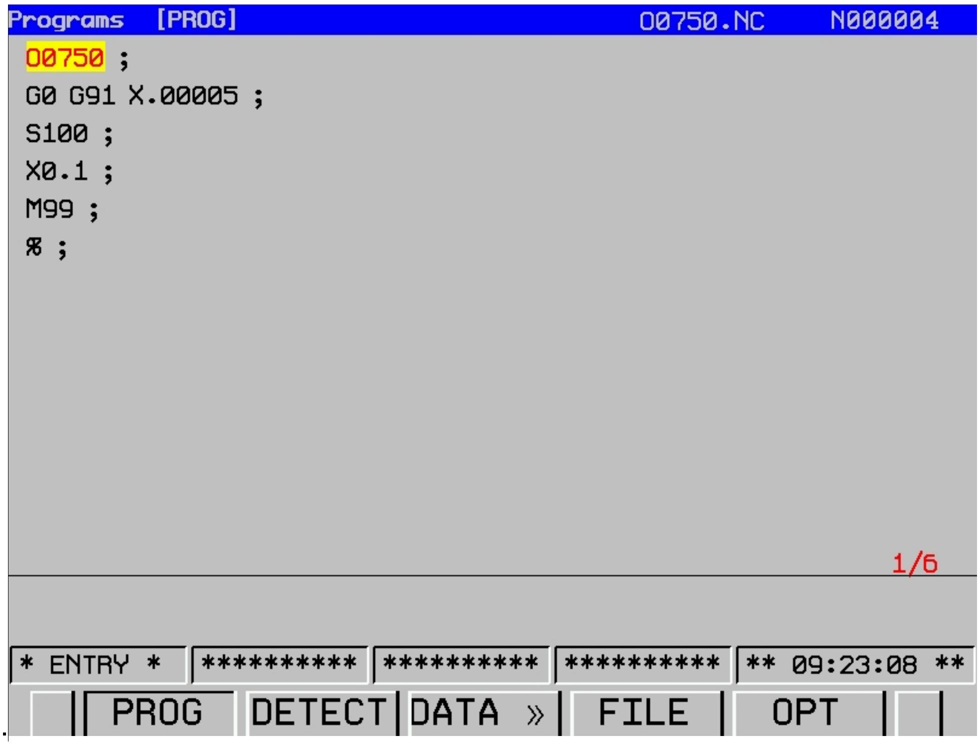


Fig. 3-2-1

Note: 1）00205.nc indicates program name（This refers to the program name on file list.）. N indicates line number of actual executing codes for current executing codes.

2）1/5, 1 is line number of current execution, 5 is total lines.

3.2.2 Setting a Program

This section introduces program editing which includes program setting-up, character insertion, alteration, deletion, duplication, stick and replacement. Moreover, deletion of a whole program and sequence number automatic insertion functions.

The section also includes search of program number, sequence number, word and address during program edit.

Part Ⅱ Operation

Steps of setting a program:

1. Press  key to enter edit mode, and press  key to enter【**FILE**】interface.

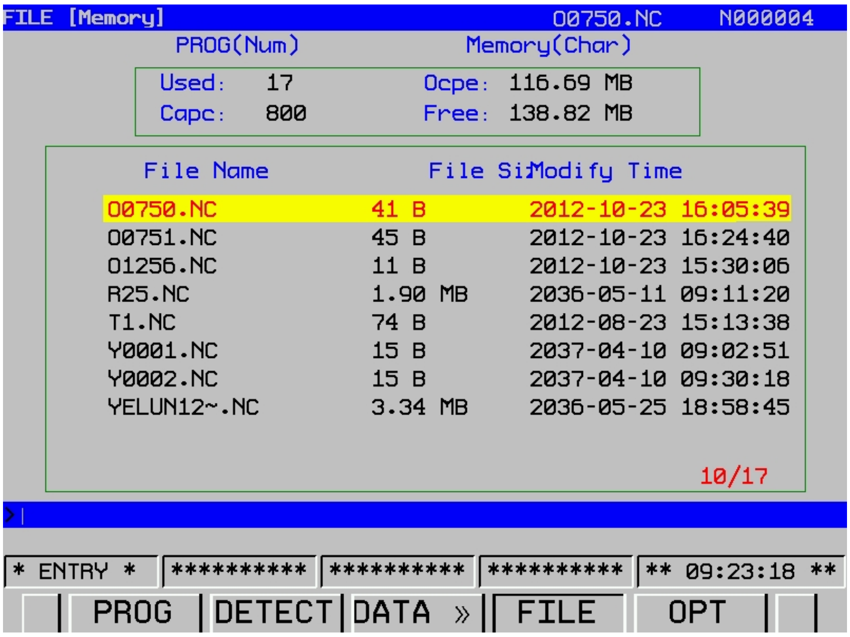
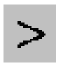


Fig. 3-2-2 File list interface

2. Press softkey【**OPT**】->  to enter the following interface.

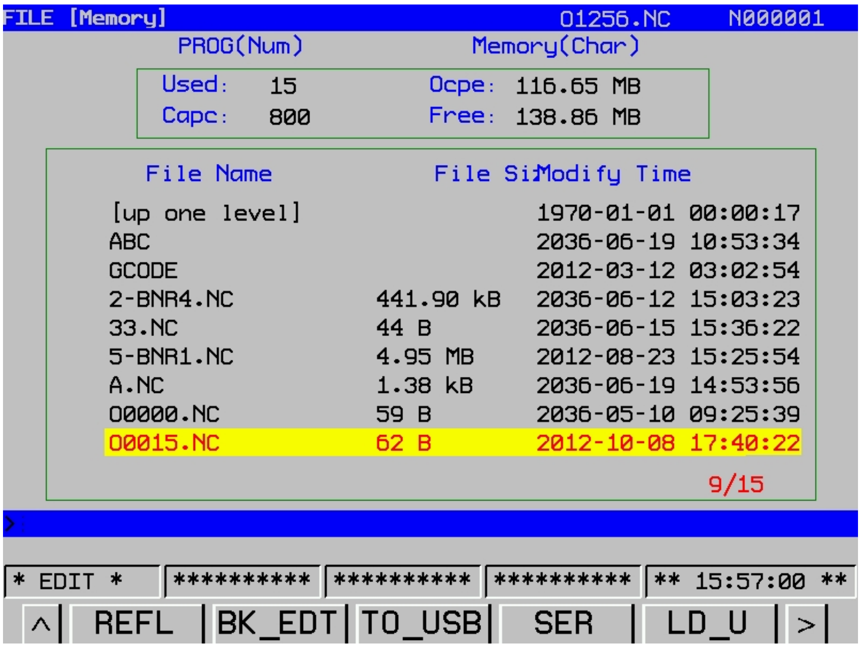
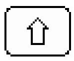
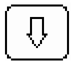


Fig.3-2-3

Fig.3-2-4

Note:

（1）**【up one level】** in figure indicates entering the list of the last level

（2）When the cursor move to the file by pressing the cursor key , , the key 【OPEN】at the lower side of the screen will be changed to **【LOAD】** automatically.

（3）In edit mode, only NC programs less than 8M can be loaded. The one greater than 8M can be loaded in DNC mode.

（4）It is suggested that newly-built program name should be within 8 characters. If exceeds 8 characters, the 8th character will be displayed as “~”and following characters will be omitted.

3. For example: input program name 0749 into the input column, and press **【NEWP】** to build a new program.

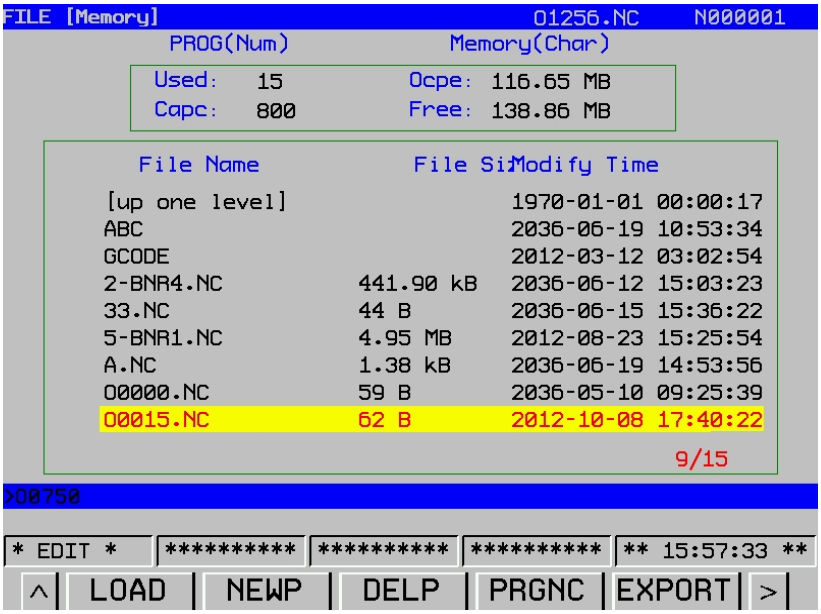


Fig. 3-2-5

4. New program is loaded by pressing 【**LOAD**】softkey, see the following figure:

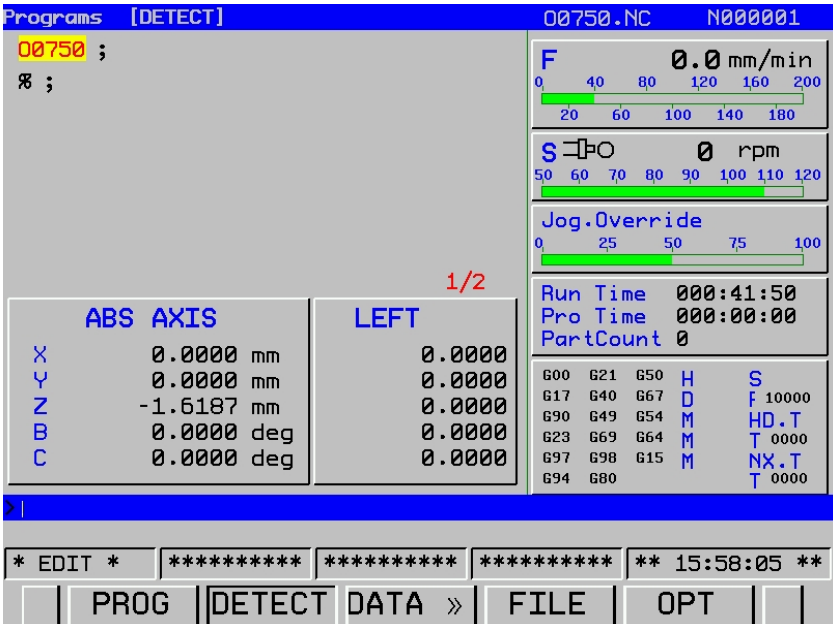


Fig. 3-2-6

Note:

1. When loading new program, parameter 100#3 decides whether to save edited file in the current edit interface.
2. When creating a new program name, only program name headed with O is allowed. If it is headed with other characters or figures are input directly, invalid input will be displayed on the input column.
3. When editing the program, program content in the input column is more than 60 characters or exceeds display area, system will prompt: invalid input.
4. When NC file name is more than 8 characters, only 7 characters can be displayed and the following ones will be replace by ~.NC.

Example: File name O12345678.NC will be displayed as O123456~.NC.

3.2.3 Edit a Program

Editing, selection and deletion of part program can be done by operation panel.

The sequence number will be inserted between blocks automatically when editing the program. Increment of the sequence number is decided by parameter No: 1621. User can set the parameter as requirements. See parameter manual for detailed instructions.

Methods of editing a program:

1. Part program editing should be done in Edit mode. Perform it according to the program editing sequence and key in input command or axis movement command to the input column.

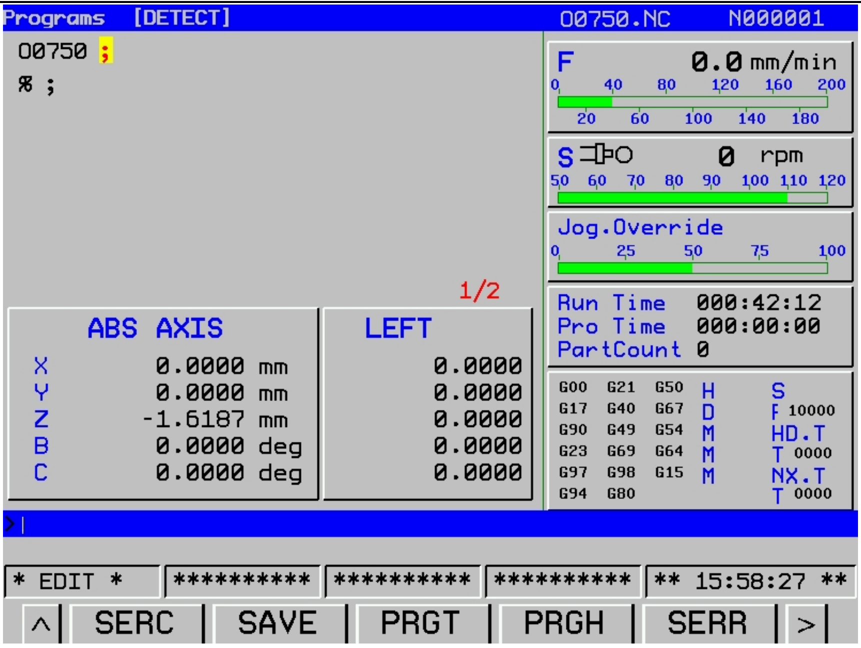


Fig. 3-2-7

2. Press EOB -> INPUT key on the operation panel, and edit is completed. See the figure below:

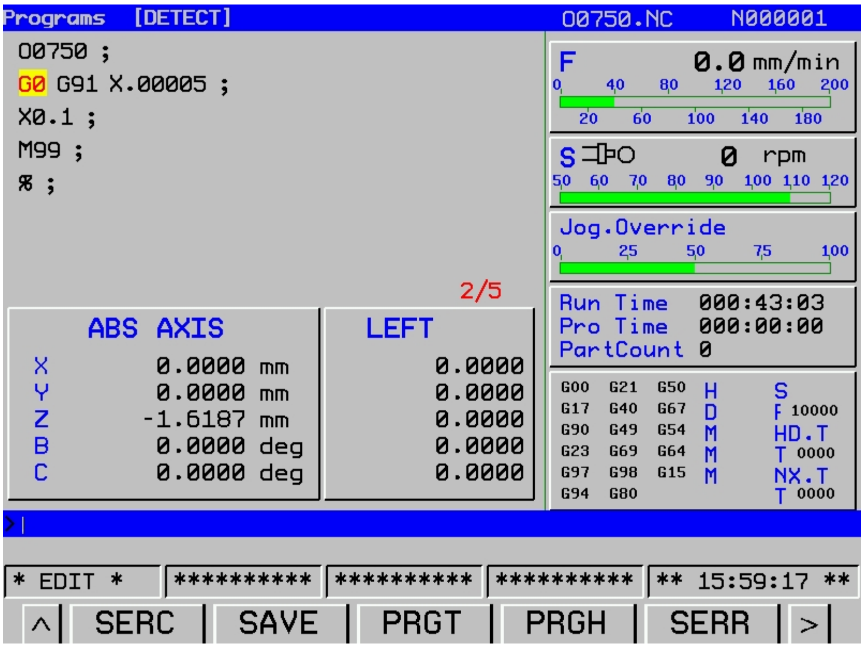


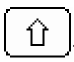
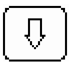
Fig.3-2-8

Notes:

（1）To make block search much easy, the system provides **【SERC】**, **【SAVE】**, **【PRGT】**, **【PRGH】**, **【SERR】** keys for program edit.

（2）The way of using **【SERC】**

① Press key **【SERC】** continuously to key in codes or axis movement commands. The contents will be searched repeatedly in program interface.

② Input codes or axis movement command, press  to search it upwards to the front of program, or press  to search it downwards to the end of program. After single-direction or cycle searching is finished, flashing alarm will occur: The searched content does not exist.

Part Ⅱ Operation

Example: To search code S1000.

A. Press S -> 1 0 0 0

B. Press cursor key ⇩ to search the back part of the program. Press key ⇧ to search the upper part.

（3）The way of using **【PRGH】**, **【PRGT】**:

Press **【PRGH】** or **【PRGT】** in edit interface, the interface will immediately change to the head or the tail of the program to be edited.

（4）The way of using **【SERR】**:

【SERR】 is used for searching certain line of block from the whole program. For example: To search the 55th line from the program edit interface, key in 55 to the input column, and press **【SERR】**, the 55th line will be searched by the system.

Example: search line number 55

A: Press 5 5 on the panel.

B: Press the key **【SERR】**, and the 55th line will be searched by the system.

（5）The way of using **【SAVE】**:

Check whether the program in edit interface are completed, then press softkey **【SAVE】->【Enter】**, the program is saved.

Character insertion, replace and deletion

The section will describe insertion, replace and deletion for the program input in memory.

Steps for insertion operation:

1. Select EDIT mode, and press function key PROGRAM.

2. Enter file list interface and select the program to be edited.

If the program is loaded, enter the program edit interface, and execute the item 3.

Note: If the program is not selected, press cursor key ⇧, ⇩ to select program and press key **【LOAD】**.

3. Search the character to be altered.

Search the program by **【SERC】** or **【SERR】**.

4. Insertion includes character insertion and block insertion

Character insertion: search the address of the block to be inserted, and press cursor key ⇨, ⇦ to select it. Then input character in input column, and press INSERT key on the panel. The character will be inserted in the specified block.

Example: Inserting character S100. See Fig. 3-2-9 below:

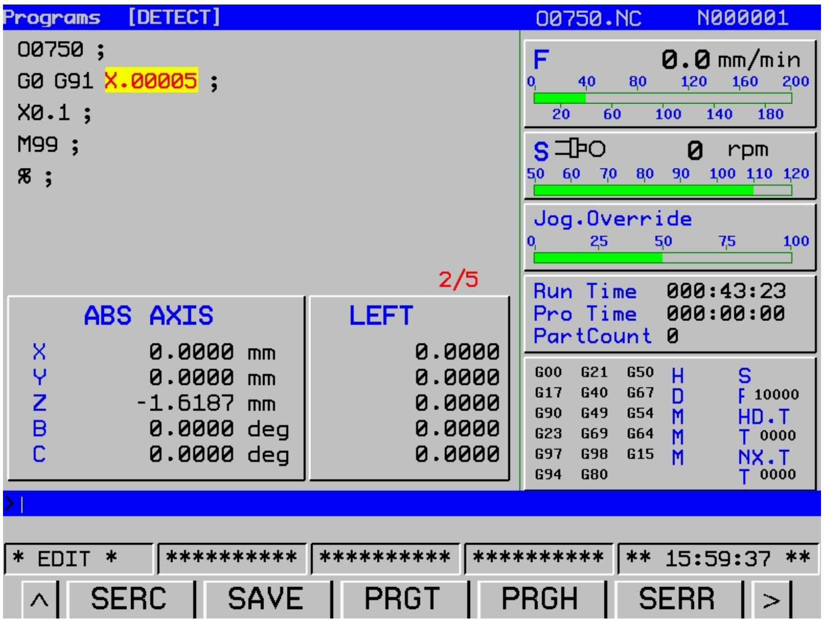


Fig. 3-2-9

After insertion, the interface will be:

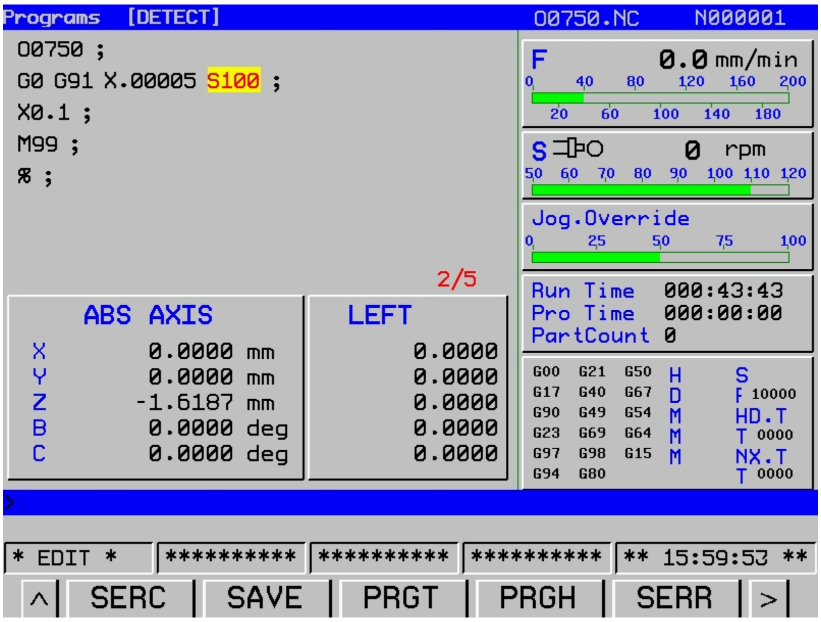
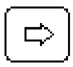
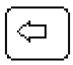
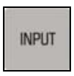


Fig. 3-2-10

Block insertion, search the address of the block to be inserted, and press key ,  to move the cursor to the semicolon of the last block. Edit block to be inserted in the input column and press key 

-> 

on the panel to finish the insertion.

Example: Inserting block M3 S100

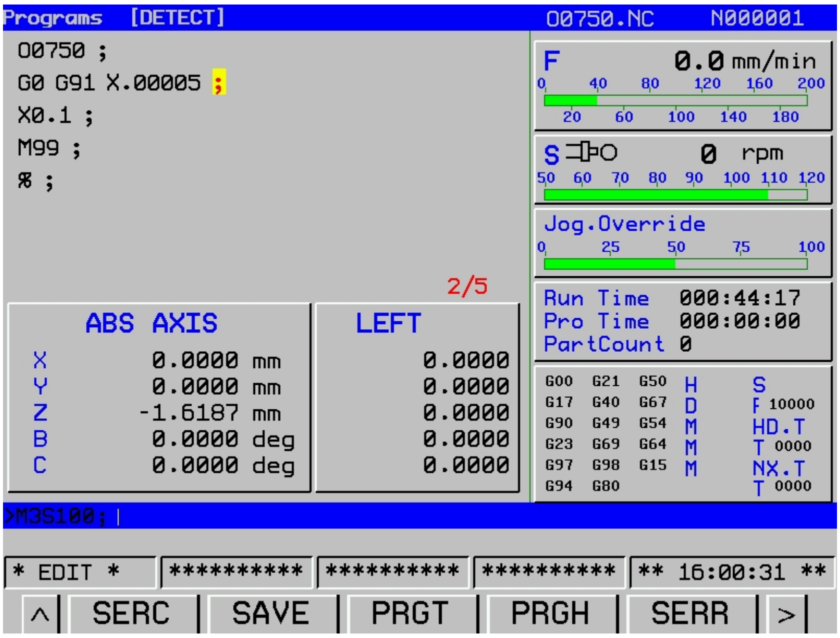


Fig. 3-2-11

After insertion, the interface will be:

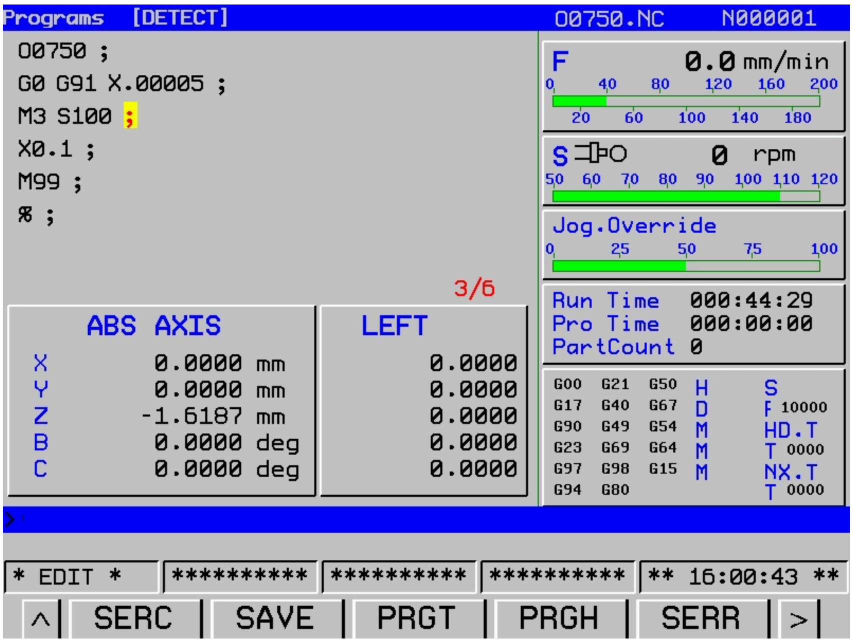
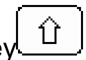
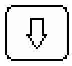


Fig. 3-2-12

Steps of replace operation:

1. Search character to be replaced in program edit interface.
2. Input the character to be altered and press cursor key ,  to search it.
3. Press 

key on the panel to replace the original character.

Example:

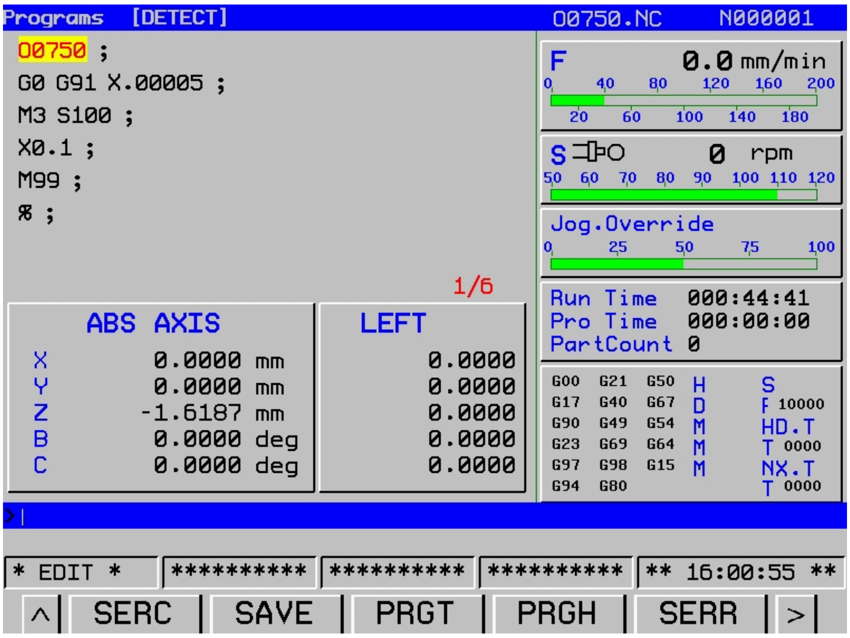
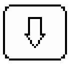
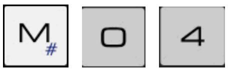


Fig. 3-2-13

Detailed operations are as follows:

A. Press key  to move the cursor to M3.

B. Input  to the input column.

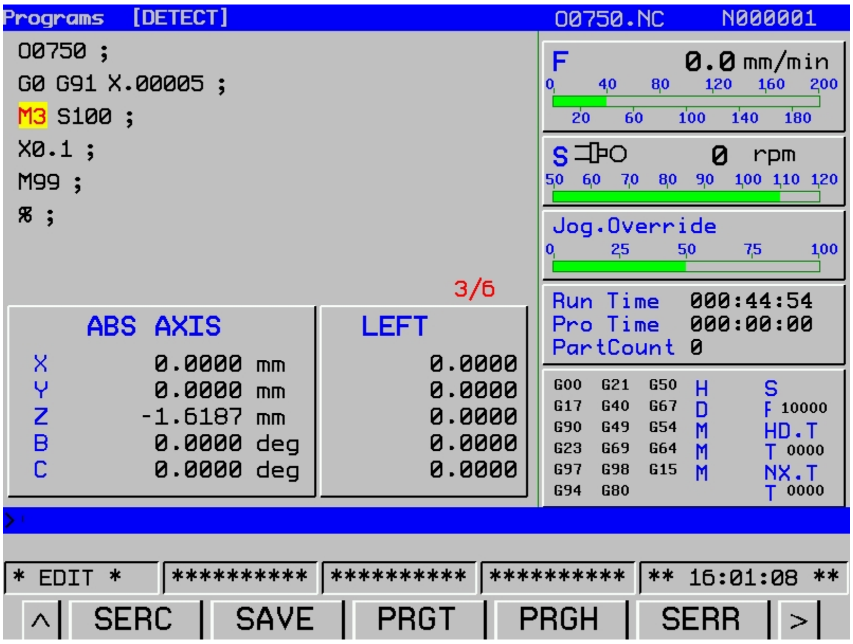


Fig. 3-2-14

Part Ⅱ Operation

C. Press  on the panel.

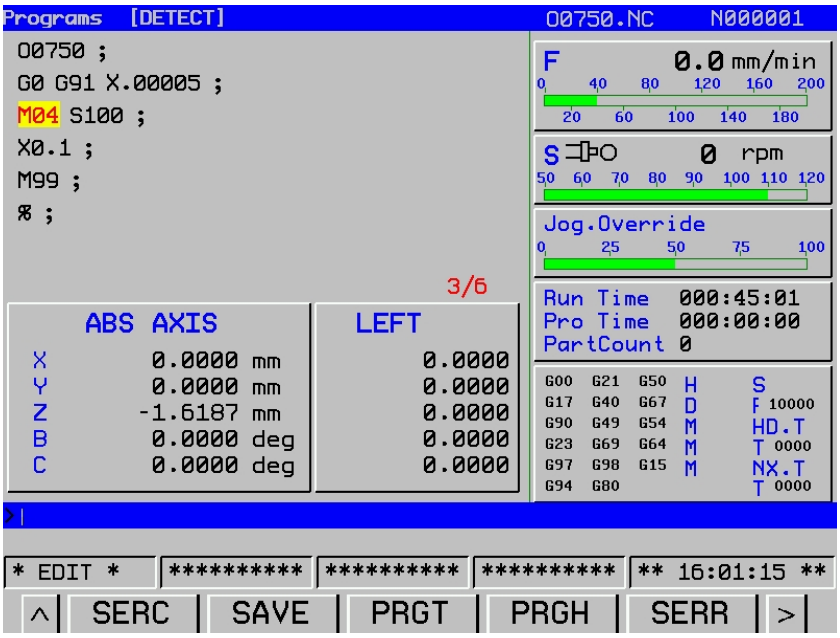
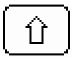
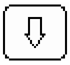


Fig. 3-2-15

Steps of deletion:

1. In  mode, enter program edit interface

2. Press  or  to move the cursor.

Example: deletion of character M04

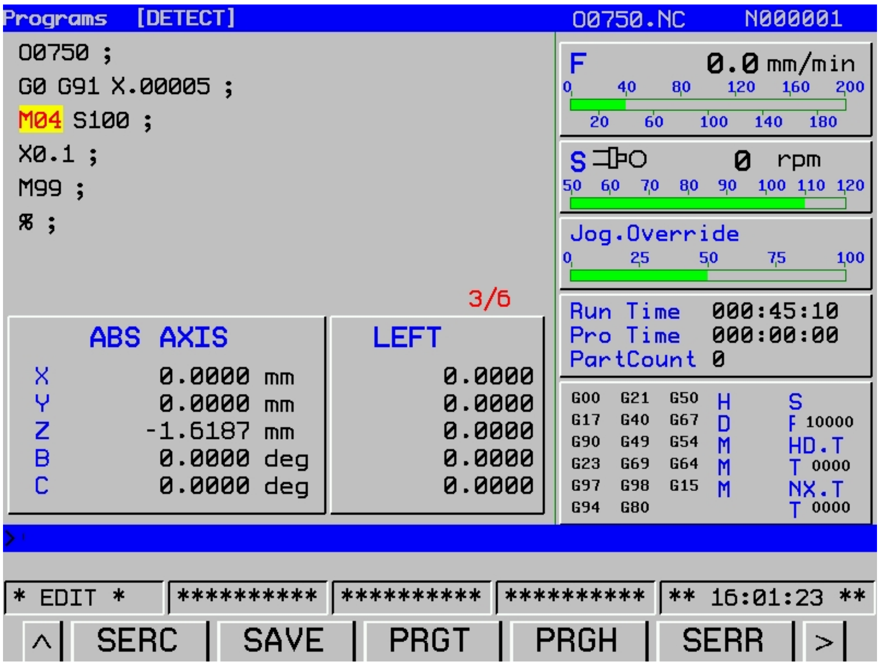


Fig. 3-2-16

Part Ⅱ Operation

3. Press 

on the panel.

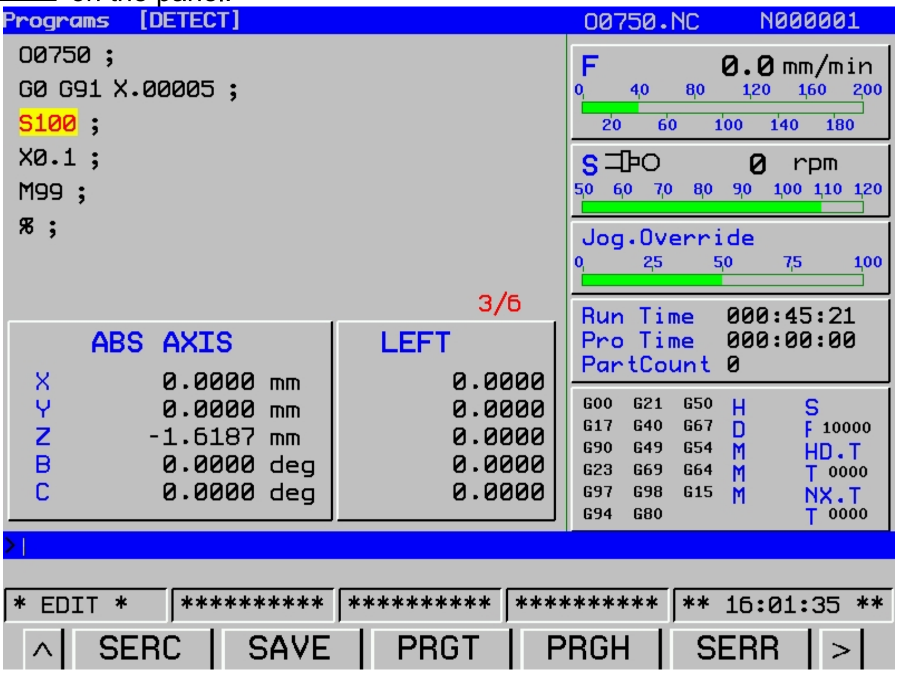


Fig. 3-2-17

Copy, cut, paste, delete and alteration in expansion edit.

By using expansion program edit function, the following operations can be performed for the program saved in the memory.

1) A whole program or part of the program can be copied or moved to the other program.

2) A program can be combined with any part of the other program.

3) Specified character or address of a program can be replaced by other character or address.

1. Steps for multiple-line copy

(1) In 

mode, load a newly-built program. See the following figures. For example, loading file O0745.NC.

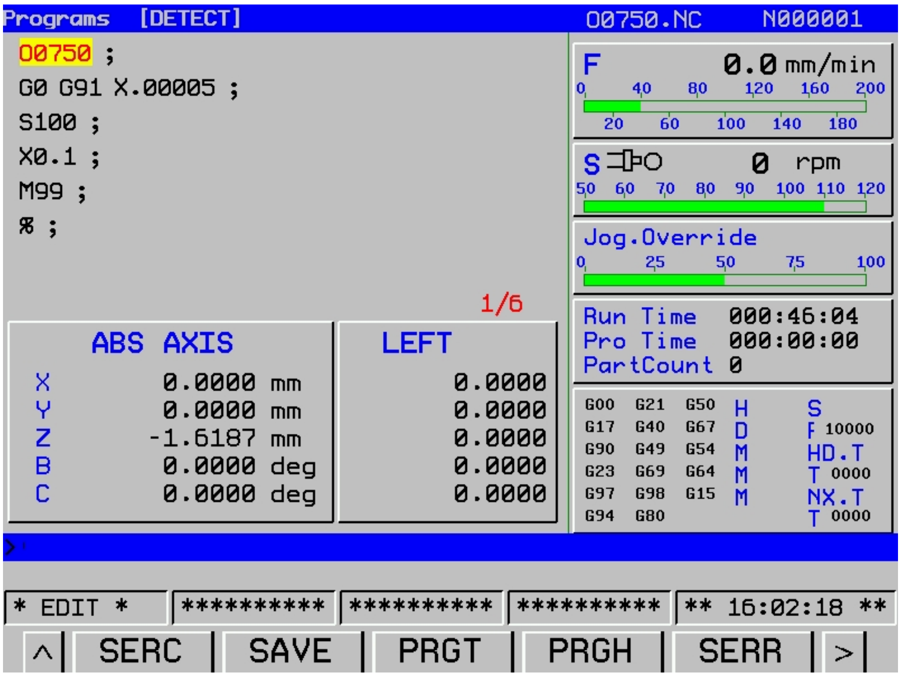


Fig. 3.2-18

(2) Press key 【OPT】 to enter the following figure.

(3) Press key > to enter the following figure.

(4) Press page key , or cursor key , to move target line of copy. See Fig. 3-2-19.

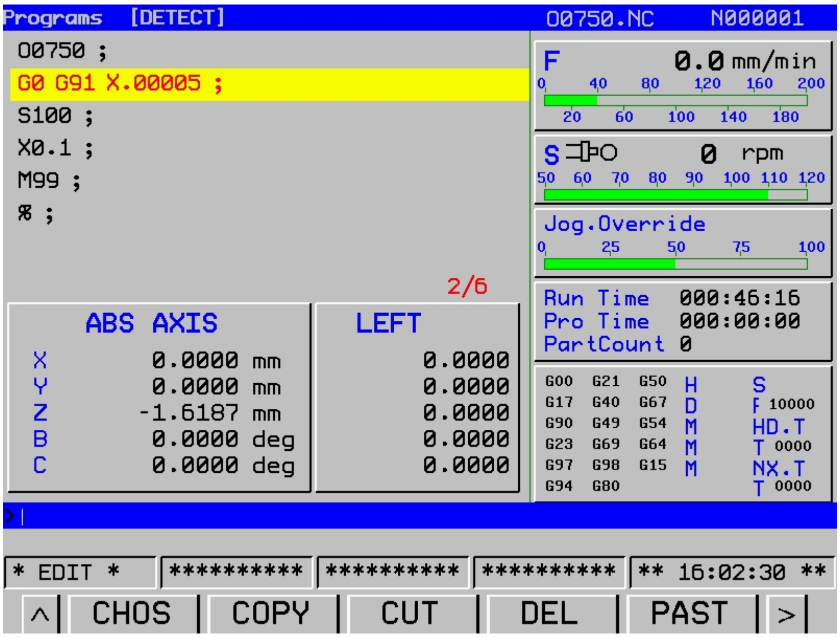


Fig. 3-2-19

Press softkey【CHOS】and then press key to move the cursor to the target line, the cursor will select all lines until %. See Fig. 3-2-20.

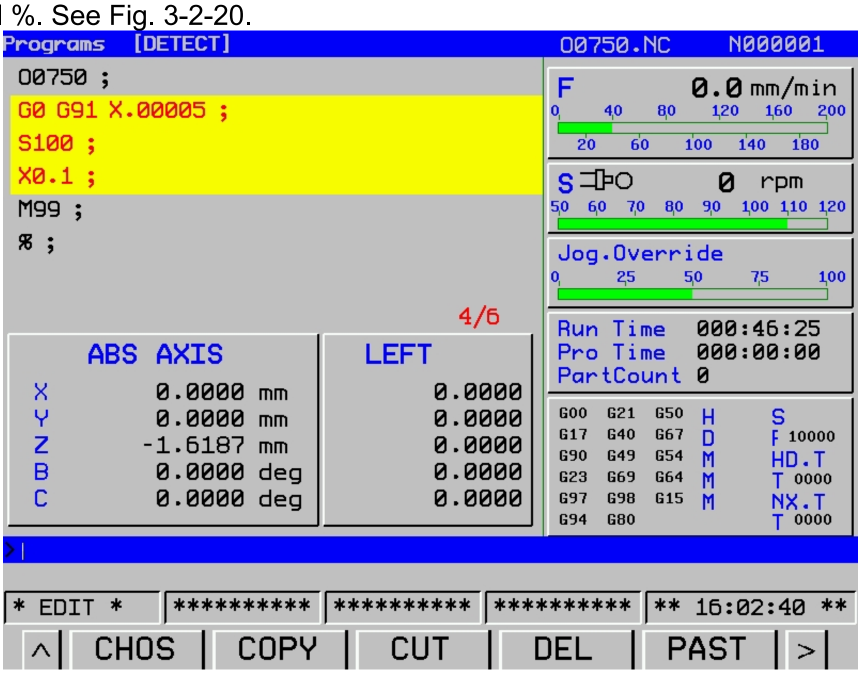


Fig. 3-2-20

Part Ⅱ Operation

(4) Press key 【COPY】and then press【PAST】to finish copy. The following is comparison figure before and after paste operation.

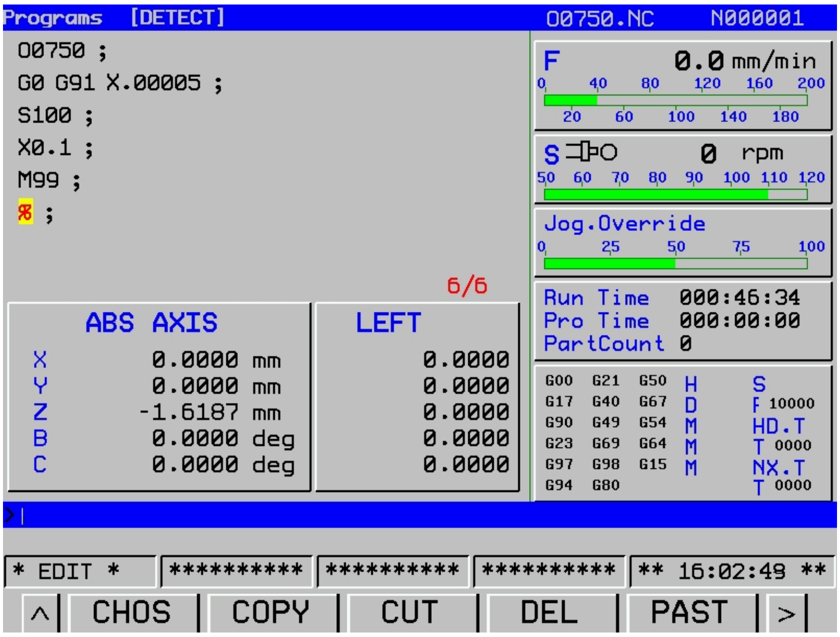


Fig. 3-2-21

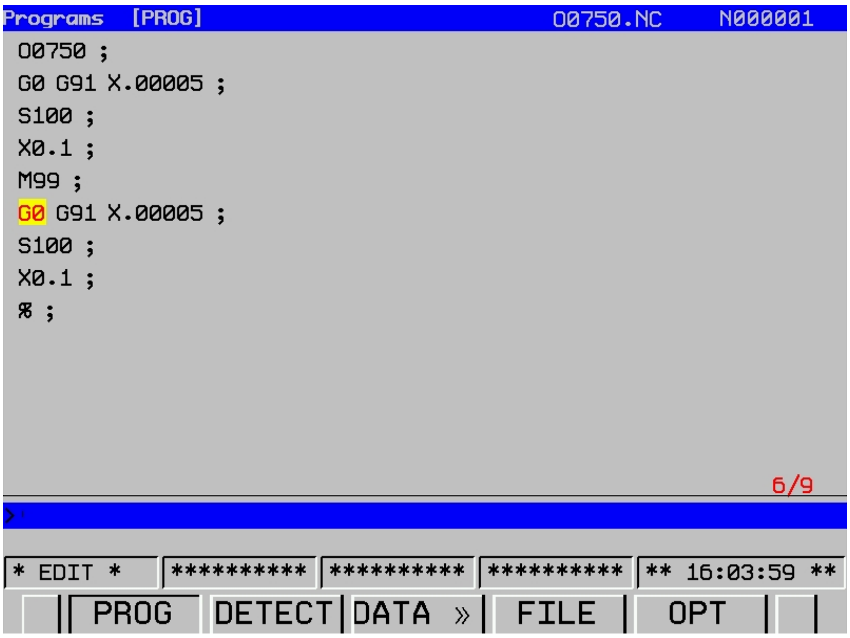


Fig. 3-2-22

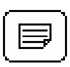
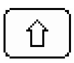
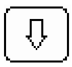
Note:

(1) When copy program, the selected contents should not include the first line (program name) and the last line (%). Otherwise, copy operation will not successful.

(2) For multiple-line program copy, the system specifies that only 5000 lines can be copied.

2. Steps of cut operation

(1) Perform items 1-3 of the copy operation and enter the figure below.

(2) Press page key ,  or cursor key ,  to move the cursor to the target block.

Part Ⅱ Operation

Press softkey【**CHOS**】and press cursor key ⇩ to move to the block as shown in the figure below:

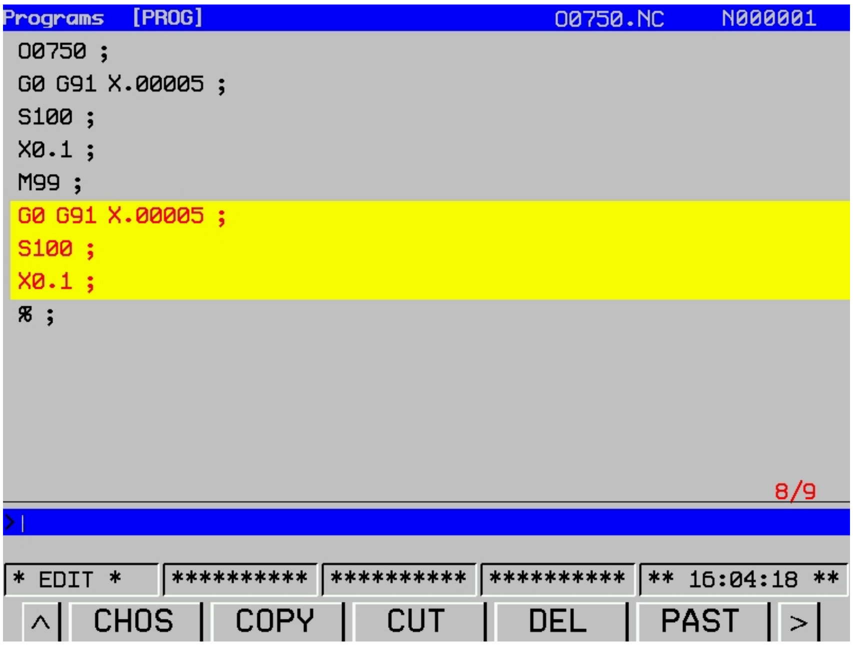


Fig. 3-2-23

（3）Press softkey 【**CUT**】, and return to the program list to set a new program name and press 【**LOAD**】, then press【**PAST**】key to complete program cut operation. See the figure below:

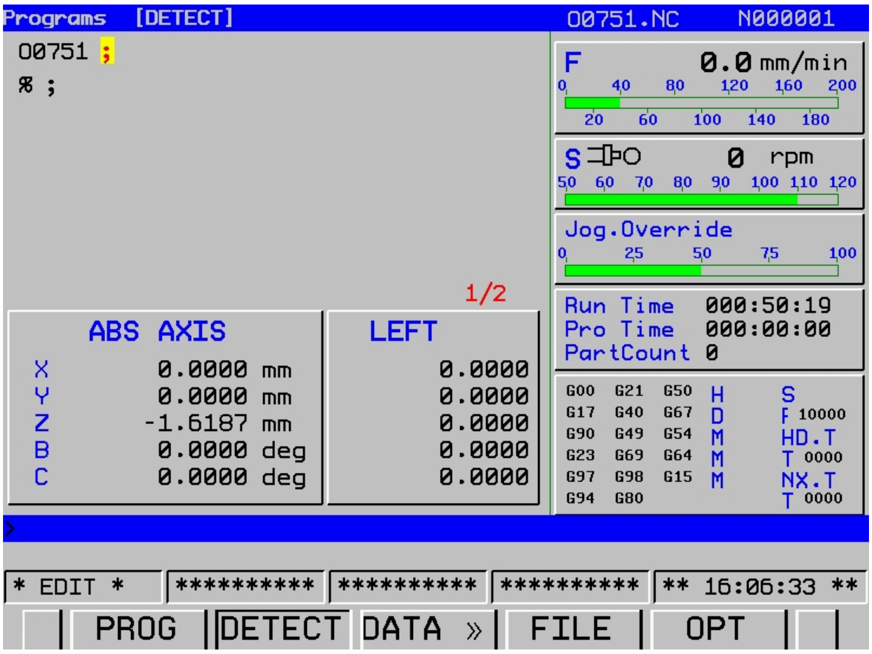


Fig. 3-2-24

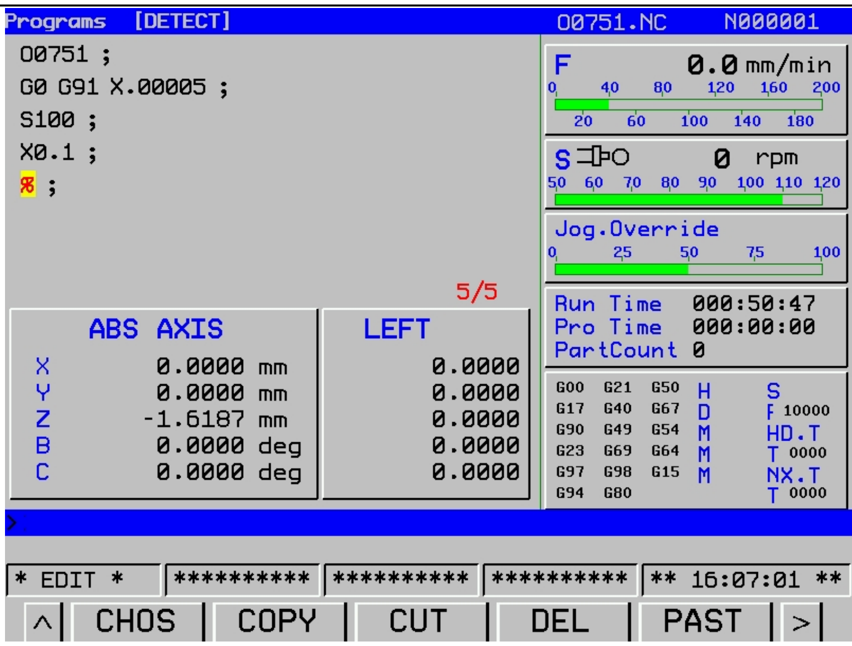
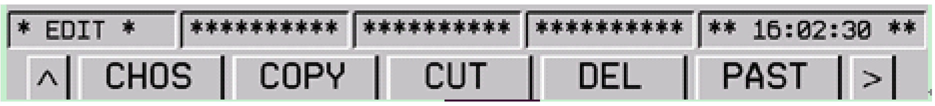
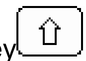
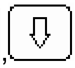


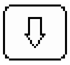
Fig. 3-2-25

3. Steps for program deletion:

(1) Perform items 1-3 in the copy operation and the following figure displayed:



(2) Press page key ,  or cursor key ,  to move the cursor to the target block.

Press key 【CHOS】and key  to move the cursor to the block. See the figure below:

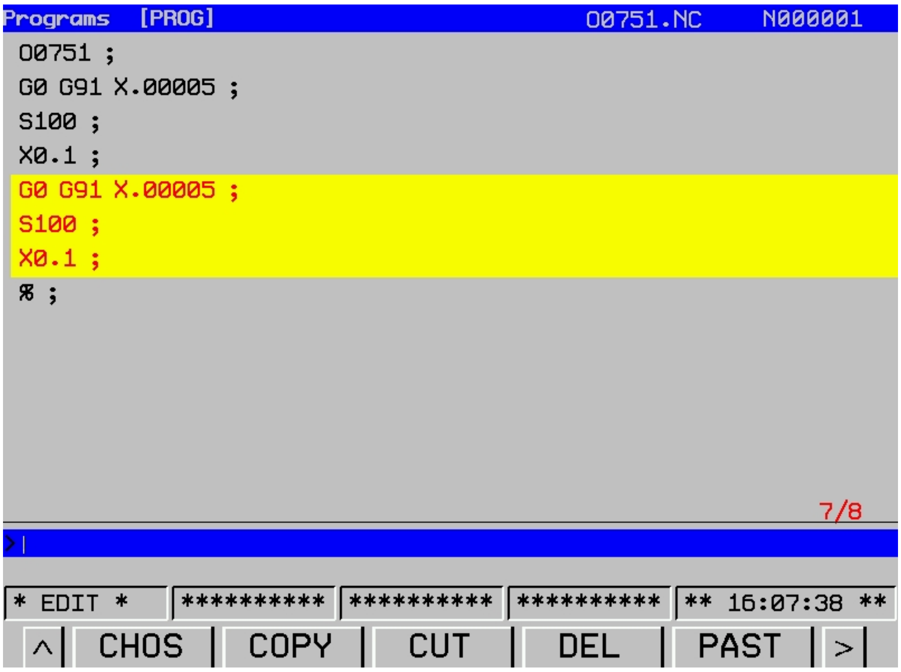


Fig. 3-2-26

(5) Press key 【DEL】, the selected block will be deleted.

Steps for paste operation:

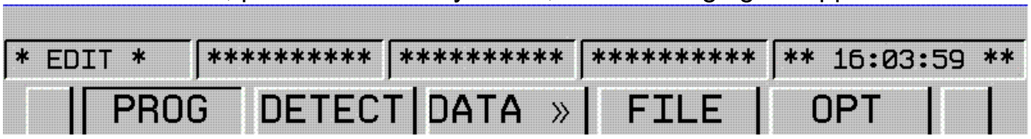
1. According to copy and cut operation steps mentioned above, paste the selected block.
2. Paste the program to the specified position.

Note:

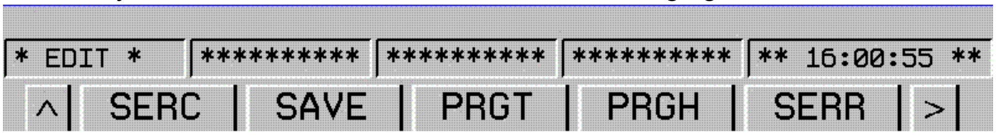
1. Only after copy and cut functions are selected, **【PAST】** softkey will appear on the newly-built program. Otherwise, **【PAST】** will not appear.
2. Because the operation of copy the first line is invalid, and paste operation is invalid also.
3. After copy operation, click **【PAST】**key, copied contents are pasted to the line before the cursor locates.

Alteration steps:

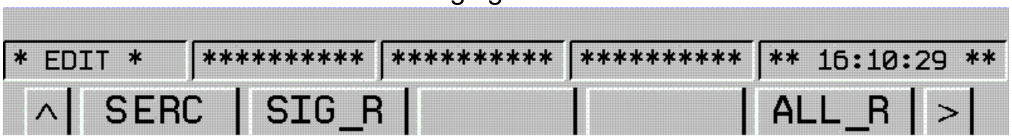
1. In EDIT mode, press function key PROGRAM, the following figure appears.



2. Press key **【PROG】->【OPT】** to enter the following figure.



3. Press ⊳ twice to enter the following figure.



4. Press page key , or cursor key , to the target position. Or input target code and press **【SEARCH】** key to search the code directly.

The following is comparison figure before and after search operation.

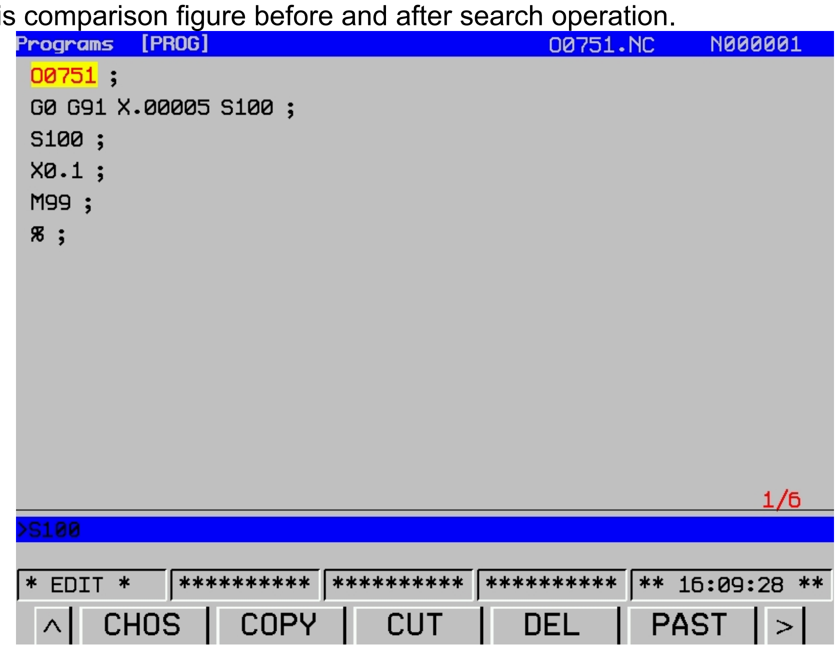


Fig. 3-2-27

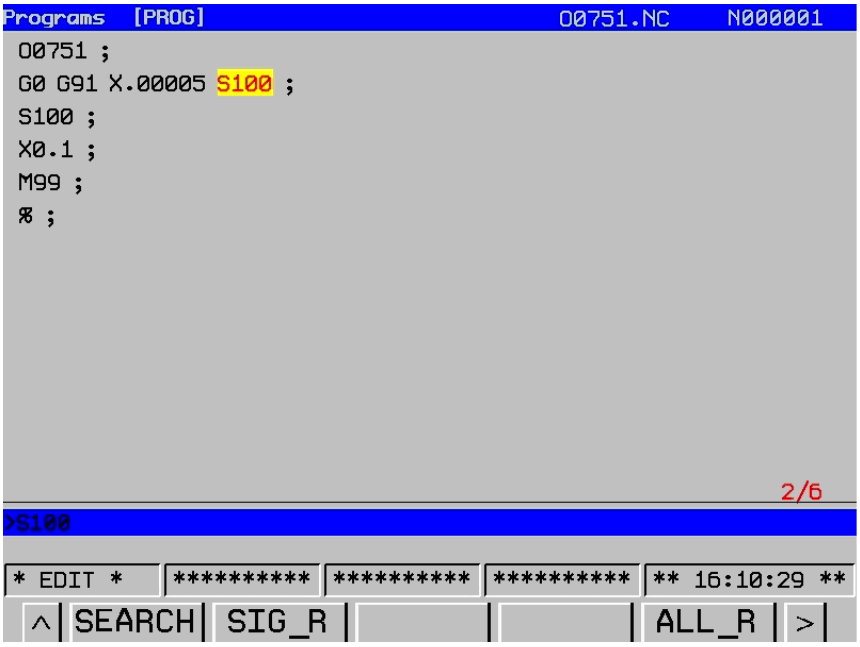


Fig. 3-2-28

At this time, if directly press 【SEARCH】cursor, the next target code will be found. The operation can be performed until the end of the program without inputting the same target code.

Note: To realize continuous search, the system will keep the last searched record.

5. After the target code is found, input the contents to be altered, and press key **【SIG_R】**, the contents where the cursor is will be replaced by new one. For example: in the figure below, input S200, and press **【SIG_R】**, the contents on the position where the cursor locates will be changed into S200.

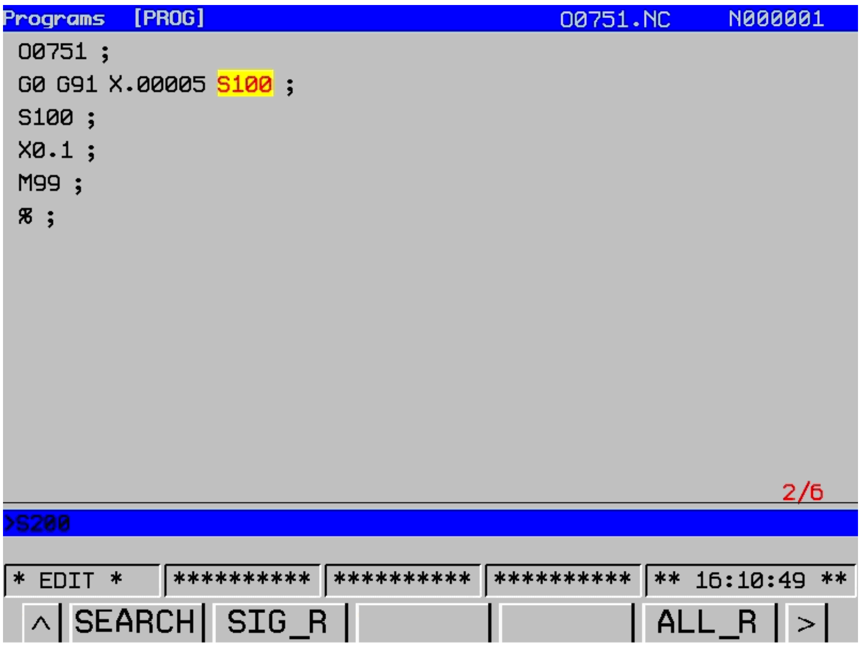


Fig. 3-2-29

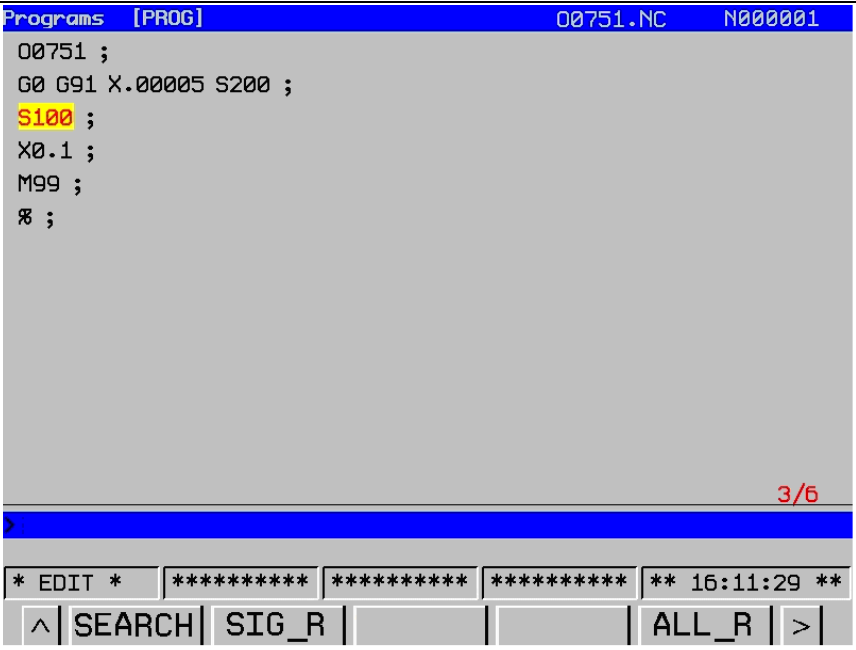


Fig. 3-2-30

At this time, move the cursor to S100 of the next line, and directly press key 【SIG_R】, and S100 is replaced by S200. As the figure below:

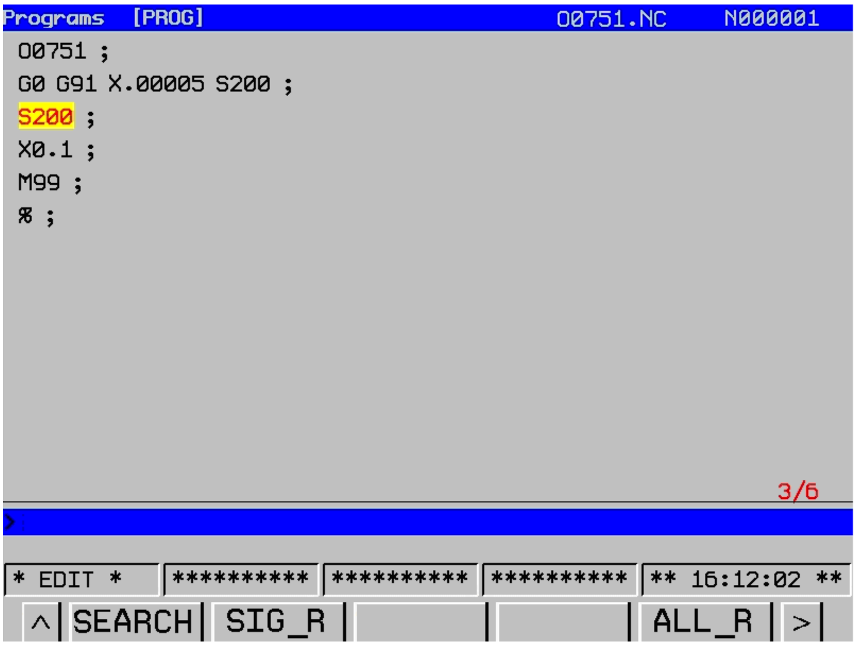


Fig. 3-2-31

Note:

（1）To make continuous alteration more easy, the system will keep the last alteration record automatically.

（2）The last alteration contents kept by system and the contents searched are different. Two records are stored in two different files on design.

6. If all same characters or addresses to be altered, perform the following operations:

①. Press cursor key ⇧, ⇩ (or perform as the methods introduced above) to find the contents which should be altered, and input the new contents to the input column. Input S300 as the figure below:

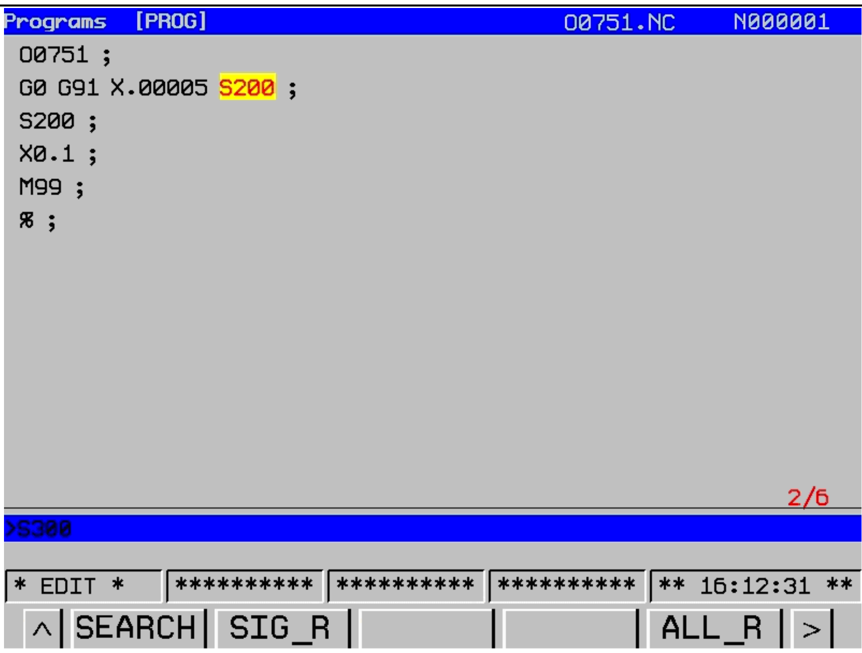


Fig. 3-2-32

②. Press key 【ALL_R】, the same contents will be altered by the system automatically from the position where the cursor locates to the end of the program.

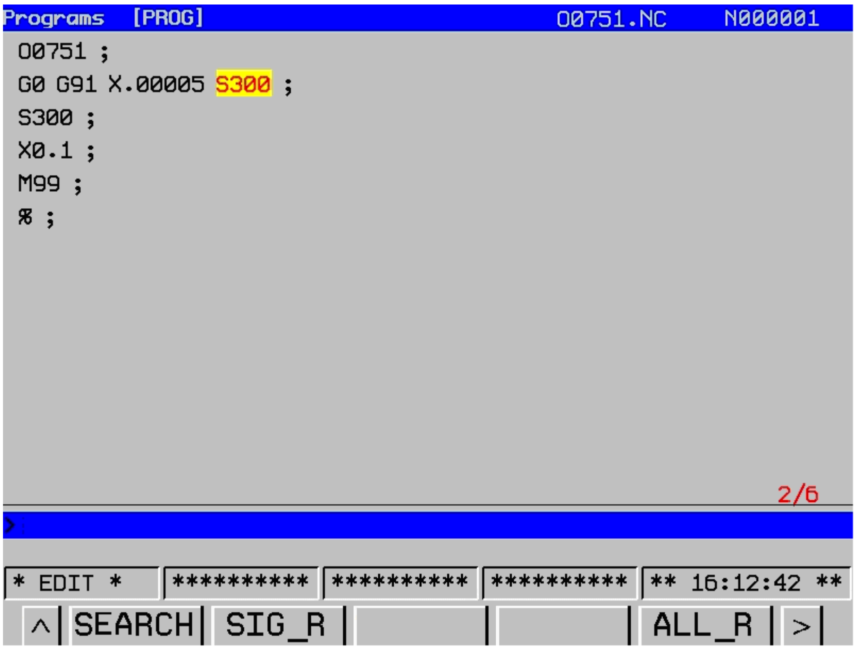


Fig. 3-2-33

As the figure above, move the cursor to S200, input S300 and press【ALL_R】key. All S200 will be changed into S300 from the position where the cursor is to the end of the program.

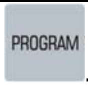
Pay attention that 【ALL_R】doesn't mean clean all contents of the line where the cursor is, but replace the content where the cursor points and the following contents same with it.

Deletion of the program

The program saved in the memory can be deleted one by one or multiple of them can be deleted once.

Note: The function of deleting programs once is only used for the programs headed with O.

1. Steps for deleting a program

（1） In  mode, the following figure appears by pressing the functional key .

（2）Press down【FILE】key to enter the following figure.

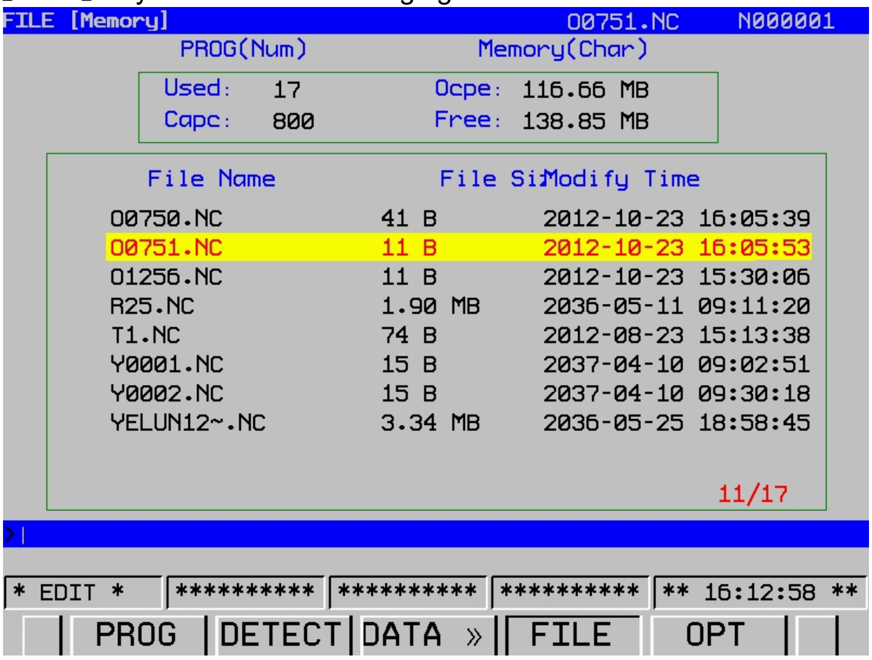
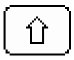
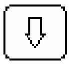


Fig. 3-2-34

（3）Press page key ,  or cursor key ,  to select the program to be deleted. As above figure shown, select O0746.NC.

（4）Press 【OPT】key on the panel to enter file list interface. Press 【DELP】softkey to delete the selected file.

Fig. 3-2-35

2. Steps of deleting multiple programs:

（1）In **<Edit>** mode, the following figure appears by pressing the functional key .

（2）Press key 【**FILE**】to enter the following figure:

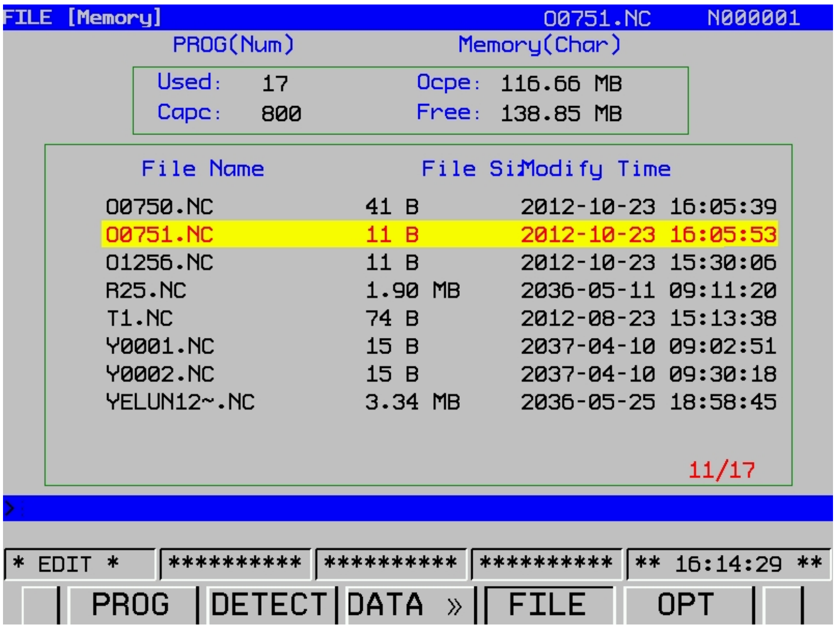


Fig. 3-2-36

（3）Input O-9999 to the input column, all programs from O0000 to O9999 are deleted.

Fig. 3-2-37

Note:

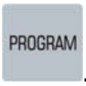
1. If all programs within 8000 need to be deleted, input O-8000.

2. When the parameter No.1610#0 and #4 are 1, programs from 8000 to 9999 can not be deleted even though the program is loaded. See parameter instruction for details.

USB disk transmission program

In order to meet customers' requirements, the system provides USB import to make USB disk transmission easy.

1. Steps of transmitting programs from USB disk to NC:

（1）In Edit mode (it is valid only in edit mode), the following figure appears by pressing function key 

.

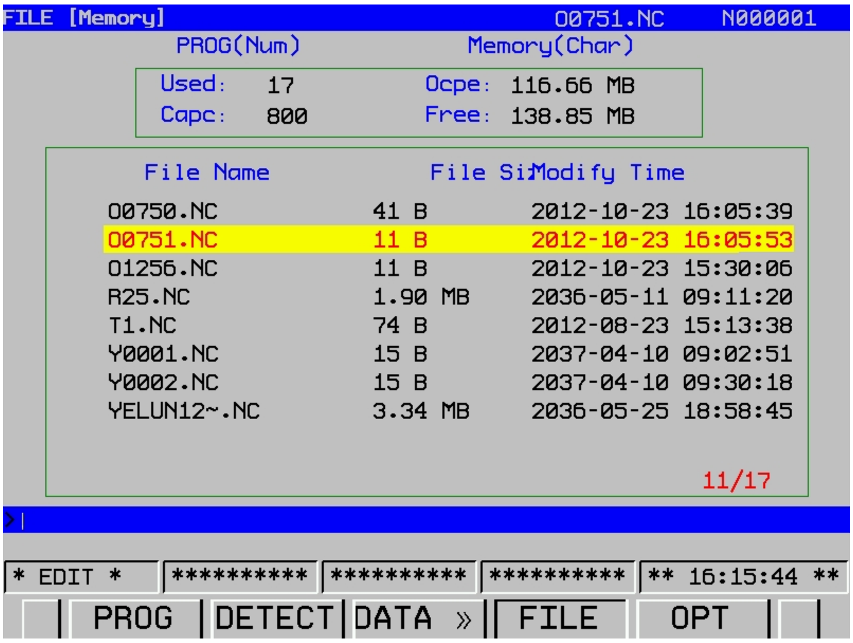


Fig. 3-2-38

（2）Insert USB disk, and press【FILE】->【OPT】to enter the following figure:

Fig. 3-2-39

（3）Press softkey > ->【LD_U】to enter list interface of USB disk files. As the figure below:

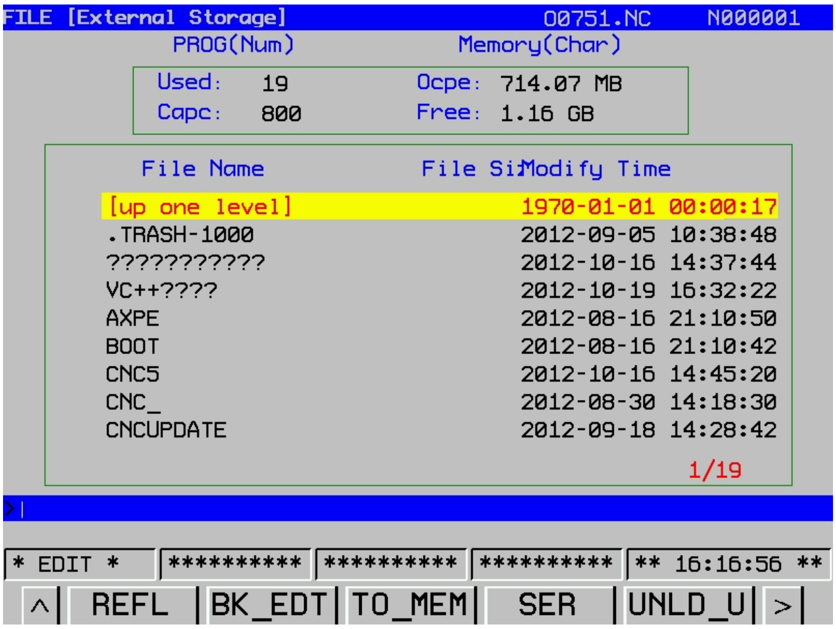


Fig. 3-2-40

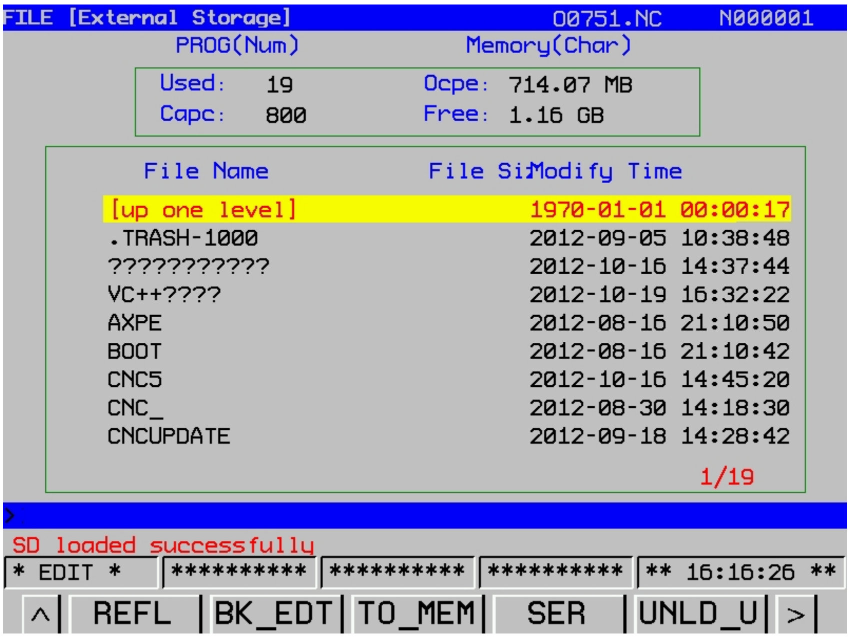


Fig. 3-2-41

(4) Press page key , or cursor key , to select the program to be input. Then enter the following figure by pressing key .

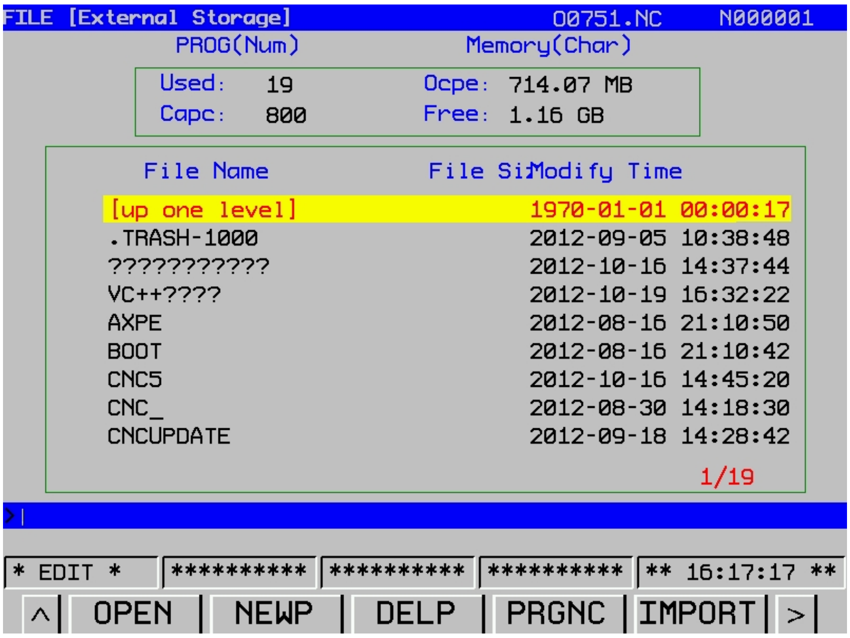


Fig. 3-2-42

(5) The selected program will be transmitted to the memory by pressing key 【IMPORT】. Then press key to enter the interface as below.

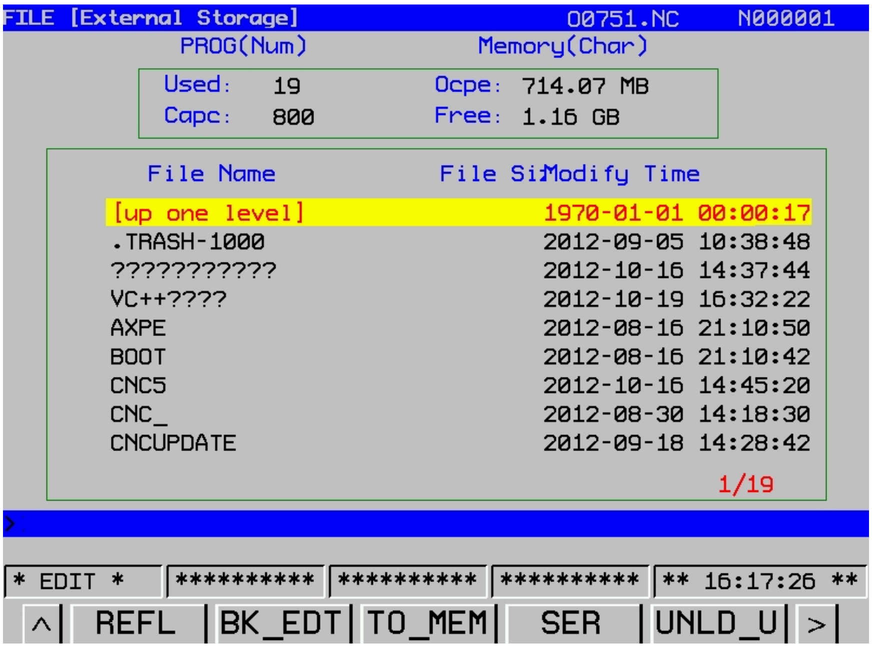


Fig. 3-2-43

(6) Press key 【TO_MEM】 to check the input program, and press key 【UNLD_U】 to unload the USB disk.

Note:

Currently, the system specifies that: When transmitting an empty file from USB disk to the CNC, the file name will not change after loading, and the file name automatically generated at the first line is O0000.

2. Steps of transmitting program from the memory to the USB disk

（1）Enter the following interface according to the steps of sending program from USB disk to the NC.

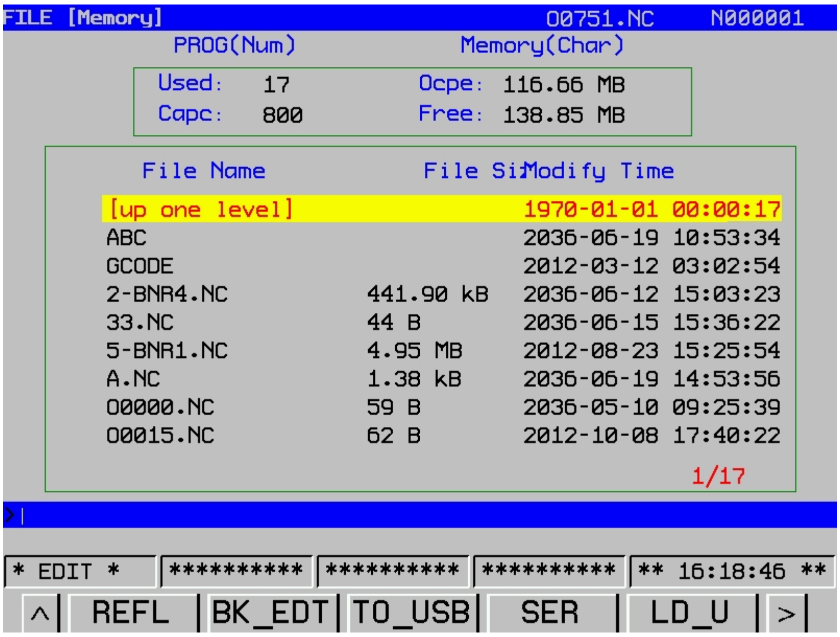


Fig. 3-2-44

（2）Press softkey【LD_U】to enter files list of the USB disk. As the figure below:

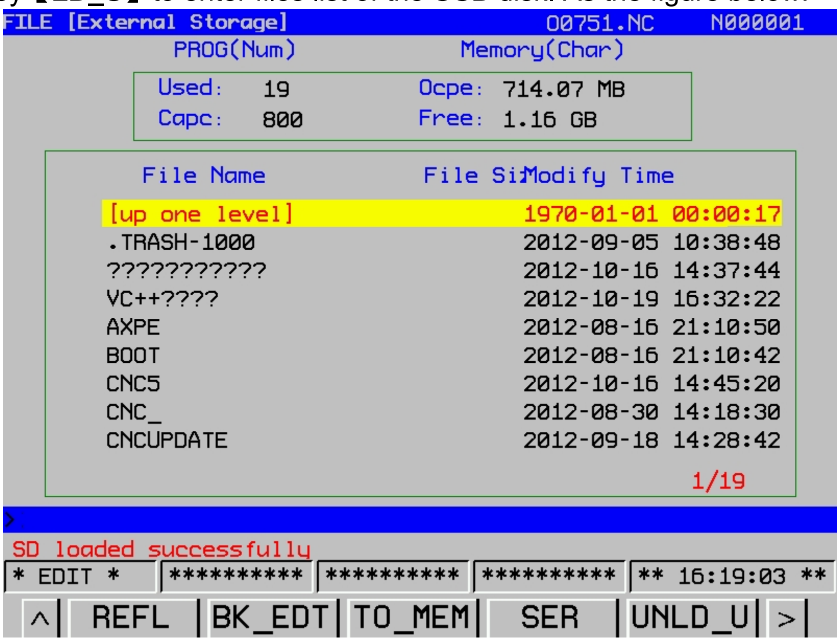


Fig. 3-2-45

（3）Press softkey 【TO_MEM】, select the program to be transmitted by pressing page key , or cursor key , .

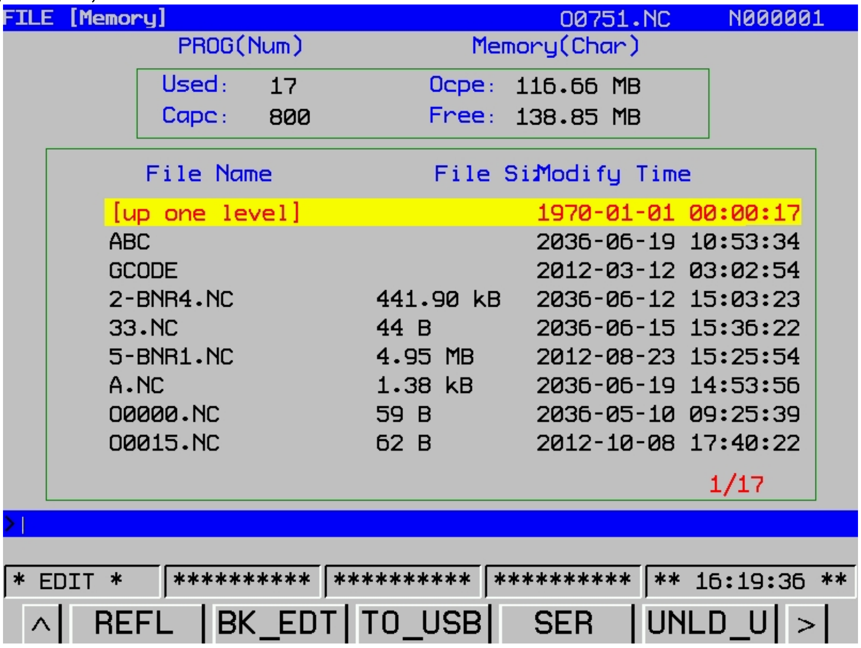


Fig. 3-2-46

（4）Press key > to enter the next interface.

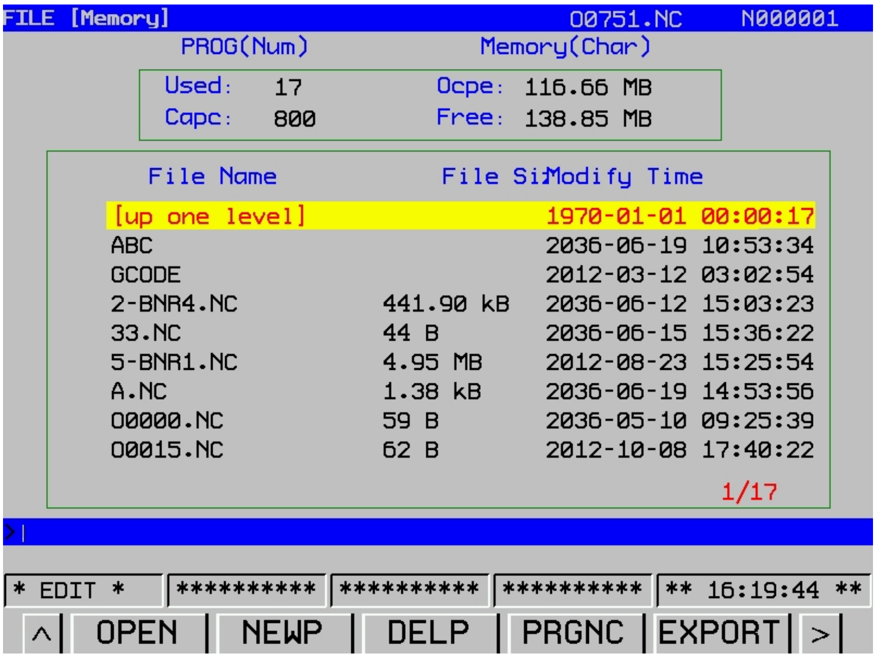


Fig. 3-2-47

（5）Programs are transmitted to the USB disk by pressing 【EXPORT】key. Press key > -> 【TO_USB】to check exported programs.

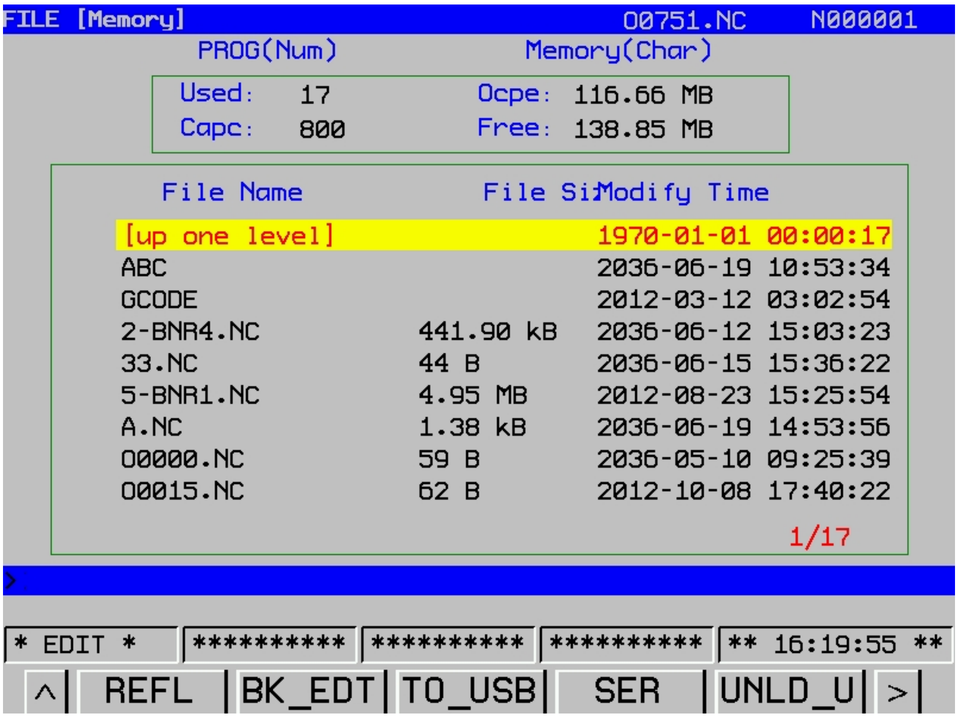


Fig. 3-2-48

Note: Only capitalized file name can be identified by the system, therefore, file name in small letters are automatically changed to capitalize one after loading. For example: file name ob.nc is changed to OB.NC after loading.

（6）Press【UNLD_U】to finish the transmission.

3.2.4 Cursor Positioning

In Edit mode, select PROGRAM key to enter program page.

a) Press ⇧ key to move up the cursor to the last line. When the line which the cursor is in is bigger than the end row of the last line, the cursor can be moved up to the end of the last line.

b) Press ⇩ key to move down the cursor to the next line. When the line which the cursor is in is bigger than the end row of the last line, the cursor can be moved down to the end of the next line longer.

c) Press ⇨ to move right the cursor to one row. When the cursor is in the end of the line, it can be moved to the home of the next line.

d) Press ⇦ to move left the cursor to one row. When the cursor is in the home of the line, it can be moved to the end of the last line.

e) Press to Page up the screen and the cursor moves to the first row of the first line in the last screen.

f) Press to Page down the screen and the cursor moves to the first row of the first line in the next screen.

3.2.5 MDI Input Display

In MDI mode , select 

key and press 【MDI】softkey to enter MDI display interface. In this interface, single command, single block and blocks can be performed. Program format is the same as edit program. MDI is used for single command or simple block operation (see Fig. 3-2-49).

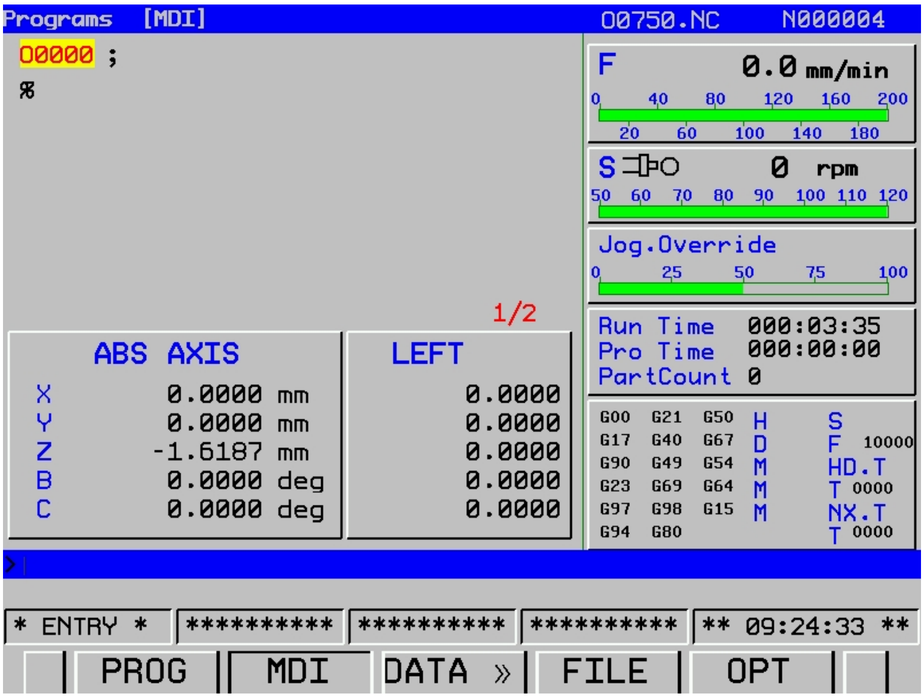


Fig. 3-2-49

Main operation points of MDI are as follows:

1) Press 【Program】key on the edit panel.

2) Enter MDI interface in【MDI】mode. Program number O0000 will be added automatically.

3) Input instruction program to be executed and move the cursor to the line. Then press【Cycle start】to execute it. Insertion, alteration, deletion and search line number can be used for programming in MDI mode. (refer to introduction of edit interface for detailed operation).

4）To stop or end MDI operation in the middle of operation, please follow the following steps:

a. Stop MDI mode

Press feed pause switch on the operation panel. Feed pause indicator lights up and cycle start indicator turns off. Responses of machine are as follows:

（i） When machine is running , the feed operation decelerates to stop.

（ii）Tool stop state is interrupted when machine on this state.

(iii) When M, S or T command is executing, operation stops after M, S, T execution is finished.

When cycle start button on operation panel is pressed again, machine is restarted.

b. End MDI mode.

By pressing 

key on MDI panel, auto operation is stopped. The system enters reset state. When reset command is executed on machine operation, operation decelerates to stop.

Note:

1. Edited program in MDI mode can not be saved**.**

2. Parameters can not be altered in MDI mode during operation.

3. For completely delete the compiled program in MDI mode, please refer to the operations of edit

interface.

4. In MDI mode, single command can be input and executed. After multiple lines of blocks are input, move the cursor to the start block to make the cycle start valid. Otherwise, only the line that the cursor locates and the following blocks will be executed, previous blocks will be omitted.

3.2.6 Data Display

Press 【DATA】softkey to enter data display interface. Command value and modal value are displayed on current executed block (Fig. 3-2-50). Modal state of current executed program is displayed in MDI mode.

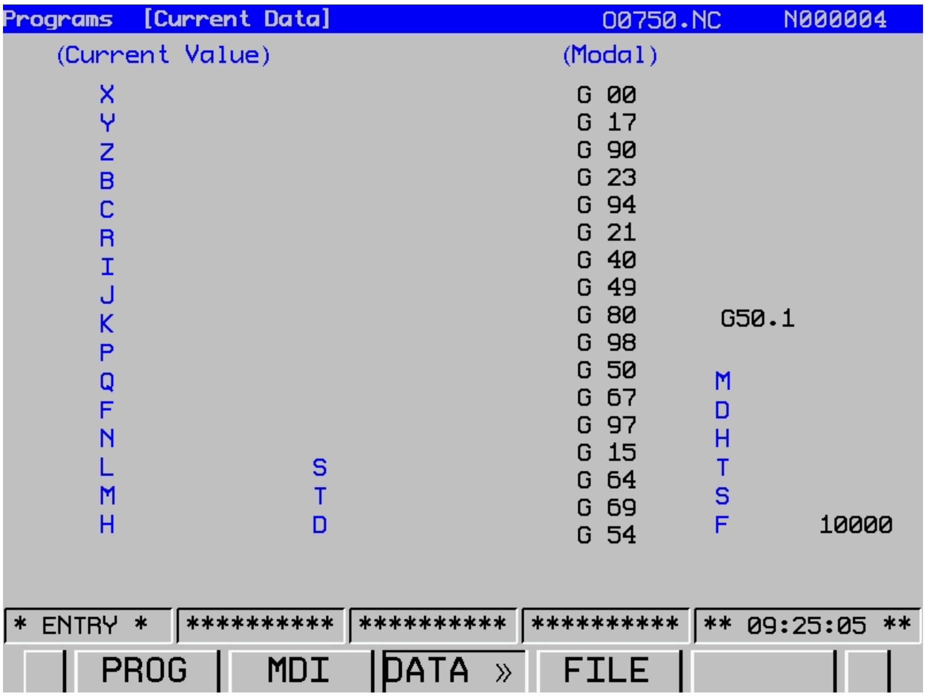


Fig. 3-2-50

Press【DATA】 key again to enter the data interface in the next block and the system displays the command value and modal value of next block following the block which is being executed.

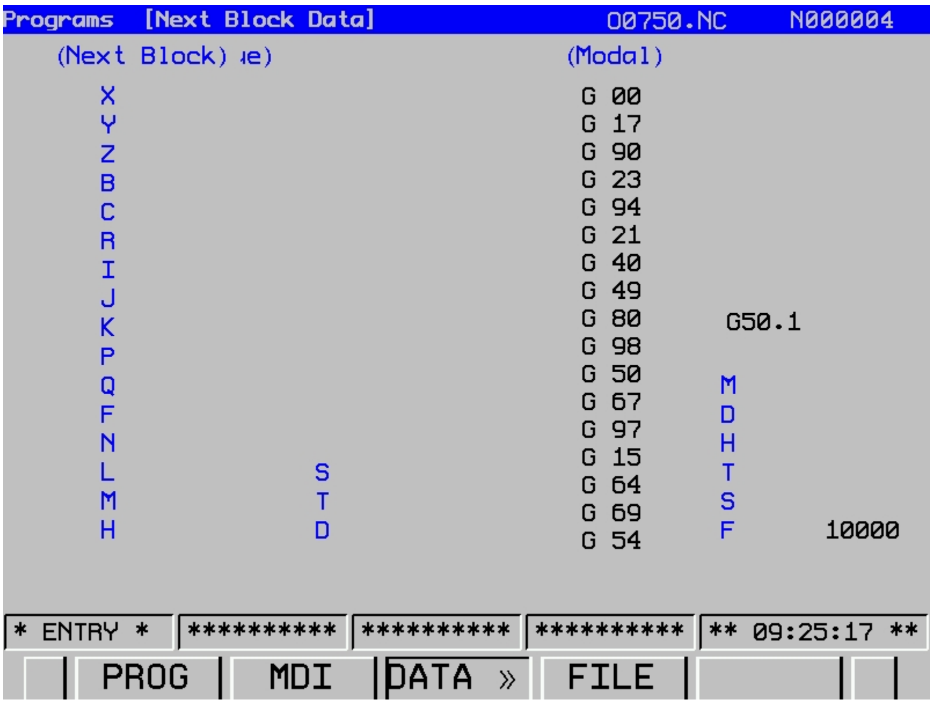


Fig. 3-2-51

3.2.7 Detection Interface

In EDIT mode, press 【DETECT】softkey to enter detection interface. The whole code execution procedures, coordinates of absolute position and remain momentum, spindle speed, feedrate, tool number, and modal can be viewed in detection interface at real time. See fig. 3-2-52. In this interface, each override can be altered by corresponding button on operation panel.

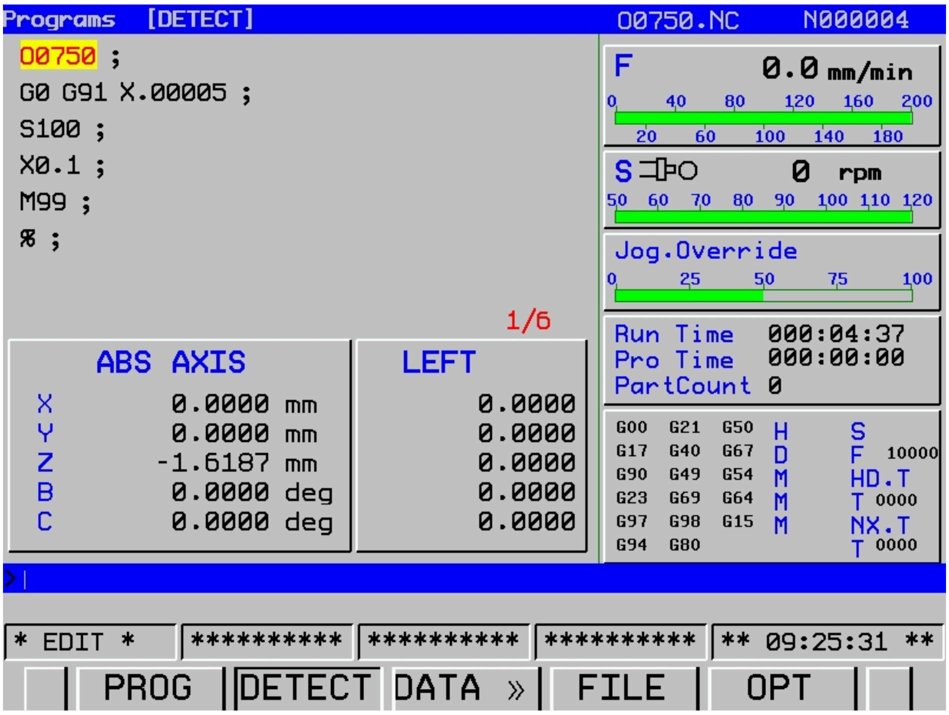


Fig. 3-2-52

3.2.8 File List Display

Press 【FILE】 softkey to enter file list display interface. Following contents can be seen here (see Fig. 3-2-53):

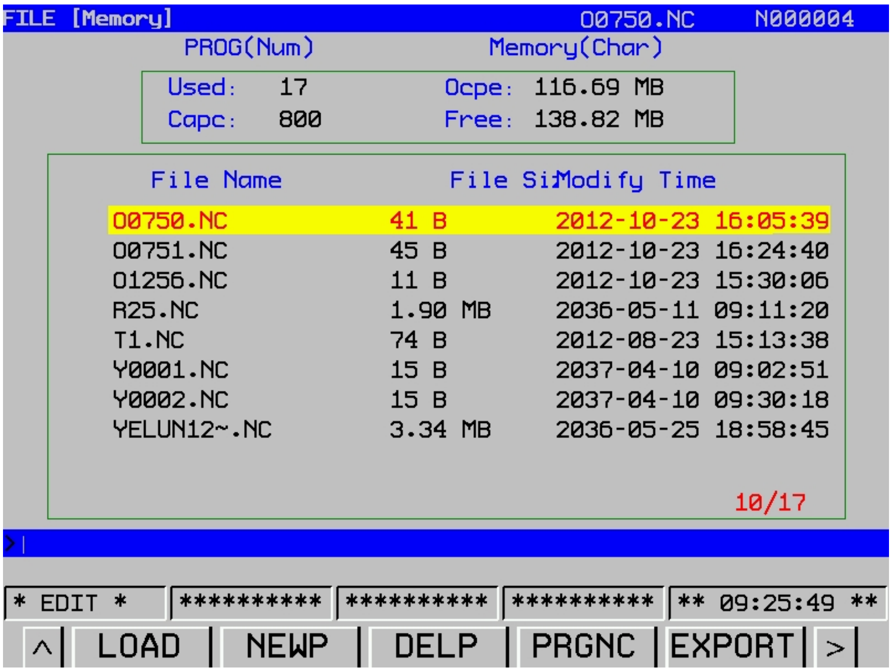


Fig. 3-2-53

Part Ⅱ Operation

(a) Used capacity: number of saved program (including contents)

Capacity: Total number of programs that can be saved in the system. The number of program that can be saved is 800 at present.

(b) Used capacity: Capacity that has been occupied by saved program (displayed by character number).

Available space: Capacity that can be used. Present maximum capacity is 30 M.

Note:

1. Program within 8MB can be loaded to the system in Edit mode. The program exceeds 8MB is displayed as "???" in file list. It is necessary to increase capacity online by DNC or USB.
2. Nine CNC program names can be displayed once on program contents display page. If CNC programs are more than 9, it can not be displayed completely on one page. Press page turning key on this page. LCD displays CNC program name on the next page, and all CNC program name can be displayed again by pressing page turning key repeatedly.
3. Only *.NC file names are displayed on the file list. Chinese character file names are displayed as "？？？？"; While English character file names can be displayed. However, the name can not be displayed if characters are more than 28.
4. It is not allowed to compile file name with Chinese characters. The name will be displayed as "？？？" and duplication error may occur. When file name with 28 or more characters is copied, the file will not display in file list.
5. Correct way to download USB is that pull out USB after downloading external memory. Otherwise, error may occur.
6. It is not allowed to pull out or plug in USB when performing upload, input and output operations. Input/output operation has nothing to do with parameter (10#) of I/O port.
7. Only NC programs in the system other than the ones in USB can be called.

3.2.9 Introduction of Background Edit

Background edit means that programs are loaded to the background and can be edited in any mode. In the foreground edit, however, program can be edited in Edit mode or MDI mode. In the background edit, other part programs can be edited when performing a part program. Moreover, the background edit is manipulated in a similar way to the foreground edit.

1. Background edit is performed in Auto, MDI, USB, DNC or Edit mode.
2. Enter Edit or Auto mode. Only one program that is not currently loaded can be edited in the background. The sum of the size of the programs edited in the background and foreground should not bigger than 8M.

Procedures for background edit

1. It is allowed while program is running in Auto mode.

2. Enter the following screen by pressing 

.

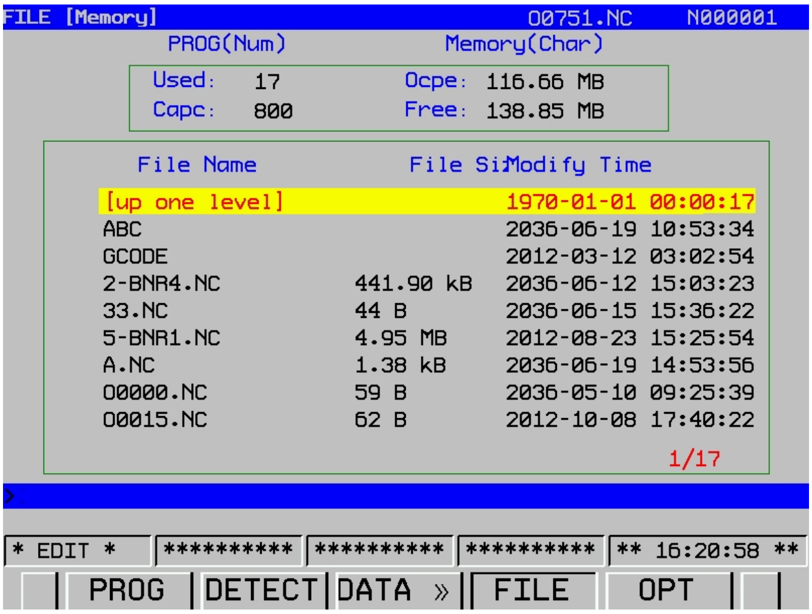


Fig. 3-2-54

3. Press 【FILE】and【OPT】softkeys to enter the following screen.

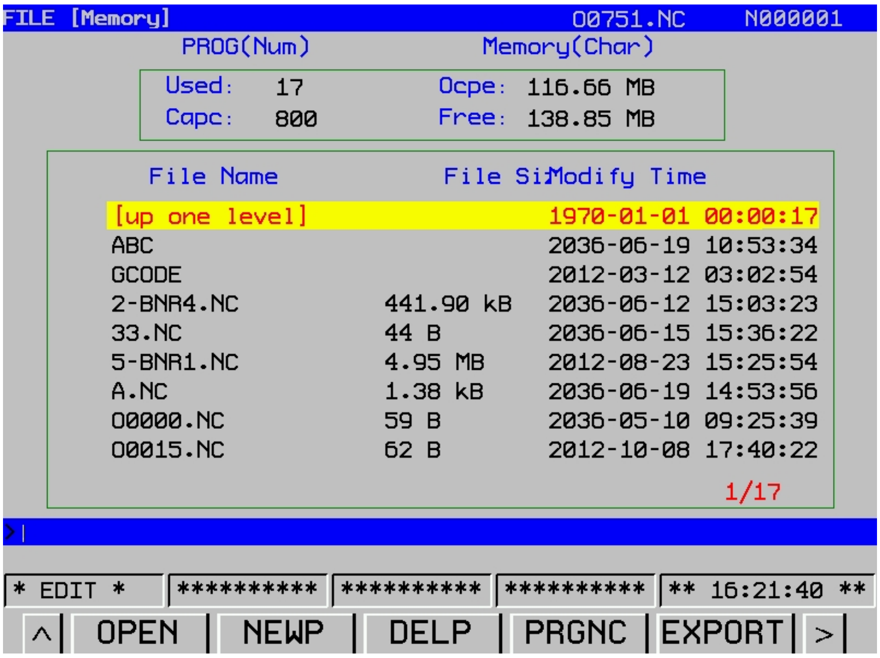


Fig. 3-2-55

4、Press > to enter the following operation interface.

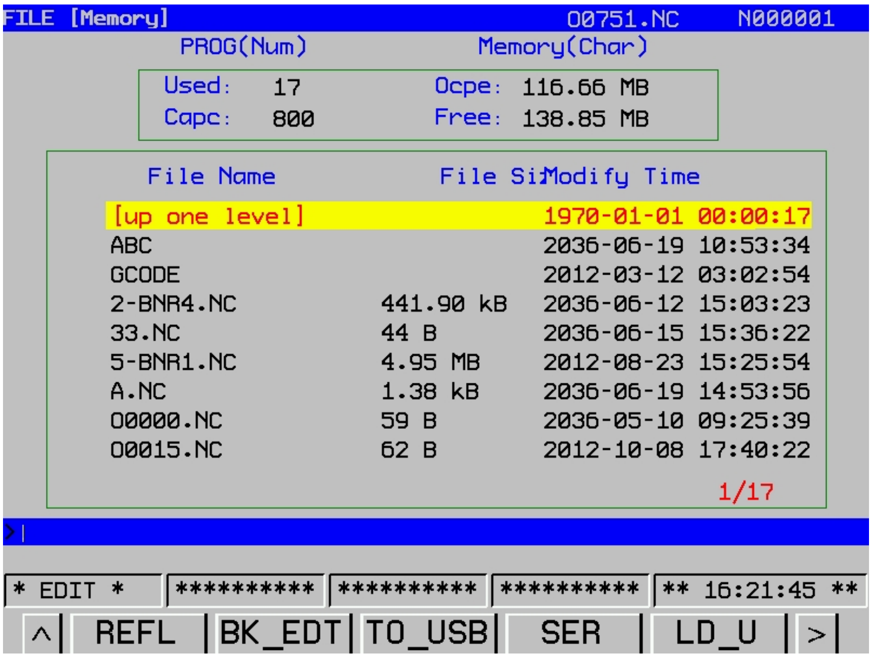


Fig. 3-2-56

5. Press page key , or cursor key , to select the program to be edited. Then press softkey【BK_EDT】to enter the following edit interface.

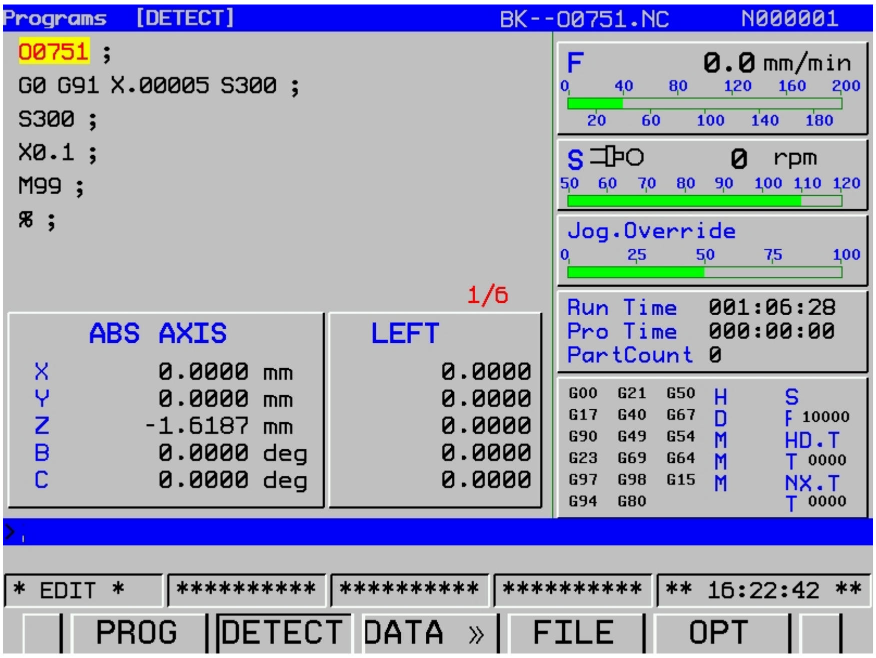


Fig. 3-2-57

6. After completing the program edit, press softkey 【SAVE】->【Enter】 to enter the following interface.

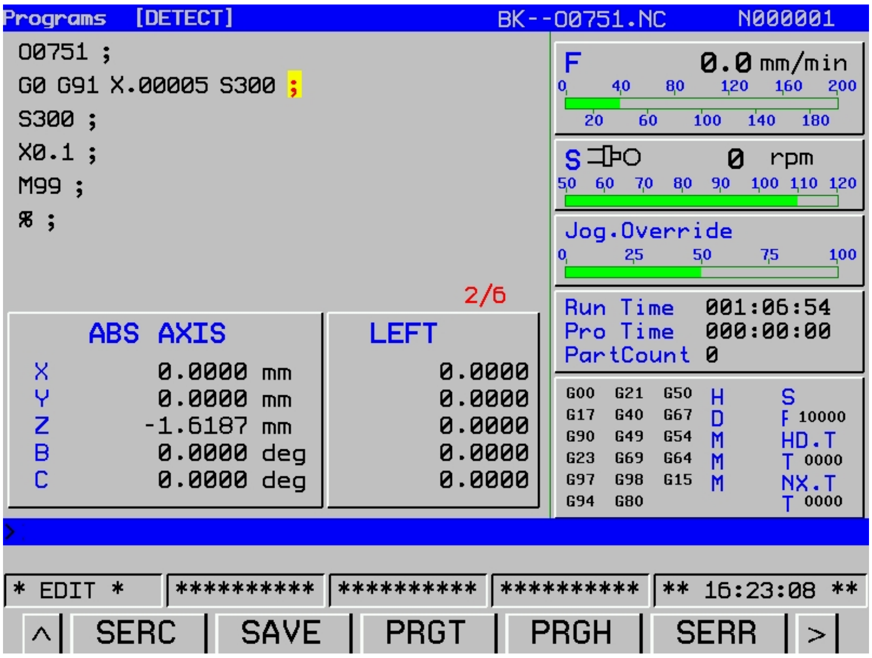


Fig. 3-2-58

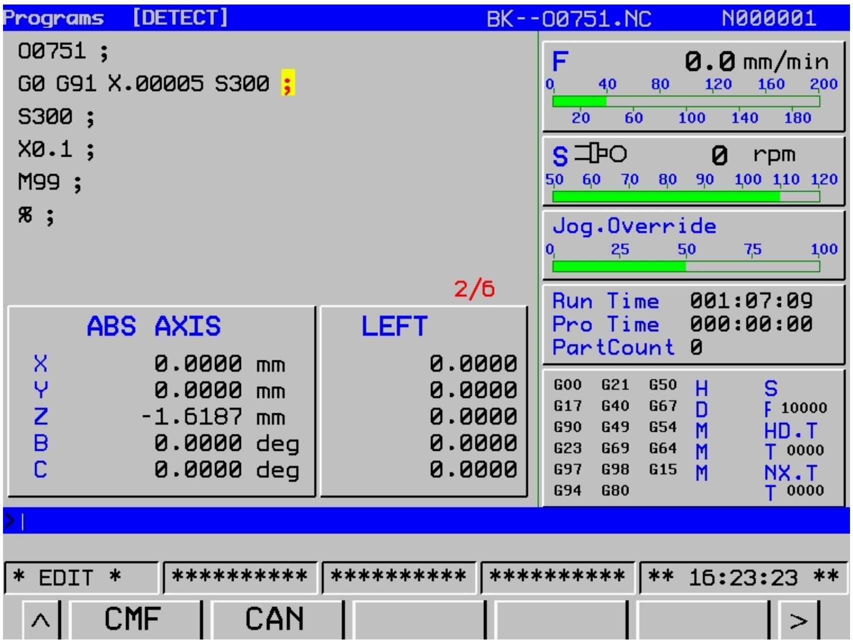


Fig. 3-2-59

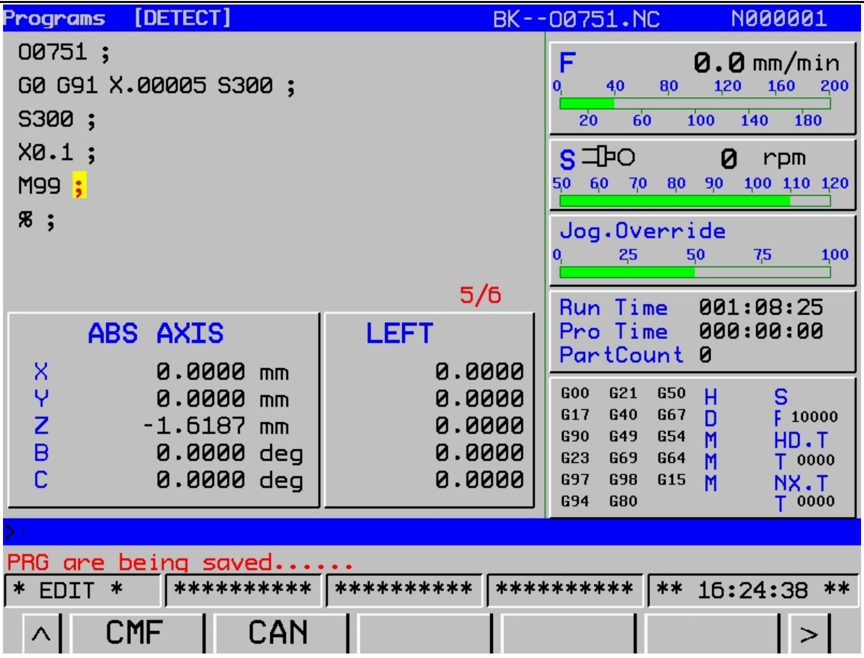
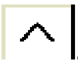


Fig. 3-2-60

7. Press key .

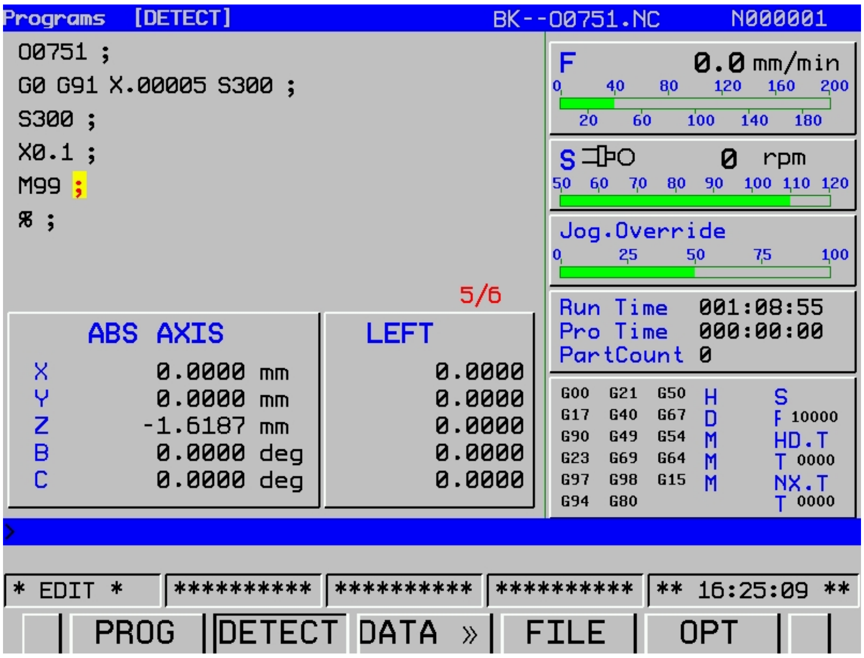


Fig. 3-2-61

8. Press key 【FILE】 -> 【OPT】to enter the following interface.

Fig. 3-2-62

9. Press key【BK_EXT】to finish the edit, and the edited program will be automatically saved.

Note: The background edit is manipulated in a similar way to the foreground edit. Refer to programming instructions for details.

3.2.10 Restarting Programs

It is used when accidents (such as the tool break, the system reset) occur in program running. After releasing the accident, programs are continuously executed by program restart function.

The program can be restarted in two ways: P type and Q type.

P type: program is started with block number specified by the operator.

Q type: program is started automatically from the one being performed at power off.

Procedures:

1. Confirm the current file is the program to be machined or the one being performed at power off.
2. Confirm it is in Auto mode.
3. Press 【Program restart】key on the operation panel.
4. Press 【Operation】key at the lower side of the keyboard.
5. Input the number of the block to be restarted, and press reset key. Then the cursor will skip to the necessary block by pressing 【P type】key. Inputting block number is unnecessary if 【Q type】key is pressed. System will search the number of the block being performed at power off. If currently machined program is inconsistent with the one being performed at power off, press 【Q type】key to execute the programs at beginning.
6. Check whether there is the possibility that the tool might hit a workpiece or other objects when it moves to the machining restart position. If such a possibility exists, move the tool manually to the

machining restart position without encountering any obstacles.

7. Press 【Cycle start】key to execute the program.

Notes:

1. Parameter #1950.7 decides whether to execute M, S, T, B functions based on actual machining requirements.

2. Parameter #1960 decides axes returning sequence based on actual machining requirements.

3. Please operate the system carefully according to the manual. Attempting to perform unconventional operation may result in the machine behaving unexpectedly.

3.3 Display Setting

3.3.1 Page Setting

1. Enter the page

Enter the offset and information display setting interface by pressing OFFSET SETTING. There are six interfaces such as **[OFFSET]**, **[Set]**, **[Work]**, **[Macro]**, **[Pitch]** and **[LOG]**, which can be checked and modified by the corresponding softkey or each interface can be shifted by pressing OFFSET SETTING. Refer to the following Fig. 3-3-1 for details:

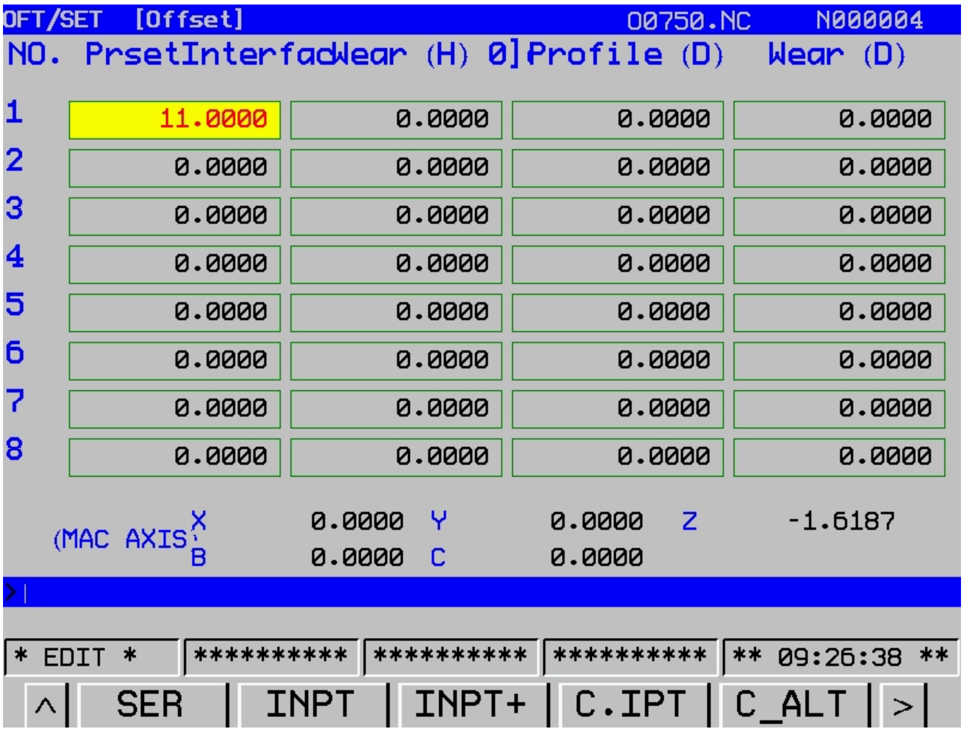


Fig. 3-3-1

Enter the next page by pressing【>】.

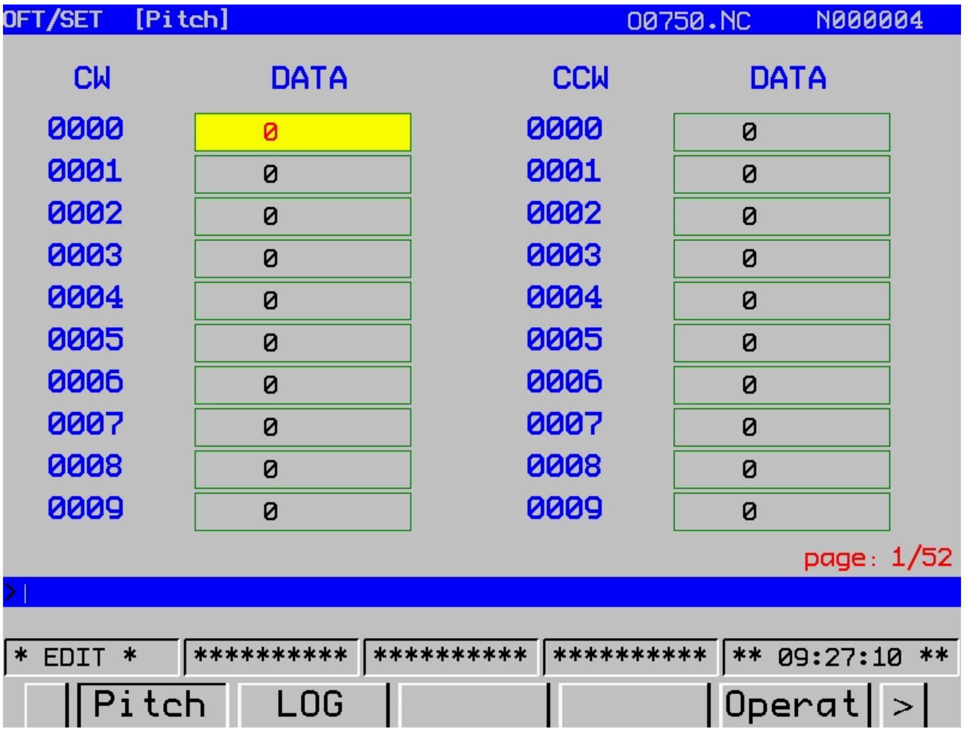


Fig. 3-3-2

Note: The pitch error compensation setting can only be shifted between -7~+7. If it exceeds its range, the system flashes with an alarm: Invalid data.

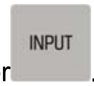
Procedures for setting and displaying the tool offset value

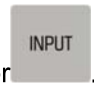
1) Press the function key. 

. Refer to the Fig. 3-3-1.

2) The tool compensation screen is displayed by pressing the softkey **[OFFSET]**. The screen varies according to the type of tool offset memory.

3) Move the cursor to the compensation value to be set or changed using page keys and cursor keys, or enter the compensation number in this case, the compensation number can be searched by controlling the softkey **[SER].**

4) Set the compensation value. See figure 3.3-3. A value is input before pressing the softkey **[INPT]** or 

. The tool compensation automation acceleration automatic adding function can be achieved by pressing **[INPT+]**. For example, D1 must be changed into 2 from 5. In this case, there are two methods can be performed: a. to write the number 2 directly before controlling the softkey **[INPT]** or 

. b. To write－3 firstly, and then the softkey **[INPT＋]** is pressed. The softkey **[C.IPT]** can be directly read a machine coordinate system of Z axis at its outline (H) (tool length compensation number). The machine coordinate, the relative coordinate, the absolute coordinate can be directly switched through pressing **[C_ALT]**, so that the user can check them in time.

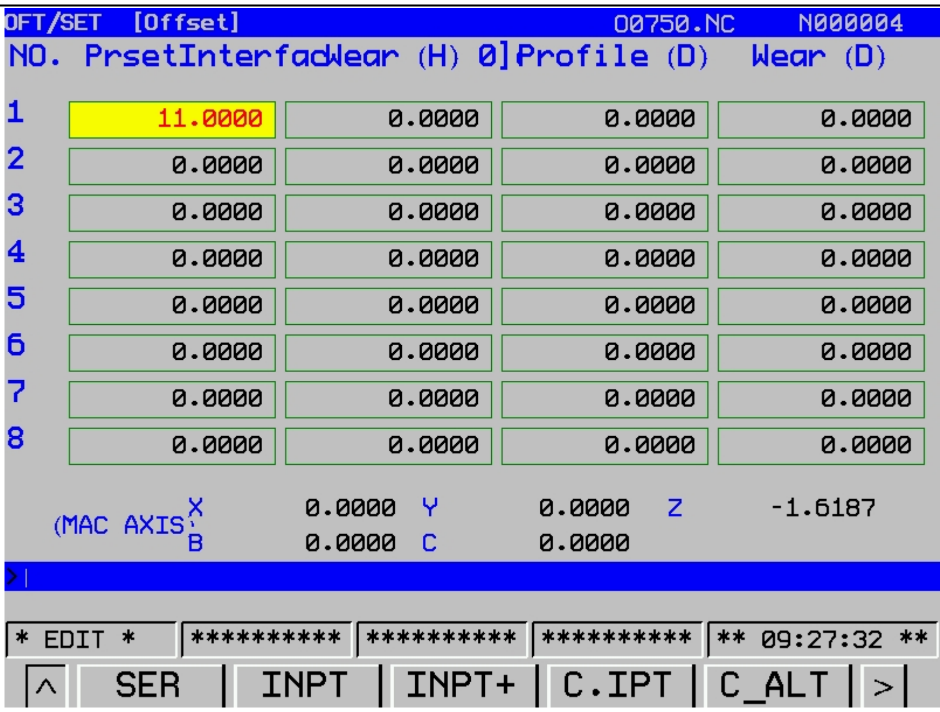


Fig. 3-3-3

2. The description of **[Set]** interface operation

Procedures for setting the data

1) Select the **<Edit/MDI>** mode.

2) Press the function key 

.

3) Press the softkey **[Set]** to display the setting data screen.

4) Move the cursor to the item to be changed by pressing the cursor keys.

5) Enter a new value and press either 

or the softkey **[INPT]**.

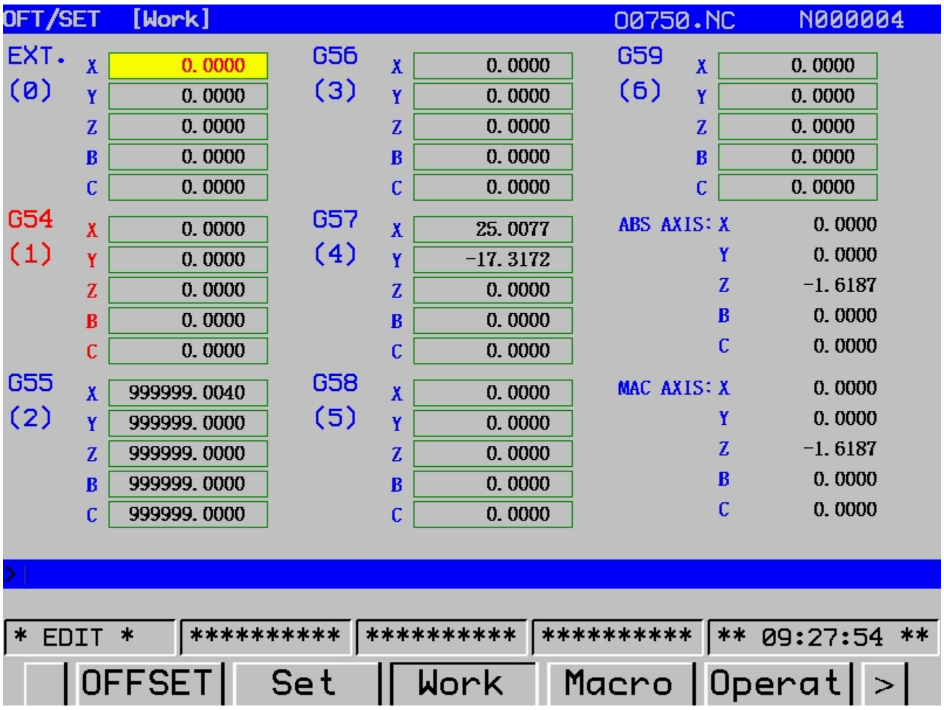


Fig. 3-3-4

Part Ⅱ Operation

Either 1 or 0 is input based on the following descriptions:

(1) IO Port

This parameter corresponds to parameter #10. The value indicates NC data source in DNC mode. Refer to parameter manual.

(2) The sequence number of automatic accumulation

The parameter corresponds to #1.5 parameter. Refer to parameter manual for details.

0: In the Edit mode, when the program is registered by the keyboard, the system would not being inserted the sequence number automatically.

1: In the **<Edit>** mode, when the program is registered by the keyboard, the system may insert the sequence number automatically.

(3) Increment

The parameter corresponds to parameter #1621. Refer to parameter manual for details.

(4) Number clear

This parameter is 0. Input 1 to clear the workpiece number. The signal is valid on its rising edge and then it is set to 0.

(5) System current date

The current data of the system can be set by inputting relevant data. It is available after power off. Note: the date should be set within the year 2037.

(6) Current time of the system

The current time of the system can be set by inputting relevant data. It is available after power off.

All of these operations are modified, and then press the softkey [INPT] to execute it.

3. The operating description of **[Work]** interface

Enter the workpiece coordinate system interface by pressing the softkey **[Work]**; refer to the figure 3-3-5:

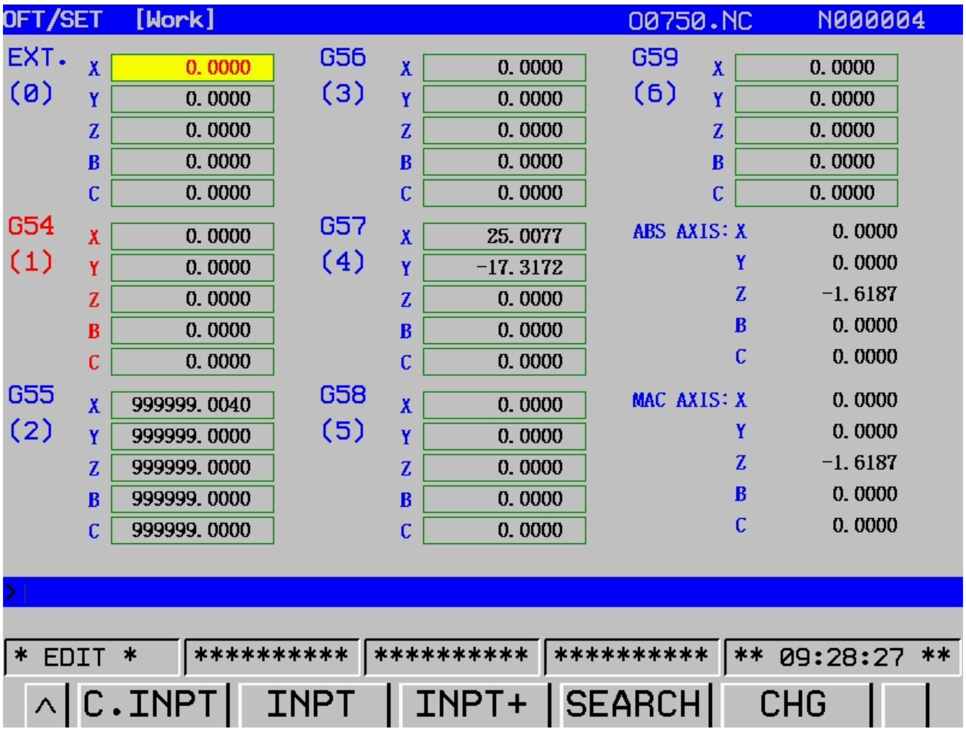


Fig. 3-3-5

The following operations are shown below:

(a) Enter the **<MDI>/<Edit>** mode;

(b) Move the cursor by the direction key (either up or down) on the item to be changed;

(c) Enter the following screen by controlling the **[Operat]** key. (Refer to the Fig. 3-3-6):

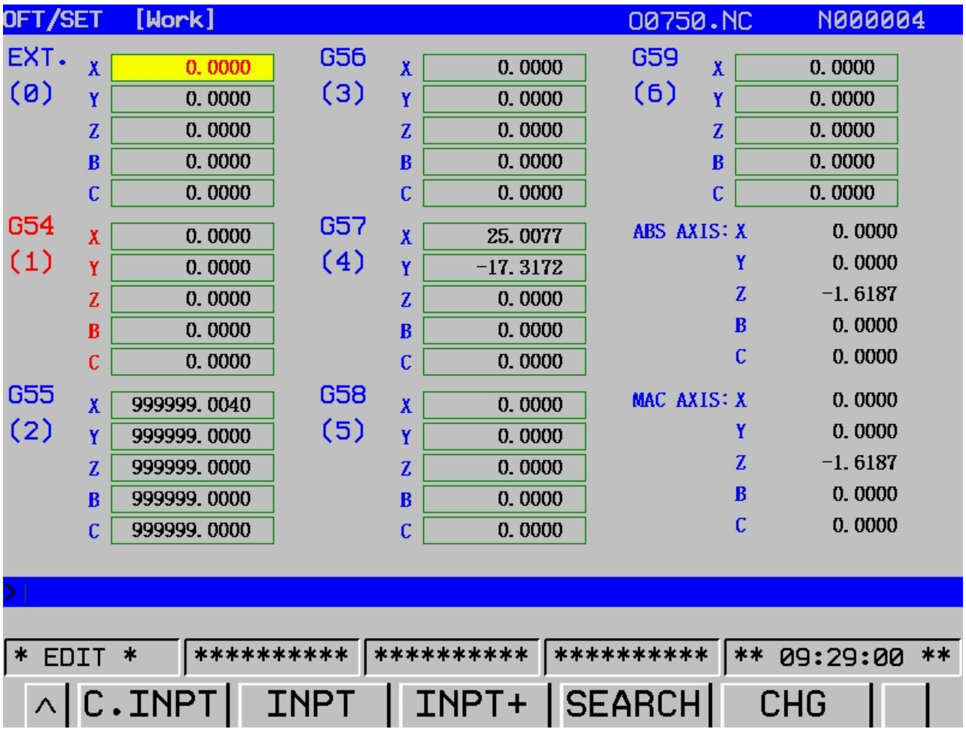


Fig. 3-3-6

The EXT. is a basic offset; the user can set it by **[INPT]** or **[INPT+]** softkey like the operation of G54~G59. It is very convenient for the user, for which the [**C. INPT**] is read into the current machine coordinate directly. The corresponding machine coordinates can be read by pressing the **[C. INPT]** when the cursor is moved on the corresponding axis. Simultaneously, the absolute coordinates displayed on the interface may vary from the read machine coordinate value based on each coordinate of G54~ G59 and it is very convenient for user to operate.

The operation of search can be performed based on the value in brackets of each coordinate. See the above figure, G57（4）, the cursor may move to the contents to be searched by inputting 4 and pressing**【SEARCH】**. When searching addition workpiece coordinates, add a letter P ahead (the scope of workpiece coordinate is 0-6, while additional workpiece coordinate scope is P1-P48)

4. The operating description of **[Macro]** interface

Enter the macro variable interface by pressing the softkey **[Macro]** (Refer to the Fig. 3-3-7):

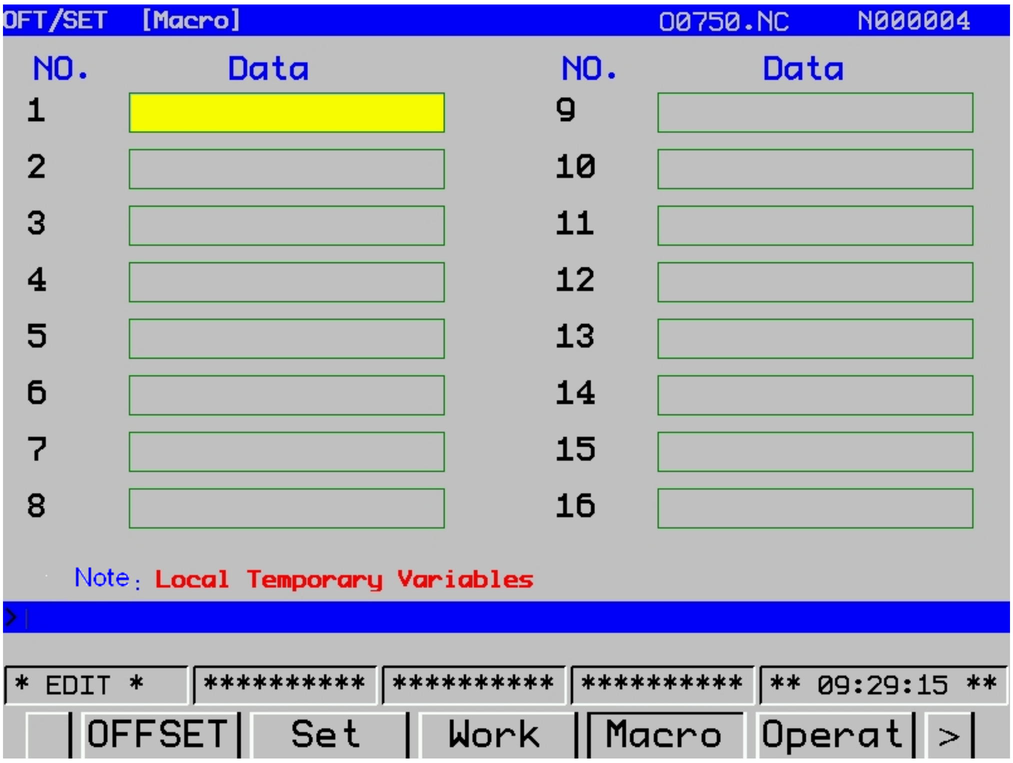


Fig. 3-3-7

Enter the macro variable setting interface by pressing the softkey **[Operat]**.

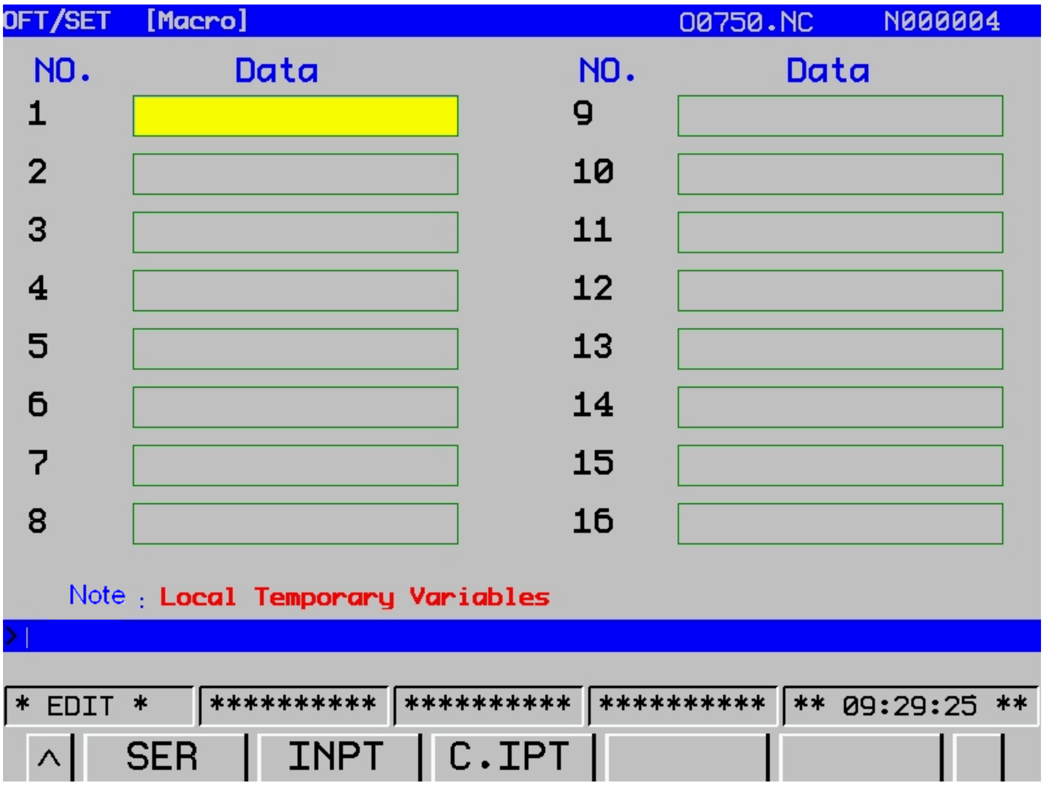


Fig. 3-3-8

The operations are shown below:

(a) Enter the **<MDI>/<Edit>** mode;

(b) Move the cursor by the page key or the direction key (either up or down) on the sequence number to be changed; or enter the variable sequence number to be modified, and then press the softkey of [Search] directly.

(c) The methods of the data modification and the machine coordinate reading on this interface are similar as the mentioned previously.

5. The operating description of **[Pitch]** interface

Enter the pitch error compensation interface by pressing the softkey **[Pitch]**, which is shown as the above figure. Both the operations of interface search and the modification can be performed on this screen by controlling the softkey of **[Operat]** (Refer to the Fig. 3-3-9):

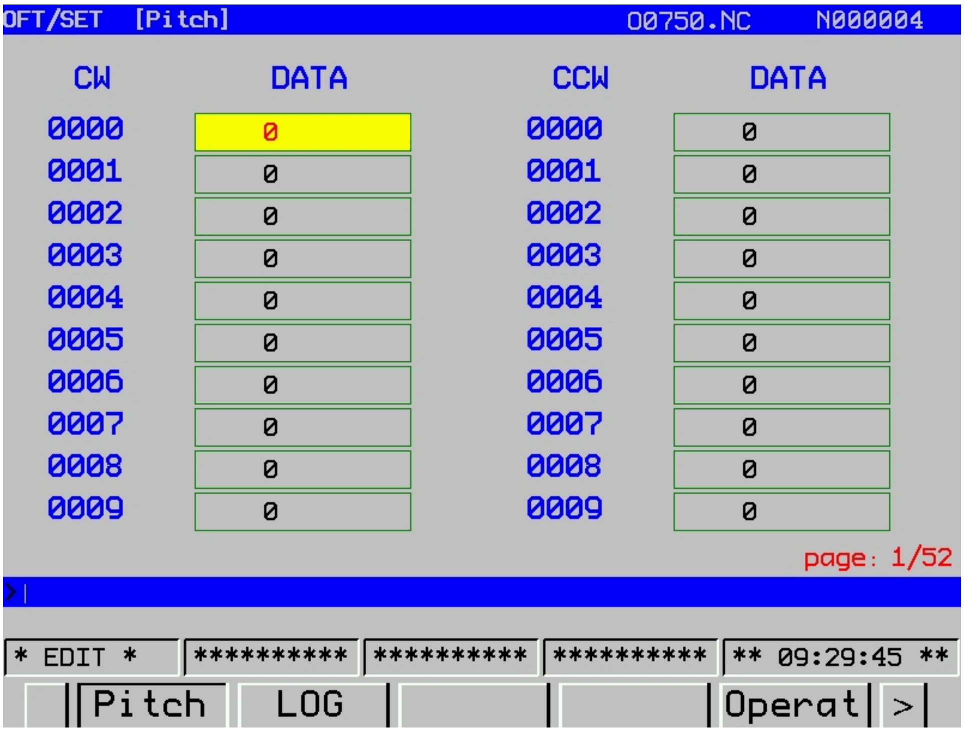


Fig. 3-3-9

If pitch error compensation data is specified, pitch errors of each axis can be compensated in detection unit per axis. Unit of compensation value is detection unit.

Pitch error compensation data is set for each compensation point at the intervals specified for each axis. The origin of compensation is the reference position to which the tool is returned.

The pitch error compensation data is set according to the characteristics of the machine connected to the CNC. The content of this data varies according to the machine model. If it is changed, the machine accuracy is reduced. In principle, the end user must not alter this data. Pitch error compensation data can be set with external devices such as the Handy File. Compensation data can also be written directly with the MDI panel.

The following parameters must be set for pitch error compensation. Set the pitch error compensation value for each pitch error compensation point number set by these parameters.

Parameter No.	Explanation
2800.0	Screw pitch compensation：0：Not perform 1：Perform
2800.1	Bidirectional screw pitch error compensation：1：Valid / 0：Invalid
2806	Screw pitch error compensation for reference return
2810	Screw pitch error compensation point No. of relevant machine reference position
2811	The number of the farthest screw pitch error compensation point at

	negative side during CCW rotation
2812	The number of the farthest screw pitch error compensation point at positive side during CCW rotation
2813	Ratio of compensation value
2814	Interval of compensation point
1068	Revolving volume of revolving axis per revolution

6. The operation of **[LOG]** interface

To prevent the machining program or CNC parameter from being maliciously modified, **GSK25i** system offers an authority function. The password can be classified into 9 levels, such as: The zero level (the system high-level), the 1st level (the system service level), the 2nd level (the machine manufacturer level), the 3rd level (the installation and debugging level), the 4th level (the terminal management level), the 5th level (the common user level 1), the 6th level (the common user level 2), the 7th level (the common user level 3) based on the rank is from high to low, the system is the lowest level by default when the machine is power on. (See the figure 3.3-10).

The zero level: the highest level that is reserved by developer.

The 1st level: manufacturer service level, variable system data can be modified.

The 2nd level: The PLC program, PLC notes as well as screw pitch compensation are allowable to be modified. Inputting and outputting of PLC and screw pitch file are allowed. The authority of user inputting/outputting interface can be modified.

The 3rd level: The parameter, the PLC resource data are allowable to be modified. PLC ON/OFF, alarm/operation information clearing, inputting/outputting files are permitted. System, interpolation and position control maintenance software can be updated.

The 4th level: the program, the tool compensation, the setting, the workpiece coordinate offset and the macro value can be modified. Inputting/outputting files are permitted. Authority to modify password of programmer is available.

The 5th, 6th, 7th: the operation authority authorized by end user manager through bit parameter.

The lowest defaulted level of the system: the operation authority authorized by end user manager through bit parameter. It's unnecessary to input the password.

Note: To modify the login password, first input the original password to enter relative user, then input the password to be modified at the appropriate position (input twice). The password of the lower level can be modified at the higher level, while the password can not be changed at the same level.

Definition of the bit parameter authorized by the end user manager

Bit	Definition	Note
0	Modifying input/output G code program authority	1 authority
1	Modifying geometric offset/input/output offset authority	1 authority
2	Modifying wear offset/input output offset authority	1 authority
3	Modifying setting authority	1 authority
4	Modifying input/output workpiece coordinate offset authority	1 authority
5	Modifying input/output macro value authority	1 authority
6	Reserved	
7	Reserved	

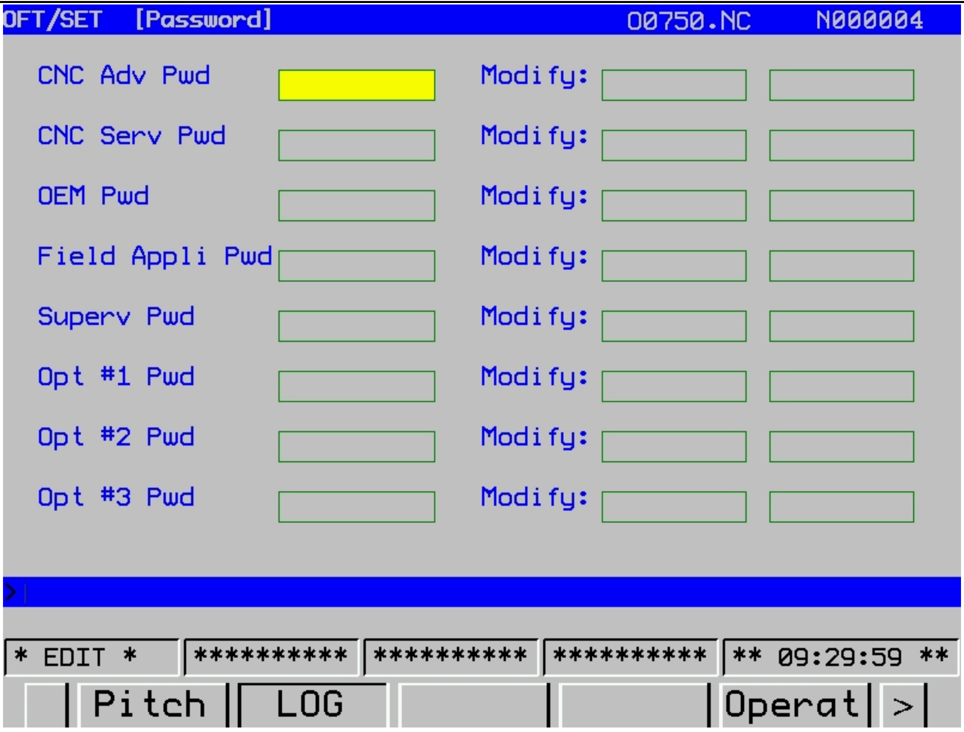


Fig. 3-3-10

Press 【Operat】key and input password. See the following figure:

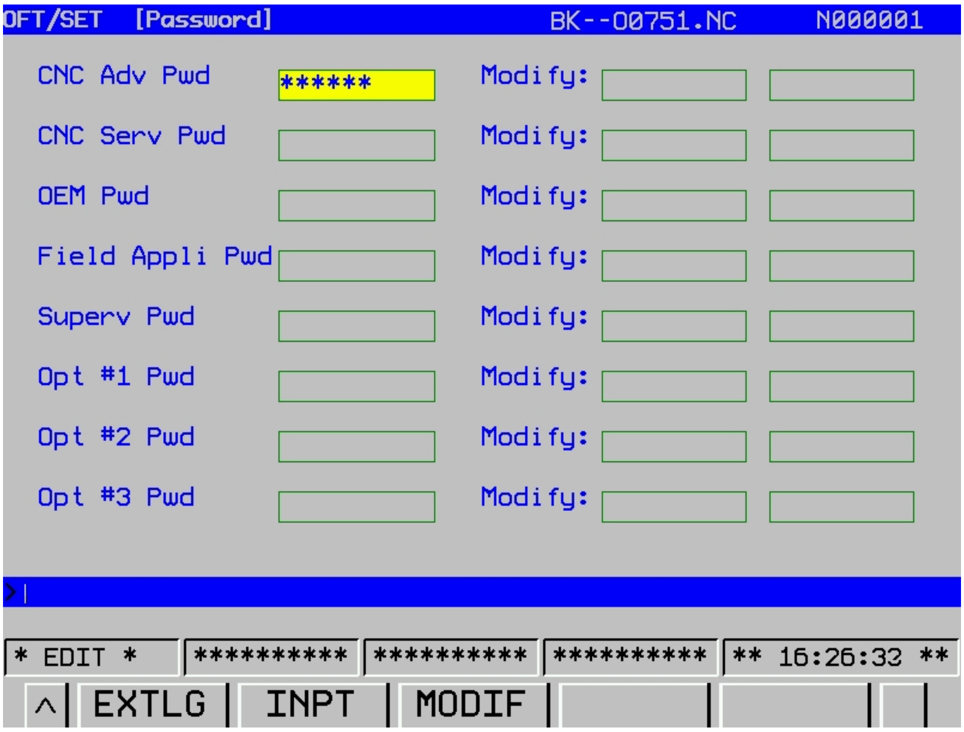
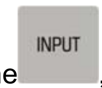


Fig. 3-3-11

The modification processes are shown below:

1) After entering the password setting interface, move the cursor to the item to be changed.

2) The corresponding level password is input by pressing the softkey **[Input]** or the 

, if it is correct, the system may show prompt "Correct password"; otherwise, "Incorrect password". The password is immediately cancelled and exit by pressing the **[Log-off]**.

3) Modify the corresponding parameter and setting.

4) The password is cancelled automatically after the modification is executed.

3.4 Figure Display

A tool path on a program can be drawn out on the screen, the machining process displayed on the figure can be checked by viewing the path on the screen, the displayed figure can be scaled up or down, and the figure parameter must be set before drawing a figure.

Enter the figure interface by pressing the 

, there are two display methods: **[GRAPH]** and **[GRA]**, which can be switched by the corresponding softkey. Refer to the figure 3.4-1 for details:

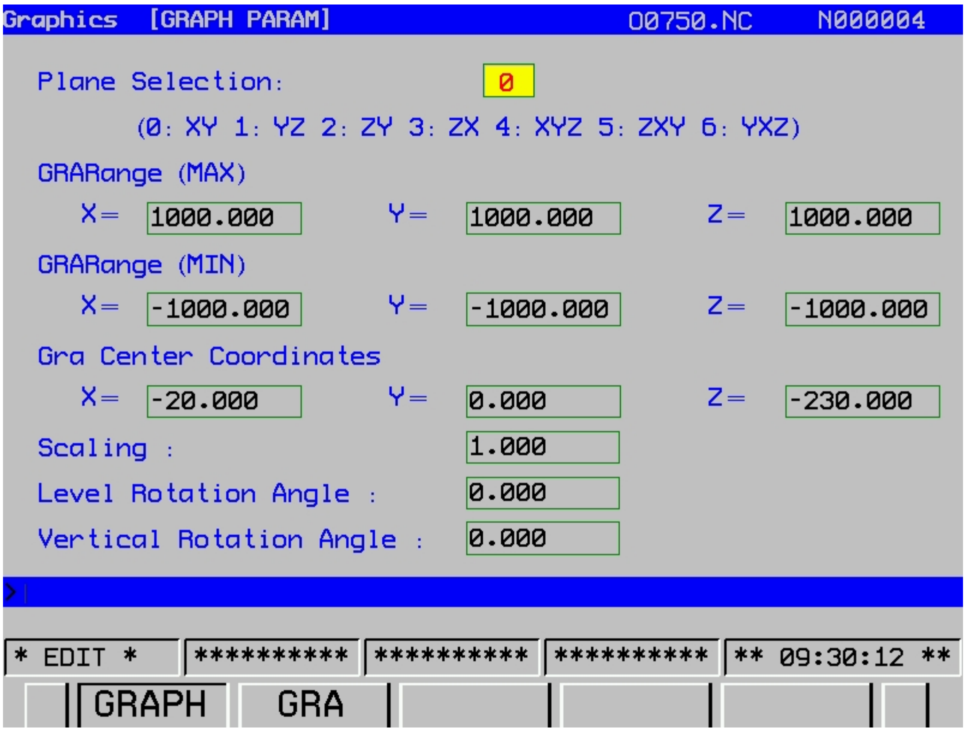


Fig. 3-4-1

1) The figure parameter interface

Enter a figure parameter interface by pressing the softkey of **[GRAPH]**; refer to the Fig. 3.4-1.

A. A signification of the figure parameter

Coordinate selection: Set a graphic plane, there are 6 methods for selecting, such as the 2nd line.

Figure mode: Set the figure display mode.

Scaling: Set the graphic proportion.

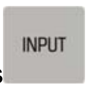
Figure center: The workpiece coordinate value corresponding to LCD center is set in the work piece coordinate system.

The maximum or minimum value: the path range of graphic effective description is set.

B. The setting method of the graphic parameter

a. Move the cursor to the parameter to be set;

b. Input the corresponding value in terms of the actual requirement;

c. Press 

 to confirm.

d. The machine moves with automatic operation start. Tool path is shown below:

Details:

● Range (the actual graphics range), the graphics screen dimension is shown below:

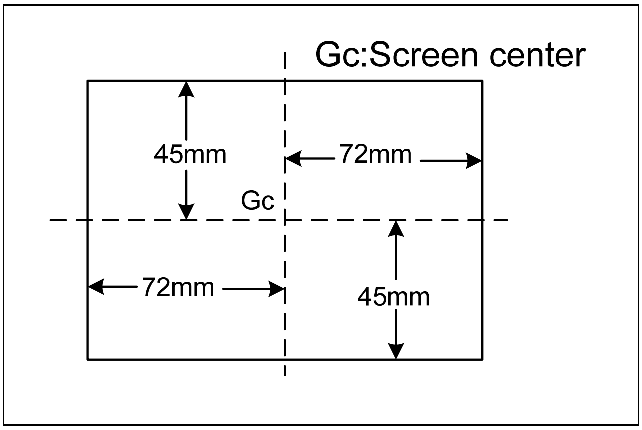


Fig. 3-4-2

The maximum graphic range value is indicated as 144mm (width) ×90mm (height)

●To draw a section of the program within the actual graphics range, set the graphics range using one of the following two methods:

Range:

1) Set the center coordinates of the range and the magnification.

2) Set the maximum and minimum coordinates for the range in the program.

Whether 1 or 2 is used depends on which parameters are set last. A graphics range which has been set is retained when the power is turned off.

1. Setting the center coordinate of the graphics range and graphics magnification

Set the center of the graphic range to the center of the screen. If the drawing range in the program can be contained in the above actual graphics range, set the magnification to 1 (actual value set is 100).

When the drawing range is larger than the maximum graphics range or much smaller than the maximum graphics range, the amplification rate should be modified, which is usually determined as follows:

Graphic magnification = Graphics magnification (**H**), or graphics magnifications (**V**), whichever is smaller Q.

Graphics magnification **H** =α/ (length along with program to horizontal).

Graphics magnification V= β/（length on program to vertical direction axis）.

α:144mm β:90mm

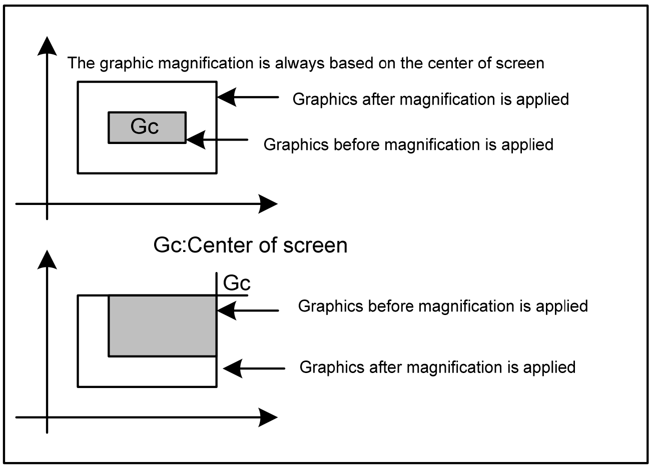


Fig. 3-4-3 Applying graphics magnification (Example of amplification)

The supplement of graphic scale up or down:

1) The rotation of the graph: It can be rotated by the four keys [F], [D], [H] and [B] on the operator panel. The graph with large-capacity may cause to a little slowly response.
2) The scaling of graph: The graph scaling can be controlled by the G code for which it can be controlled by the up or down key on the operator panel, too.
3) The drawing origin and the graphic center coordinate can be changed by the direction keys ⇧ ⇩ ⇨ and ⇦.

Workpiece coordinate system and graph

The drawing origin and graphic center point will not be changed even if the workpiece coordinate origin is changed. In another word, the workpiece coordinate system origin is always consistent with the graphic origin.

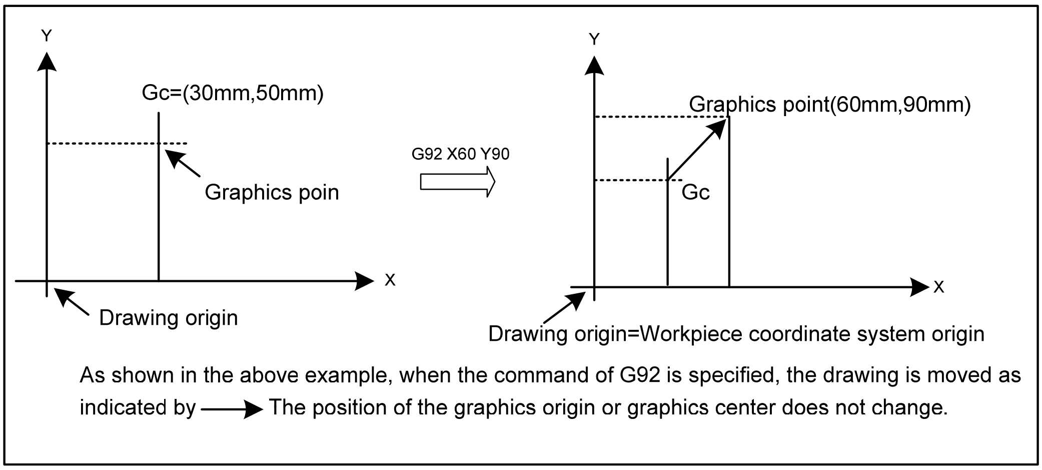


Fig. 3-4-4 Workpiece coordinates origin and graphics origin

The valid range of graphic parameter axis is: 0 ~±9999999.

Notes:

1. The unit is either 0.001mm or 0.0001inch. Note that the maximum value must be greater than the minimum value for each axis.

2. When setting the graphics range with the graphics parameters for the maximum and minimum values, do not

Part Ⅱ Operation

set the parameters for the magnification and screen center coordinates afterwards. Only the parameters set last are effective.

SCALE

The amplification rate of graph is set, namely, the graph parameter scaling is modified.

Graphic center

X=_

Y=_

Z=_

B=_

C=_

Set the coordinate value on the workpiece coordinate system at graphic center.

Notes:

1. When MAX. and MIN. of RANGE are set, the values will be set automatically once drawing is executed

2. When setting the graphics range with the graphics parameters for the magnification and screen center coordinates, do not set the parameters for the maximum and minimum values afterward. Only the parameters set last are effective.

EXECUTING DRAWING ONLY

Since the graphic drawing is done when coordinate value is renewed during automatic operation, etc., it is necessary to start the program by automatic operation. To execute drawing without moving the machine, therefore, enter the machine lock state.

1) Deleting the drawn graph

The previous drawn graphs can be randomly deleted by pressing the softkey **[CLEAR]**.

2) Graph interface

Enter a graph interface by pressing the softkey **[GRAPH]**. (Refer to the Fig. 3-4-2):

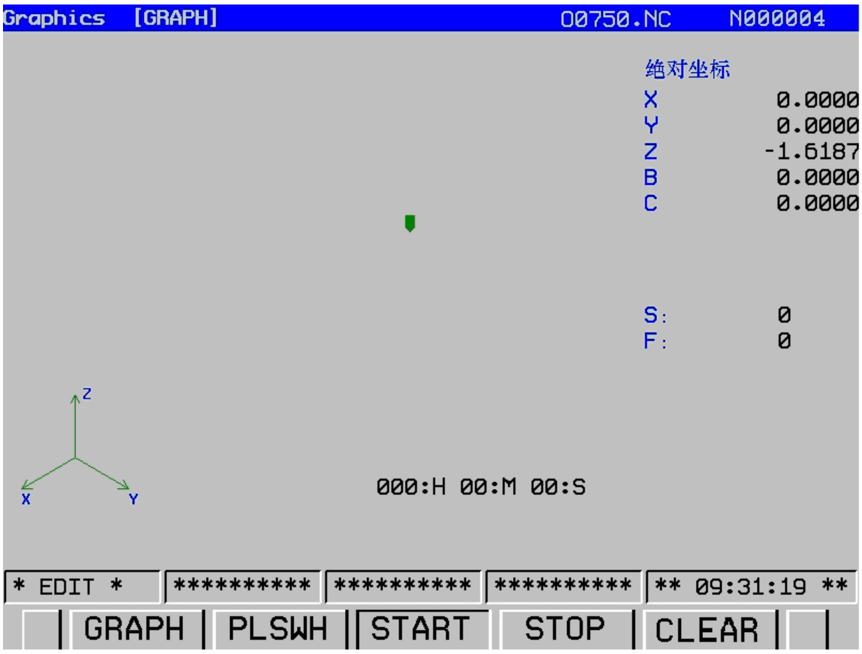


Fig. 3-4-2

On the page of the graph, the program machining path operation can be monitored.

A. The drawing is performed by pressing the softkey of **[START]**, the selected state is performed after the softkey of **[START]** is displayed, you can view that the tool head moves to draw;

B. The **[STOP]** softkey displays a selected state by pressing the softkey **[STOP]**; in this case, the drawing is stopped;

C. The graph is shifted to display on the coordinate systems corresponding to **0**～**6** when the **[PLSWH]** softkey is pressed each time.

D. The drawn graph is eliminated by pressing the softkey**[CLEAR]**.

E. This system has the functions of both the graph rotation and scaling: (Refer to the above-mentioned description)

3.5 Alarm Display

When the system alarms, the "alarm" information is displayed with flashing at the last line of LCD. In this case, the alarm page is appeared by pressing the key 

, the operation softkeys such as **[ALM]**, **[ALMR]**, **[OPTR]** **[PROR]** and **[CLEAR]** are shown on this interface; shift or view can be performed by these corresponding softkeys (**Refer to the following figures**).

1. Alarm interface

Check the current alarm information on the <ALM> interface, which is shown as figure 3-5-1:

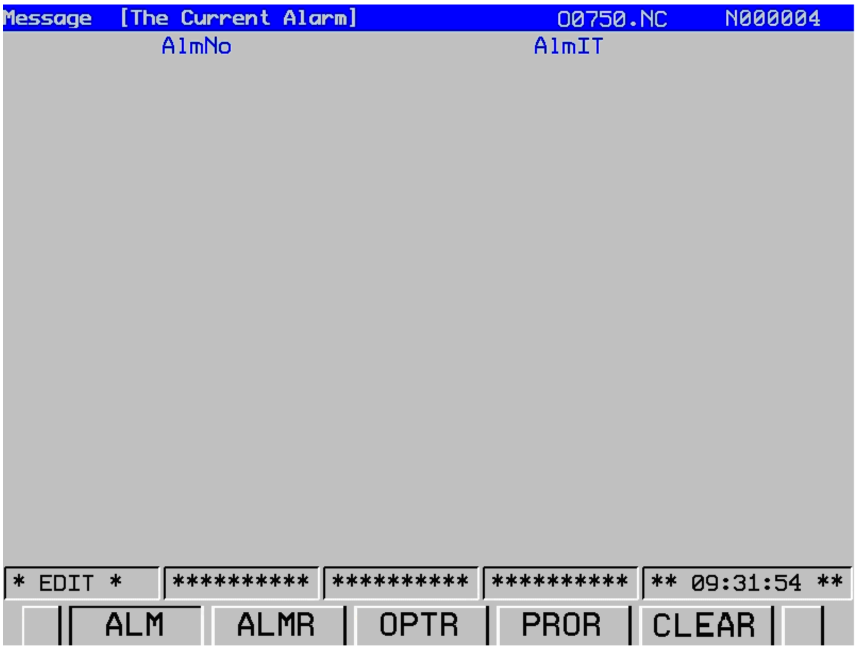


Fig. 3-5-1

The details of current P/S alarm number are displayed on the alarm display screen. Refer to the appendix for the alarm content.

2. **Alarm record**

Enter the alarm record interface on **<ALM>** interface by pressing the softkey of **[ALMR]**. Refer to Fig. 3-5-2:

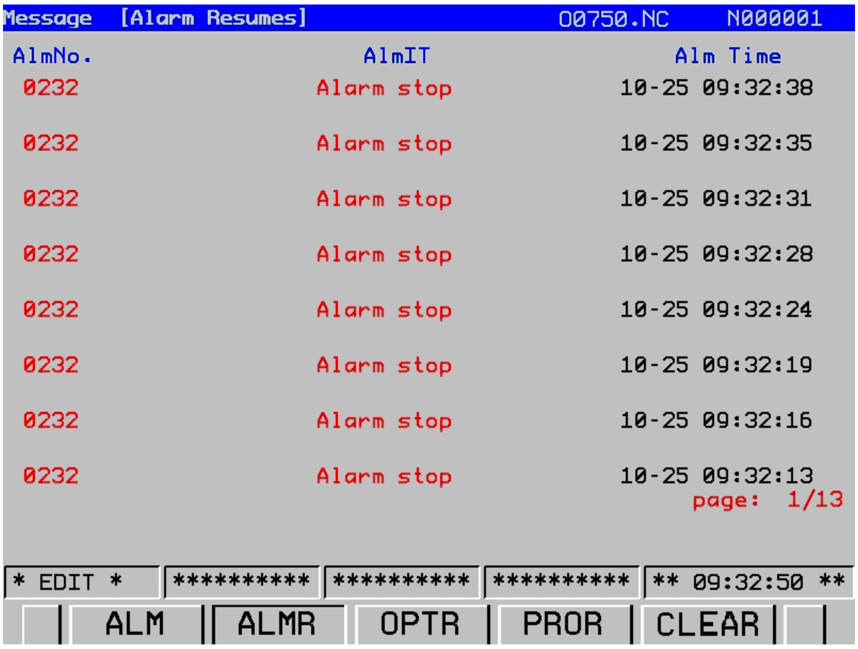
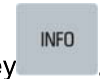


Fig. 3-5-2

On the interface, the alarms are listed in the time sequence so that the user can easily check it.

Procedure for Alarm Record Display:

1) Press the function key 

.

2) Press the chapter selection softkey **[ALMR]**.

The following information items are displayed:

a. The alarm date issues

b. Alarm No.

c. Alarm message

3) Switch the interface by the page keys.

4) Press the [Clear] key to delete the recorded information.

3. Operation record

On the alarm interface, enter the operation record interface by pressing the softkey **[OPTR]**. Refer to Fig. 3-5-3:

The content displayed on the operation record interface is the detailed modification information about the system parameter and ladder diagram, such as the content and time.

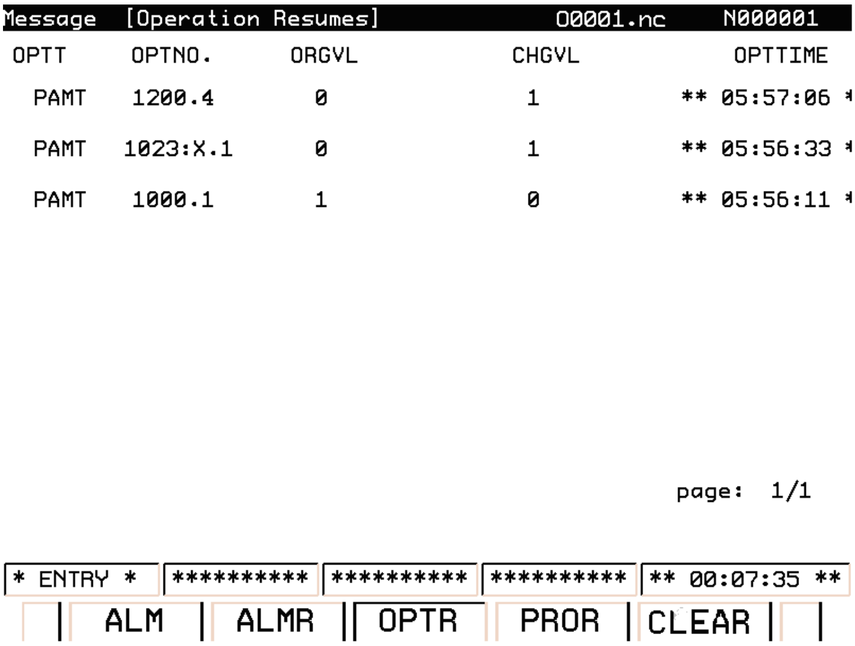
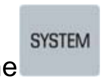


Fig. 3-5-3

The operation record is checked by the page keys and can be deleted by pressing **<CLEAR>** (on debugging level or above)

3.6 System Interface Display

3.6.1 System Interface Display

Enter display screen by pressing the , four display methods are available, **[ALLPAR]**, **[Diagnosis]**, **[PLC]** and **[System]** which are shifted by the corresponding softkeys. Enter each operation interface by the **[OPT]** key. Refer to the following content for details.

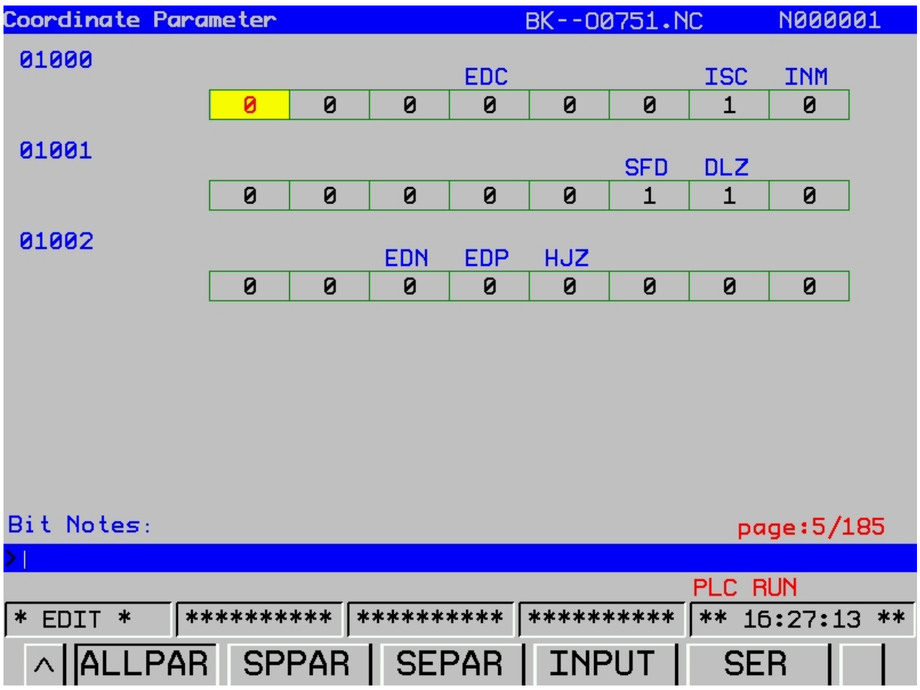


Fig. 3-6-1

1. [ALLPAR]

Press the **[ALLPAR]** key, then the **[OPT]** key to enter the parameter setting interface, five keys are available: **[ALLPAR]**, **[SPPAR]**, **[SEPAR]**, **[INPUT]** and **[SER]**.

Procedures for displaying and setting parameters:

1) Firstly, press the OFFSET SETTING on edit panel to enter the **[LOG]** setting interface, and then input the corresponding password.

2) Enter the system interface by pressing the function key SYSTEM.

3) The parameter interface is displayed by pressing the softkey of **[ALLPAR]**; refer to the Fig.3-6-2.

4) Move the cursor to the parameter number to be set or displayed in either of the following ways.

a. Enter the parameter number and press softkey **[SER]**.

b. Move the cursor to the parameter number using the page keys and , and the direction keys , , and .

5) Input a numerical value by digit keys or **[INPUT]** key.

Checking and modification can be performed by corresponding softkeys, which are as follows:

1) All-parameters interface

Enter the parameter interface by pressing the softkey of **[ALLPAR]**

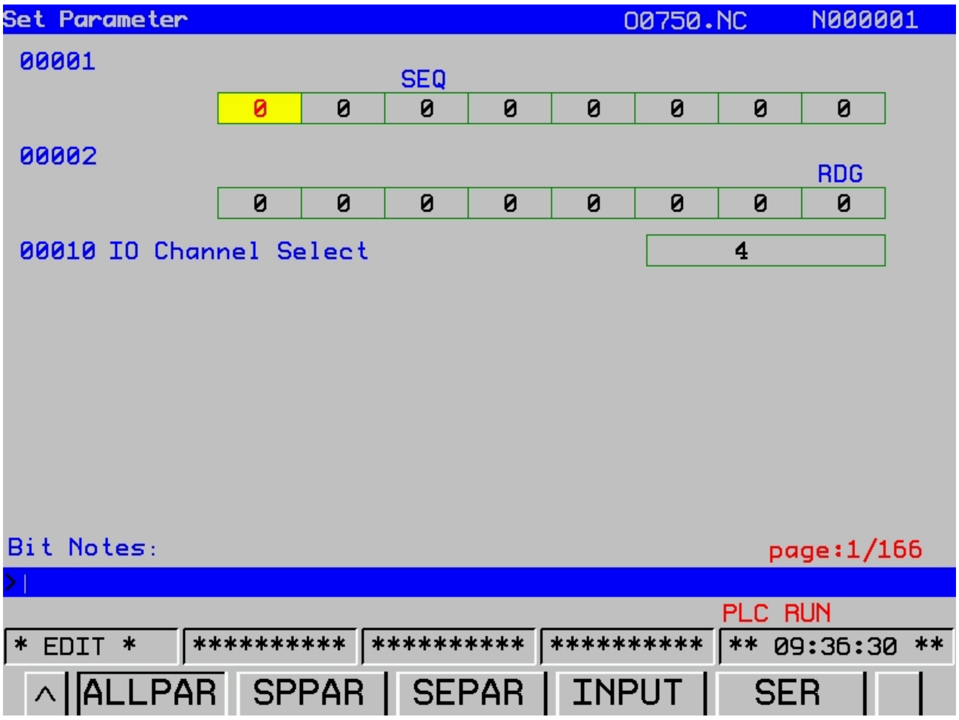


Fig. 3-6-2

2) Spindle parameter page

Enter the spindle parameter interface by pressing the softkey of **[SPPAR]**. (Refer to the figure 3-6-3)

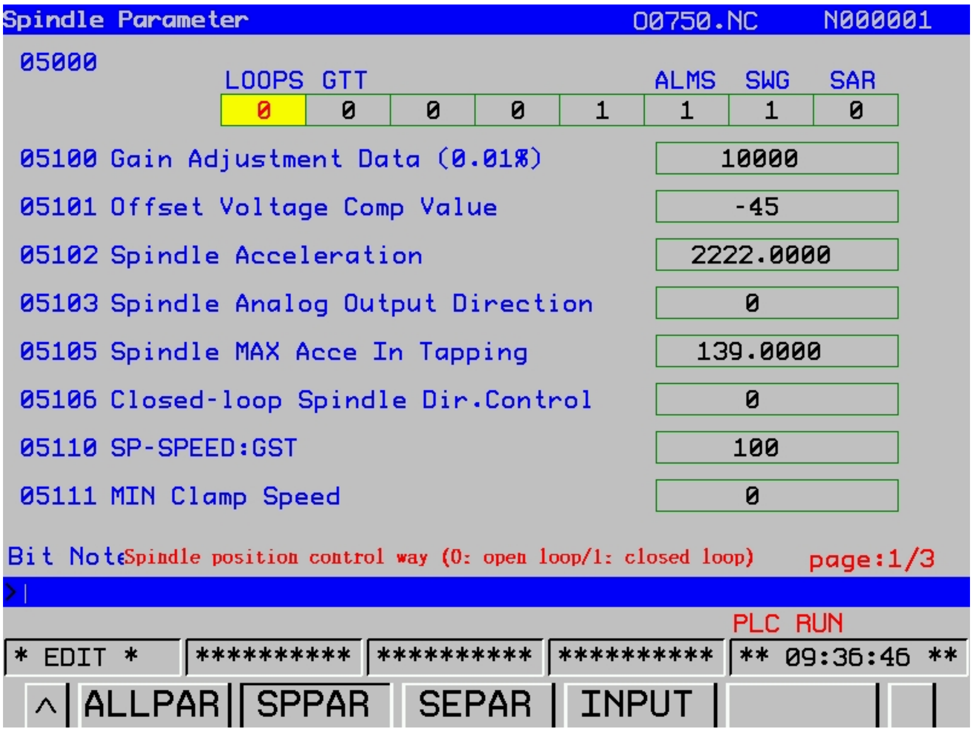


Fig. 3-6-3

Servo parameter interface

Enter the servo parameter interface by pressing the softkey of **[SEPAR]**. (Refer to Fig. 3-6-4):

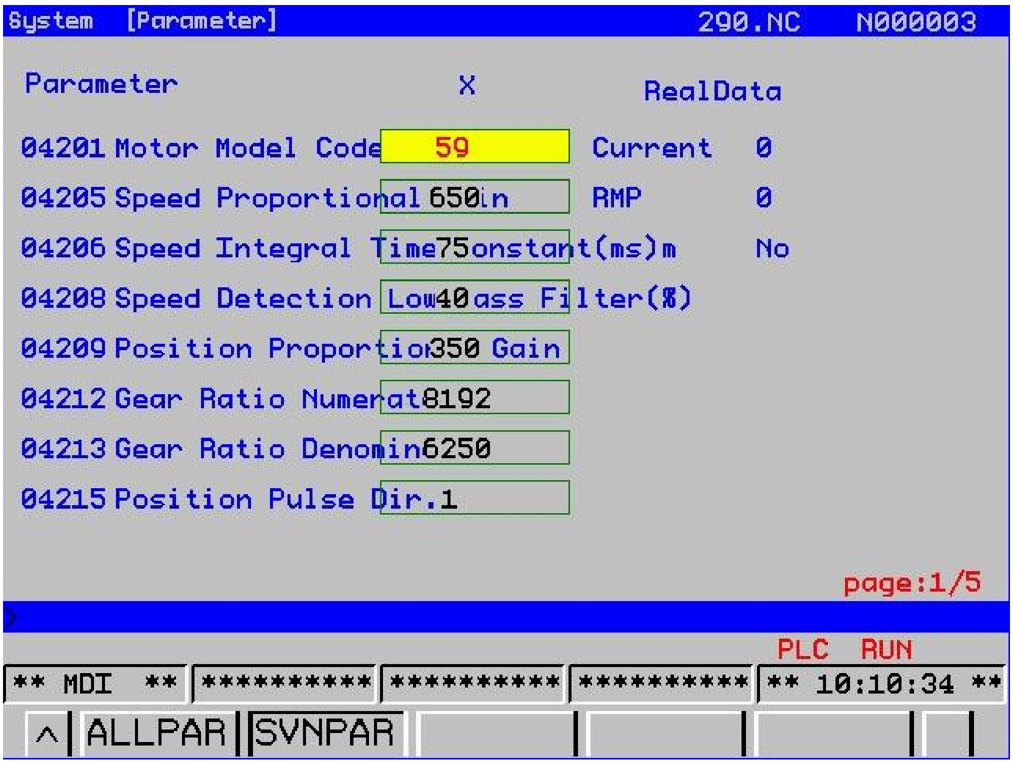


Fig. 3-6-4

Note: Refer to *Volume Ⅲ PARAMETER* of the manual for the definition of each parameter.

2 [Diagnosis]

Press the softkey **[Diagnosis]**, and then the **[OPT]** key to enter a diagnosis display interface.

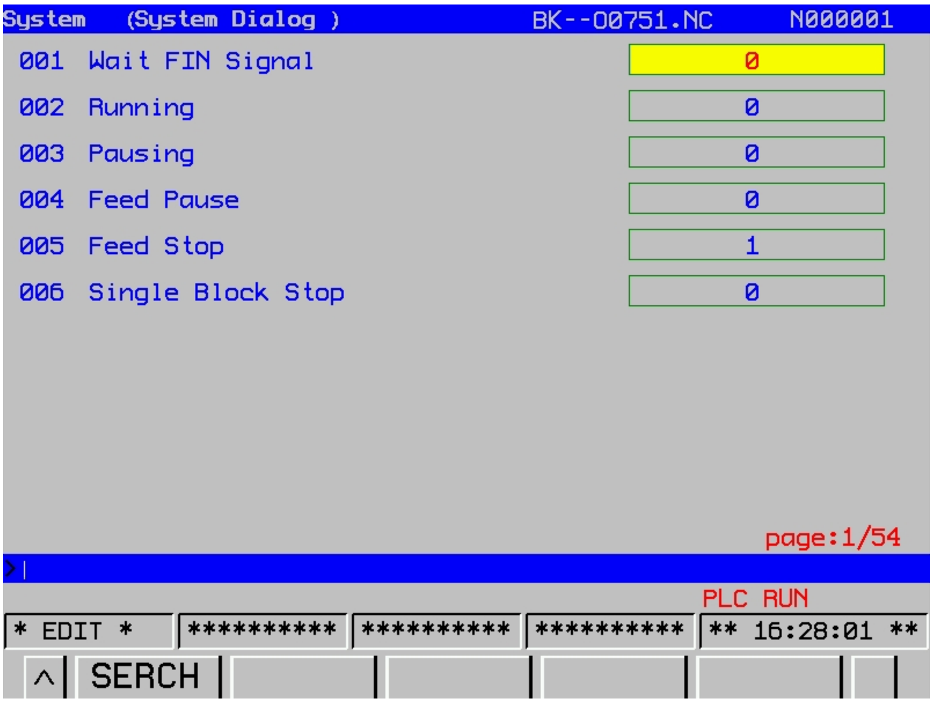


Fig. 3.6-5

Refer to the whole diagnoses by the upward and downward page keys. Search each number by pressing the softkey of **[OPT]**.

3.【PLC】

Enter PLC operation interface by pressing the **[PLC]**. The softkeys, such as **[Integrated display]**, **[PLC diagnosis]**, **[PLC parameter]**, **[File list]** and **[Operation]** are available, wherein, the **[Operation]** is performed for another interface. Enter next interface by controlling the softkey **[>]**, which includes four softkeys: **[Set]**, **[Edit]**, **[Stop]** and **[Operation]**. (Note: The **[Stop]** option does not performed by pressing **[>]** based upon both on the **[PLC parameter]** and **[PLC diagnosis]** interfaces) The details are as follows:

1) Integrated display interface

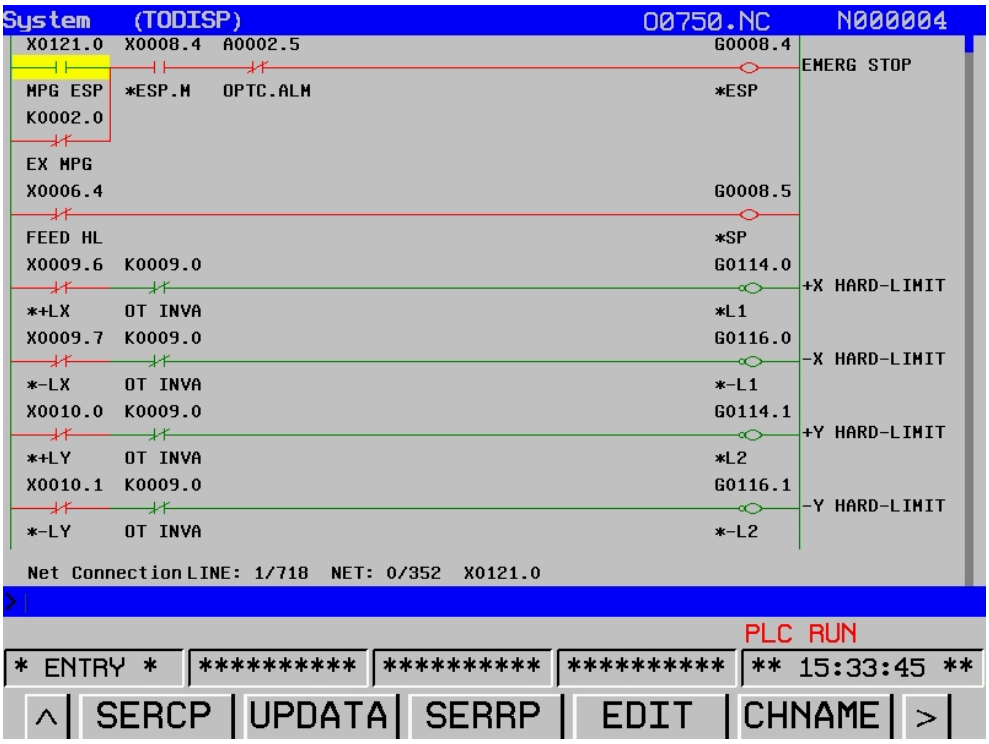


Fig. 3-6-6

Enter the following screen by pressing the **[OPT]** in the mode of **[Edit]**, the PLC program can be modified or edited.

Procedures of operation:

a. The displayed content can be set by the **[Set]** interface. (Refer to Fig.3-6-7)

The integrated display of the ladder diagram can be controlled by the cursor, for example the component name display: move the cursor to the address where it may turn into red, which means that it is selected, the address displays in Fig. 3-6-6 (X0008.4 etc.). The component note display is similar as that of the above component name; for example, the (EEEEEEE) displayed on X0008.4 is a note for this element component. Network line note is at the end of each line at the right side.

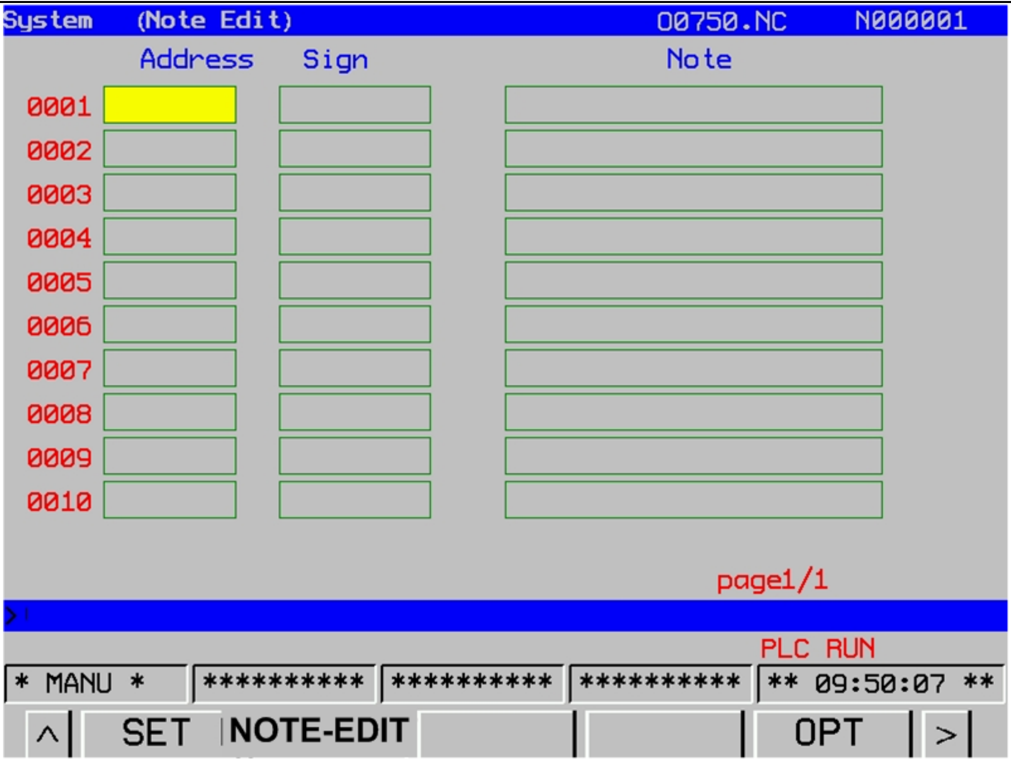


Fig. 3-6-7

b. The network line and component notes can be changed by controlling the softkey **[NOTE EDIT]** and the note can be deleted and the line can be searched.

c. After the display format is set, select the **[Integrated display]**, and then the **[OPT]** to enter an editing and modification interface of the ladder diagram. (Refer to Fig. 3-6-8)

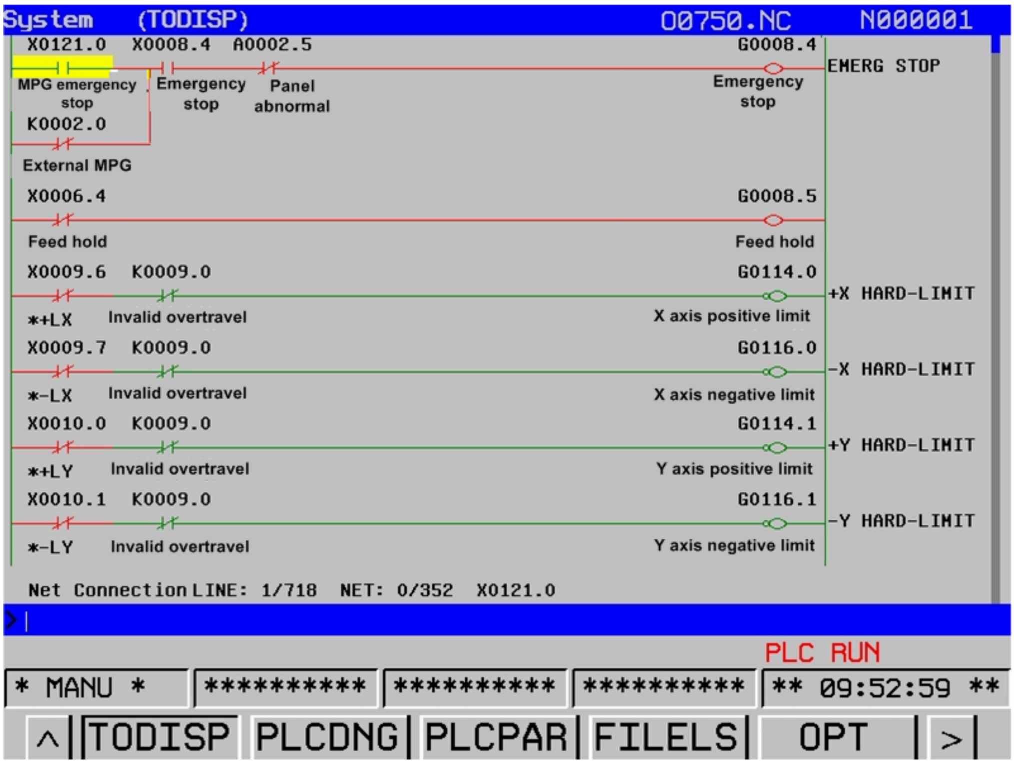


Fig. 3.6-8

The integrated edit of PLC can be performed in the case of the allowable operation authority, for example, the functions of the selection, copying, cutting or and deletion, and it is basically similar as the program interface editing function.

2) **[PLC diagnosis]** interface

The signal states are all displayed on the diagnosis interface, such as the signal state between **CNC** and **DI/DO** of machines, **CNC** and **PLC**, and **PLC** internal data and **CNC** internal state. This diagnosis is used for checking the **CNC** interface signal and internal signal operation state, which can not be modified.

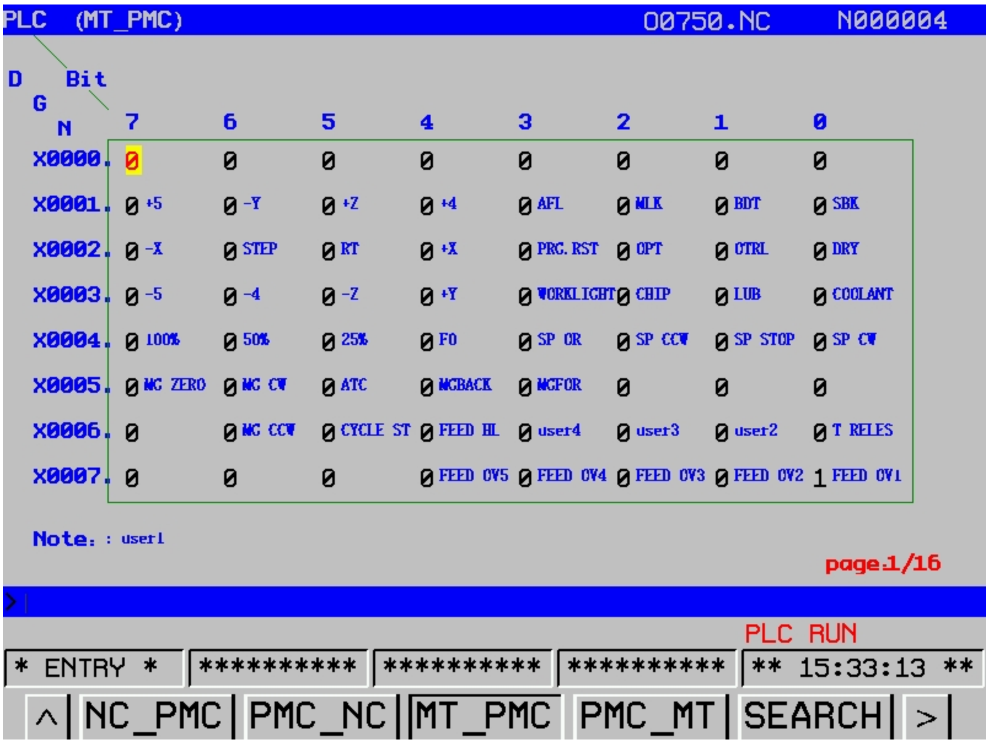


Fig. 3-6-9

Check each parameter by the page key. Enter the following interface by the softkey **[OPT]**:

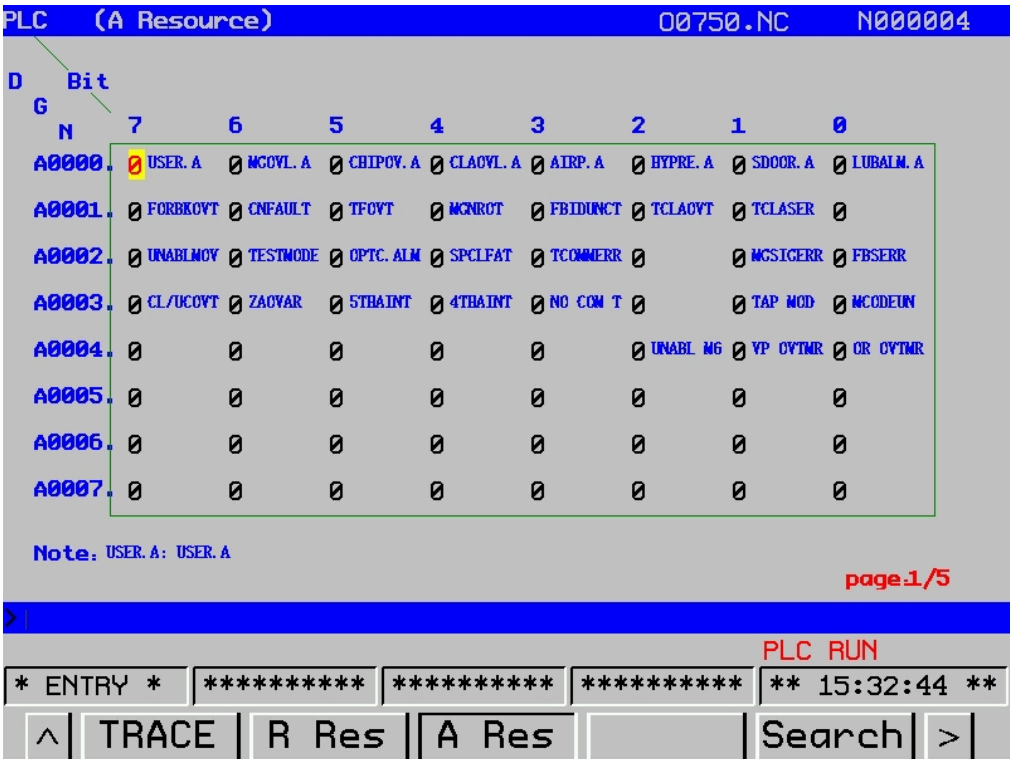


Fig. 3-6-10

Part Ⅱ Operation

Several corresponding interfaces are available: **[F resource]**, **[G resource]**, **[X resource]**, **[Y resource]** and **[TRACE]**, please refer to *Volume Ⅱ: PLC and installation connection* of the manual for the significance of each diagnosis number and setting method. A signal trace is shown on Fig. 3-6-11.

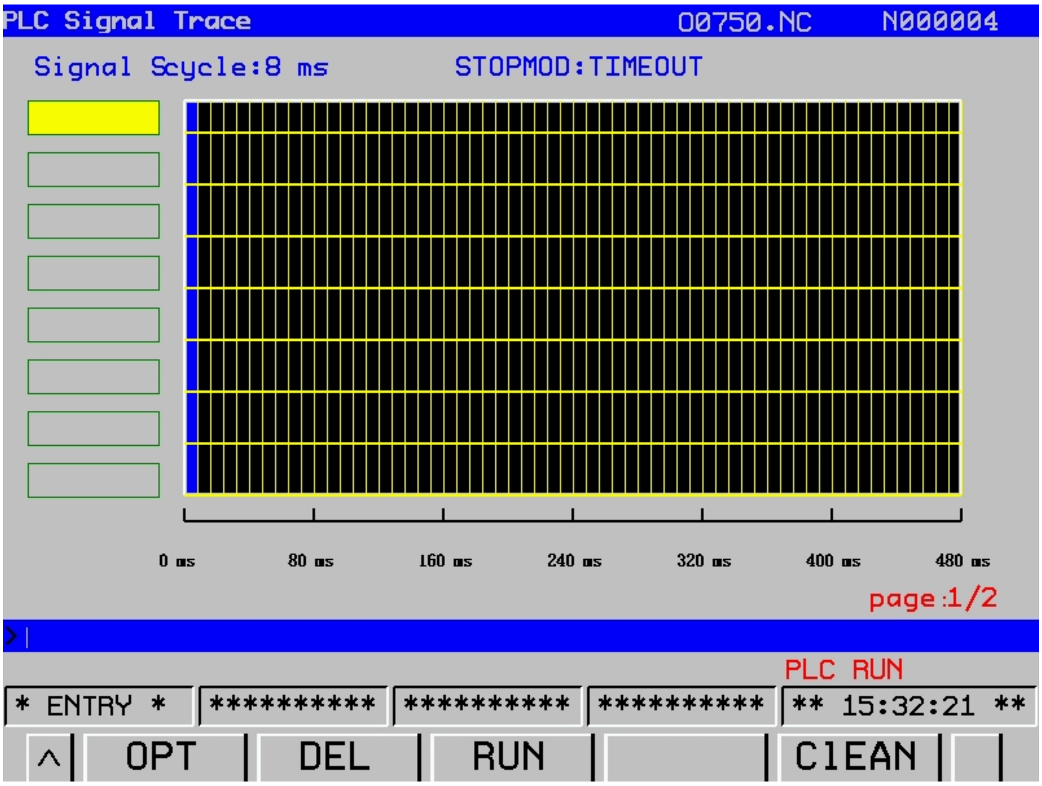


Fig. 3-6-11

The operations are as follows:

A. Enter PLC signal trace interface: press the softkey **[TRACE]**

B. The signal address is input regardless of any operation mode, for example, G0000.5 is input as above-mentioned figure.

C. Press the softkey **[OPT]**, when this signal is performed, the figure 3-6-11 frame may occur. If no signal is transmitted, a straight line is displayed in this case.

3) **[PLCPAR]** interface

Press **[PLCPAR]** to enter PLC parameter setting interface:

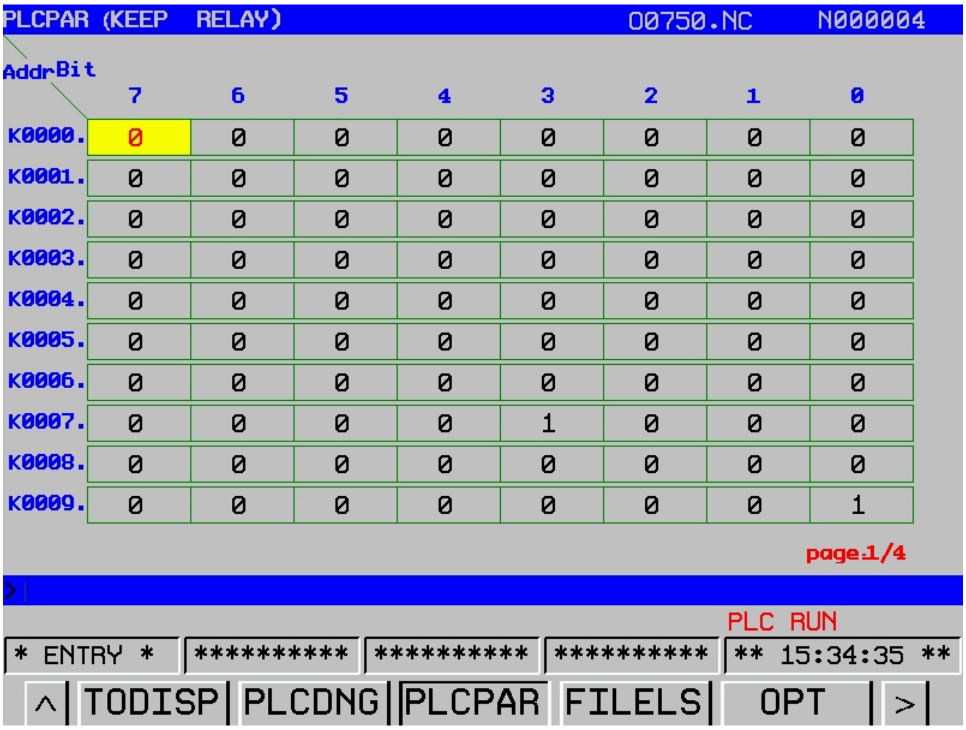


Fig. 3-6-12

Press the softkey **[OPT]** to enter a detailed parameter modification interface:

Relay:

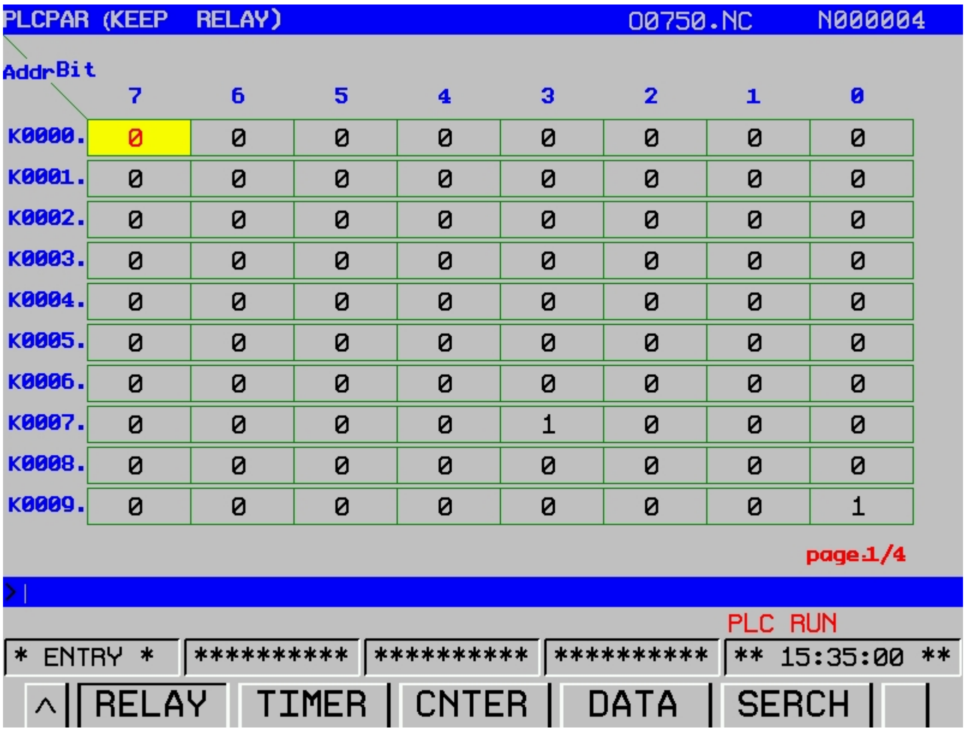


Fig. 3-6-13

Part Ⅱ Operation

Timer:

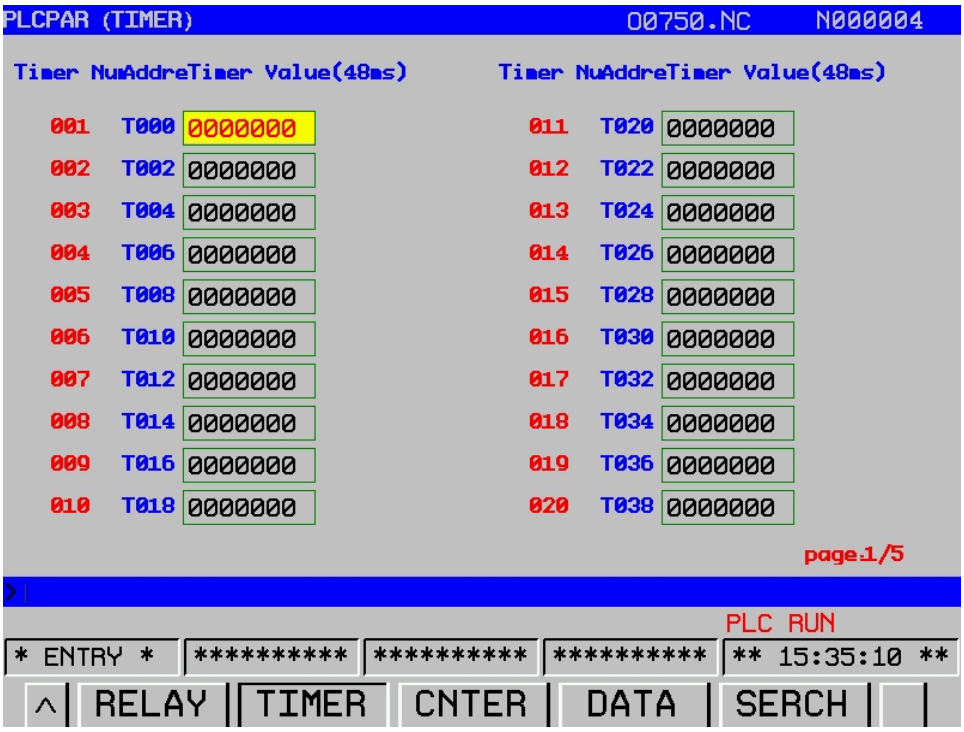


Fig. 3-6-14

Counter

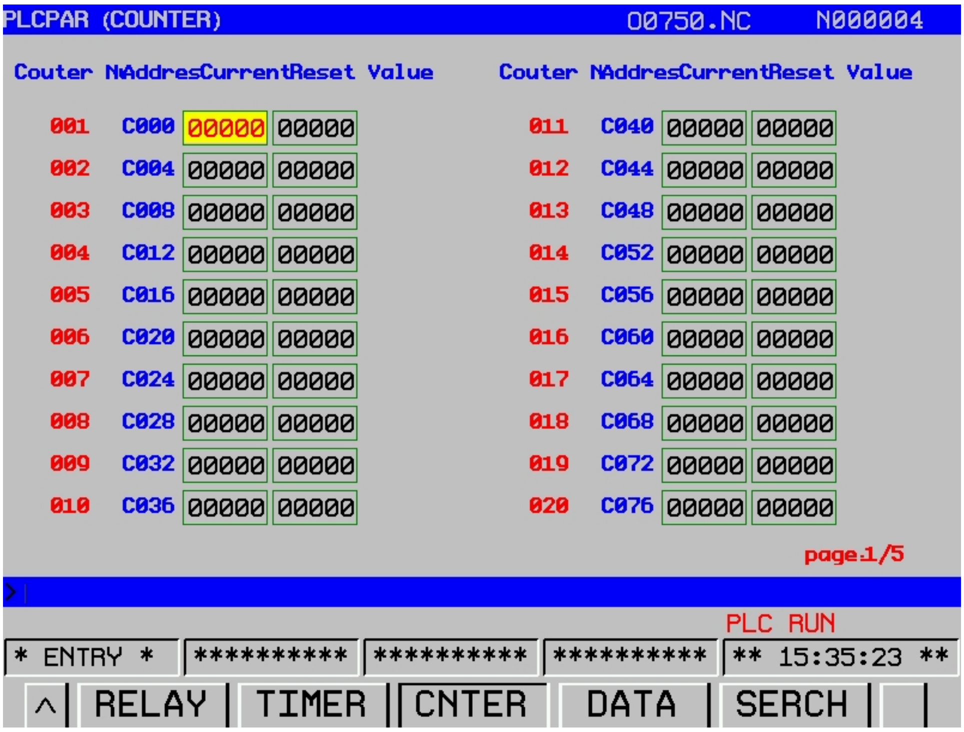


Fig. 3-6-15

Data:

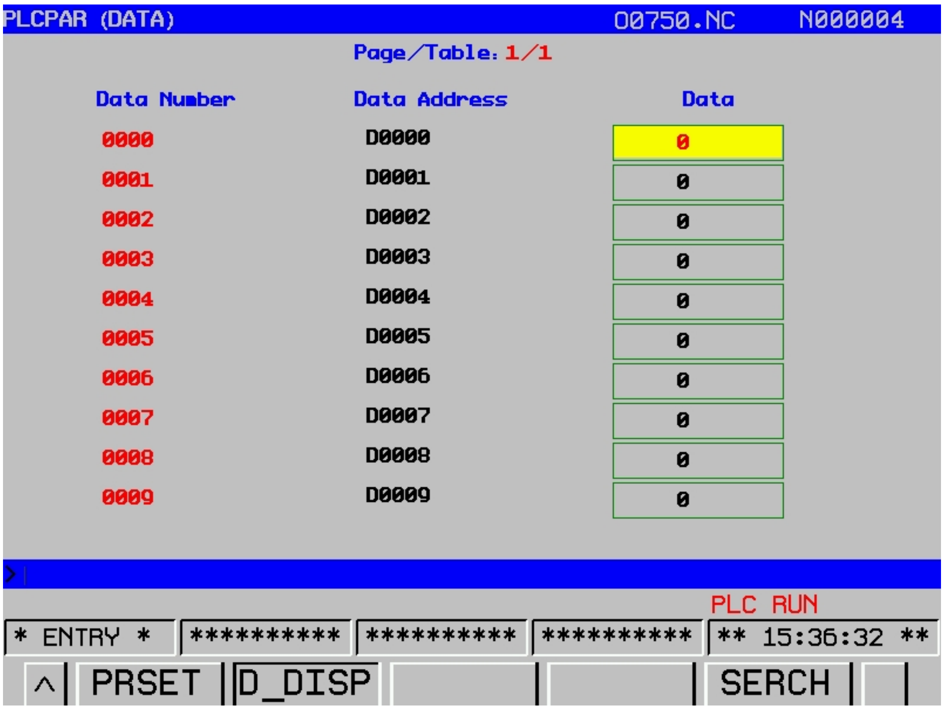


Fig. 3-6-16

4）**[FILELS]** interface

Enter PLC file list operation interface by pressing the softkey of **[FILELS]**, this interface includes:

(a) The stored program number: it includes subprogram.

Remainder: the program number to be registered.

(b) The spent storage capacity: the stored program occupies the storage capacity (it indicates by characters)

Remainder: The unoccupied storage capacity.

(c) The list of existing file name and file size:

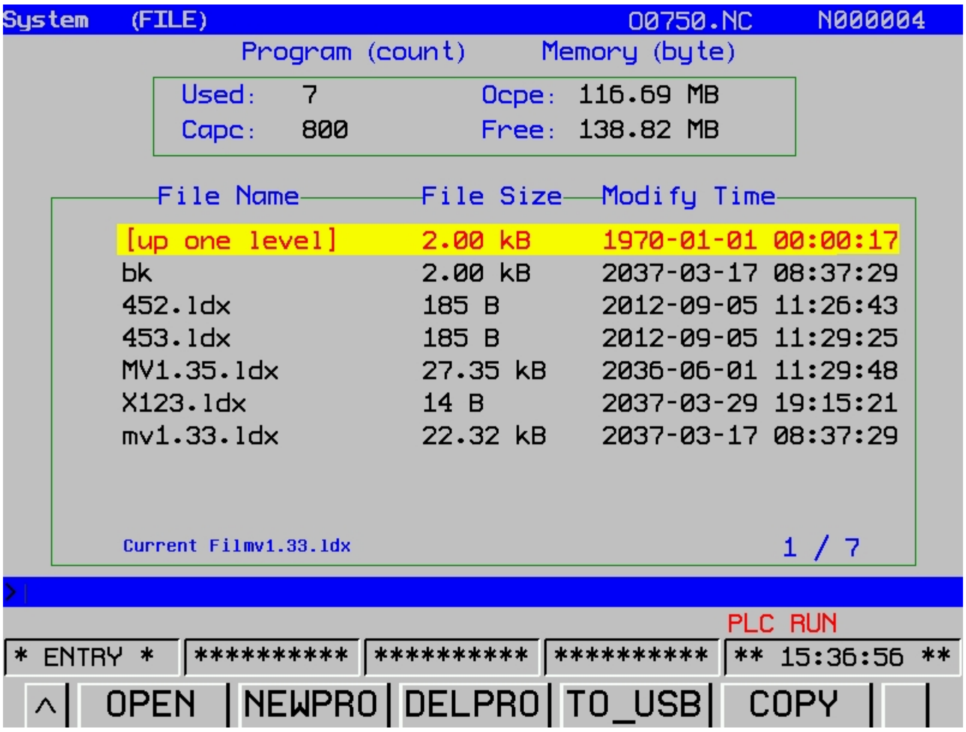


Fig. 3-6-17

Press the softkey **[OPT]** to change and operate the memory program which is similar as the corresponding operation of the file list interface within the program function.

4. **[System]** interface

1) **[SYS]**

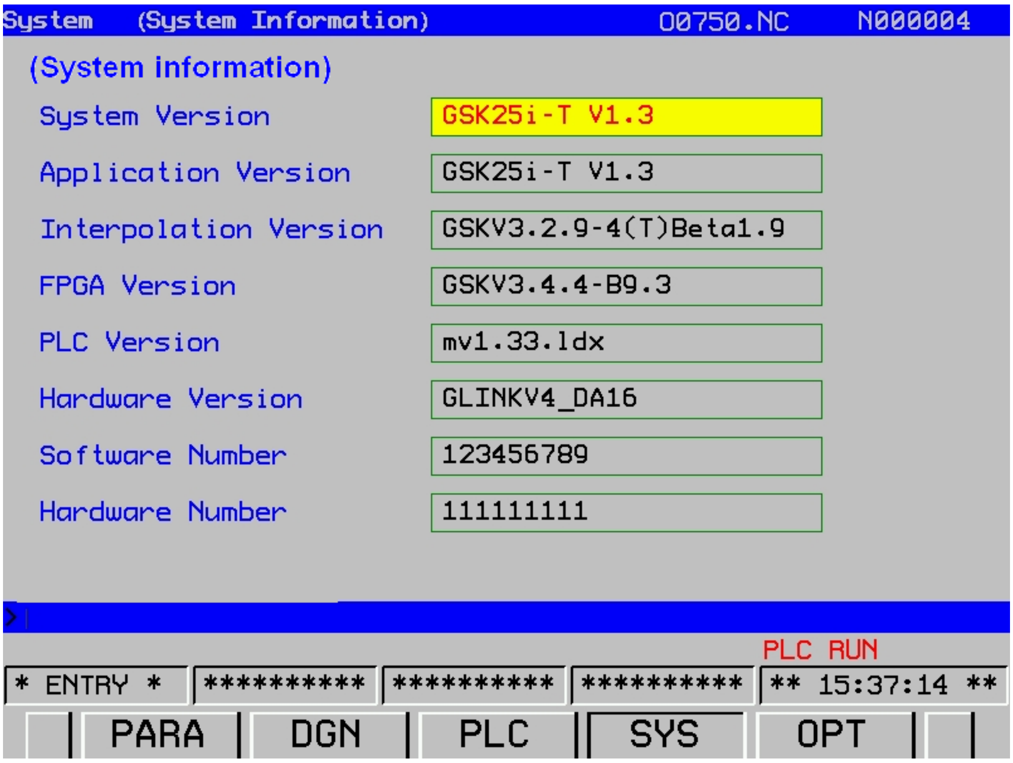


Fig. 3-6-18

This interface displays the current software and hardware version information in the system; the software information can not be modified, but the hardware information can be done in the case of the allowable condition.

2) **[SVNINF]**

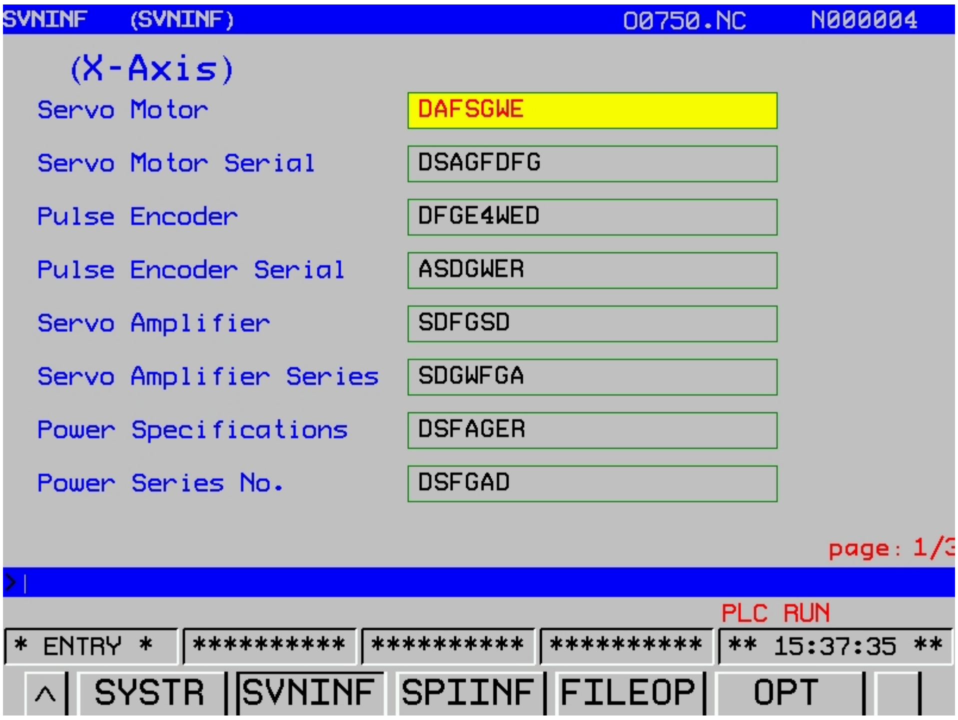


Fig. 3-6-19

This interface displays some character of each axis, which can be modified in the case of the allowable authority.

3) **[SPIINF]**

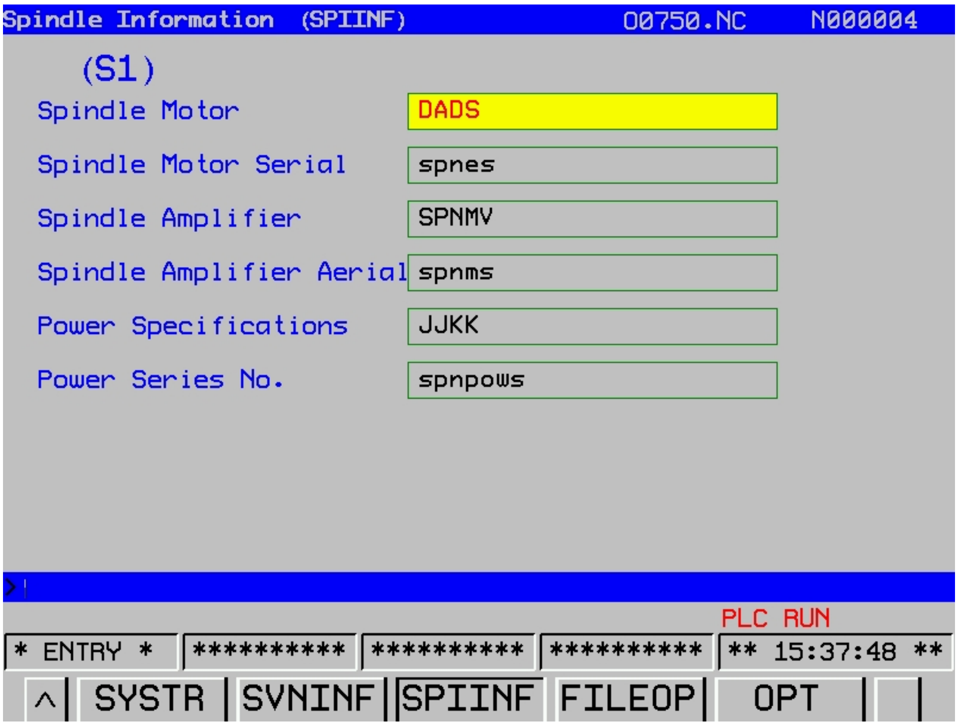


Fig. 3-6-20

This interface displays some relative attribute of the spindle, which can be modified in the case of the allowable authority.

3.7 Help Interface Display

Enter help display interface by pressing the 

, seven display methods are available: **[OPT]**, **[ALM]**, **[GCODE]**, **[PARA]**, **[Macro command]**, **[PLC address]** and **[Calculator]**, which can be checked by the corresponding softkeys. Refer to the following content for details.

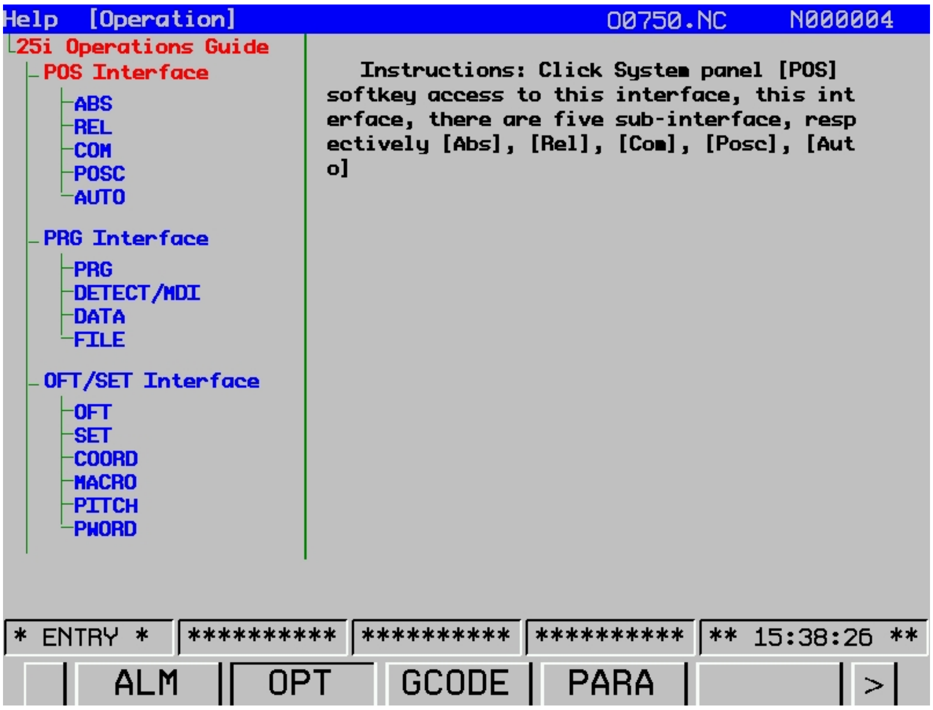


Fig. 3-7-1

1. Operation interface

On the **<Help>** interface, press the softkey **[OPT]** to enter the operation interface.

On the **[OPT]** interface, the manual operation steps and methods on each interface may be generally introduced. If the user does not familiar with the operation or unclear about the content, the search and check can be performed on this interface. Check the relative operation by selecting the corresponding items by pressing the keys ⇧, ⇩, ⇨ and ⇦.

2. Alarm interface

On the **<Help>** interface, enter the alarm table interface by pressing the softkey of **[ALM]**.

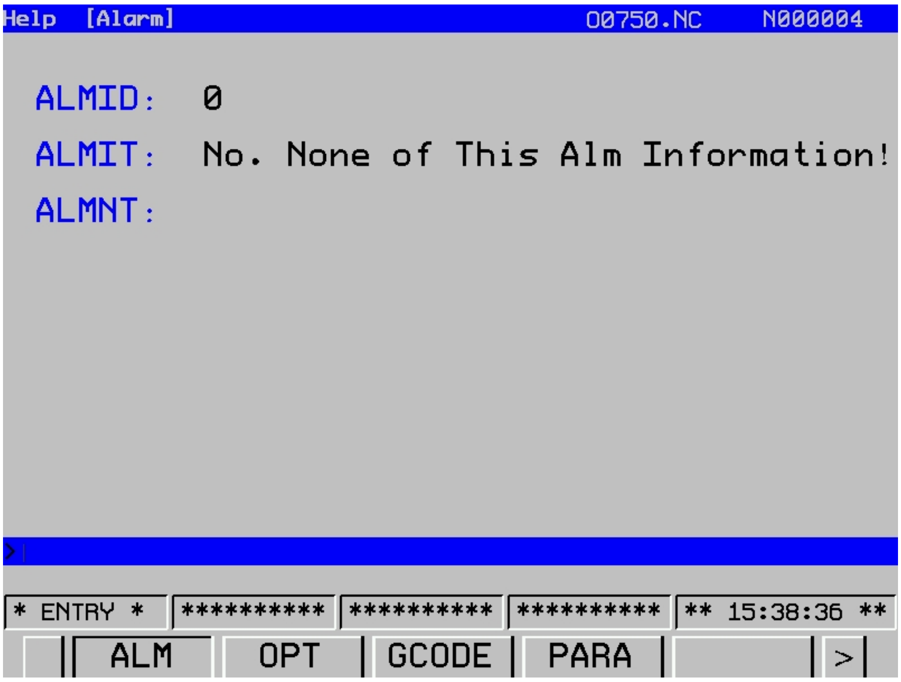


Fig. 3-7-2

The meaning and troubleshooting for each alarm number are described on this interface.

The corresponding content can be checked gradually by the direction keys ⇧ and ⇩. An alarm number can be input in the input column, and press the **[Input]** key to check the alarm number and its meaning which is the related treatment method.

3. G code interface

Press the softkey of **[GCODE]** to enter a G code interface based upon the **<Help>** interface.

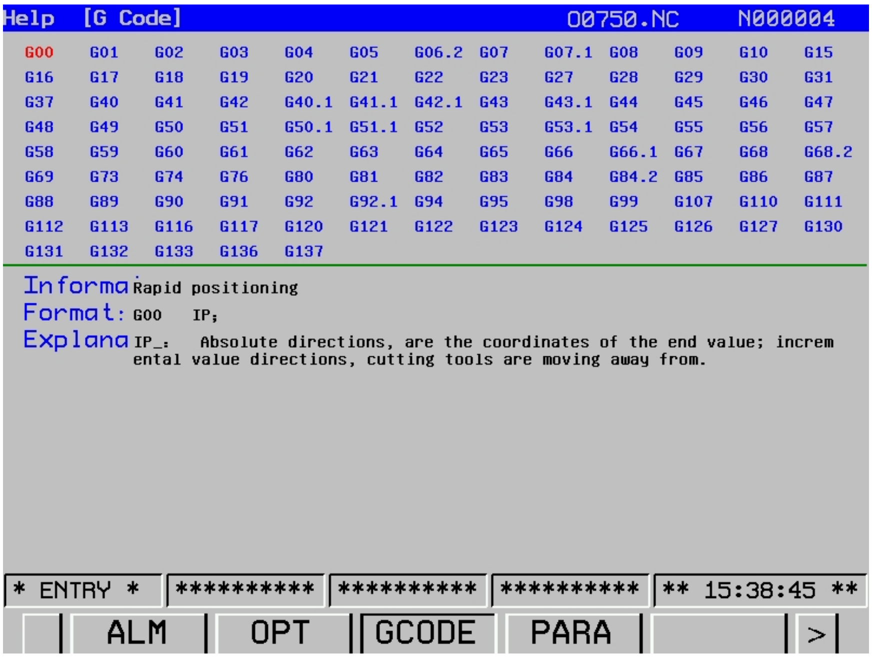
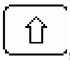
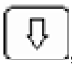
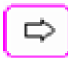
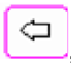


Fig. 3-7-3

G code definition used in the system are described on G code interface, the G code to be viewed should be selected based on , ,  and , and the G code definition is displayed below the interface. Refer to the fig.3.7-3, if the format and usage of G codes is being known, the G code’s relative information can be checked after the G code is selected. The command format, function and explanation are described on this interface, and you can search and check the command that you are not familiar with or clear about on the interface.

4. Parameter table interface

On <**Help**>interface, press the softkey **[PARA]** to enter the parameter table interface.

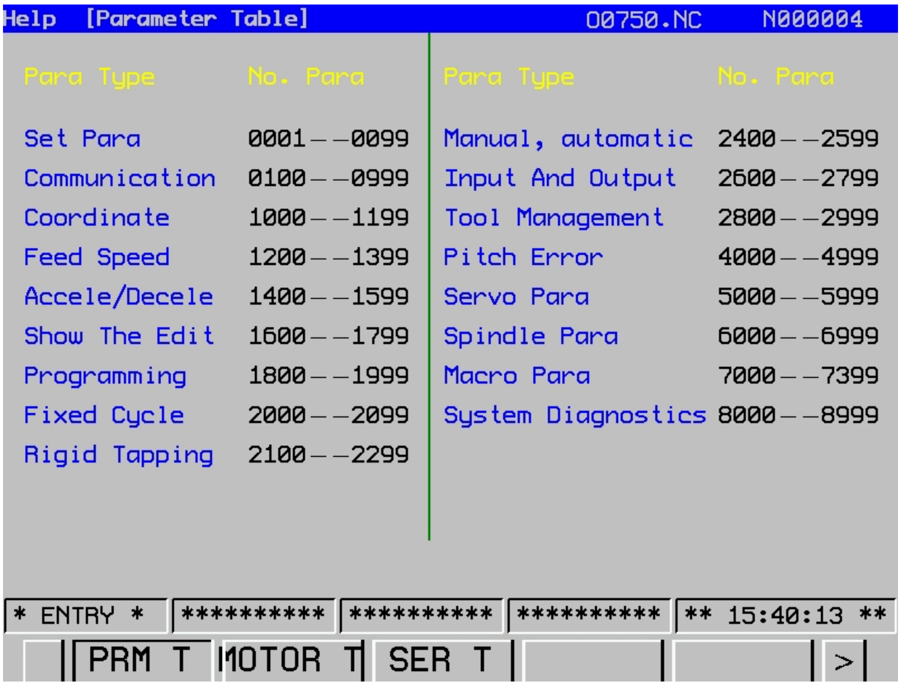


Fig. 3-7-4

On this interface, the parameter range corresponding to the parameter of each function is described, if you are unfamiliar or unclear about the parameter, you can check each parameter for each function in terms of the following parameter appendix on this interface, or the related function parameter search can be performed based upon this range on parameter interface.

5. Marco command interface

Press the softkey **[MACRB]** to enter the macro command interface on the **<Help>** interface.

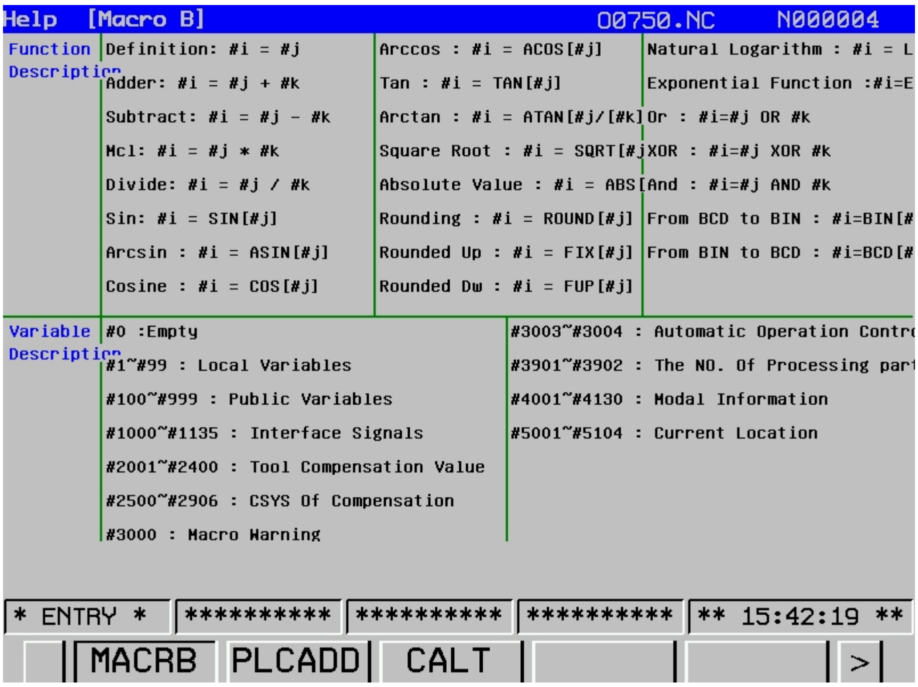


Fig. 3-7-5

The format of macro command and various calculation commands are described on this interface, the local variable, the common variable and the setting range of the system are available. You can search and check the command that you are unfamiliar or unclear about the Marco command on the interface.

6. PLC address interface

Press the softkey **[PLCADD]** to enter the PLC address interface on the **<Help>** interface.

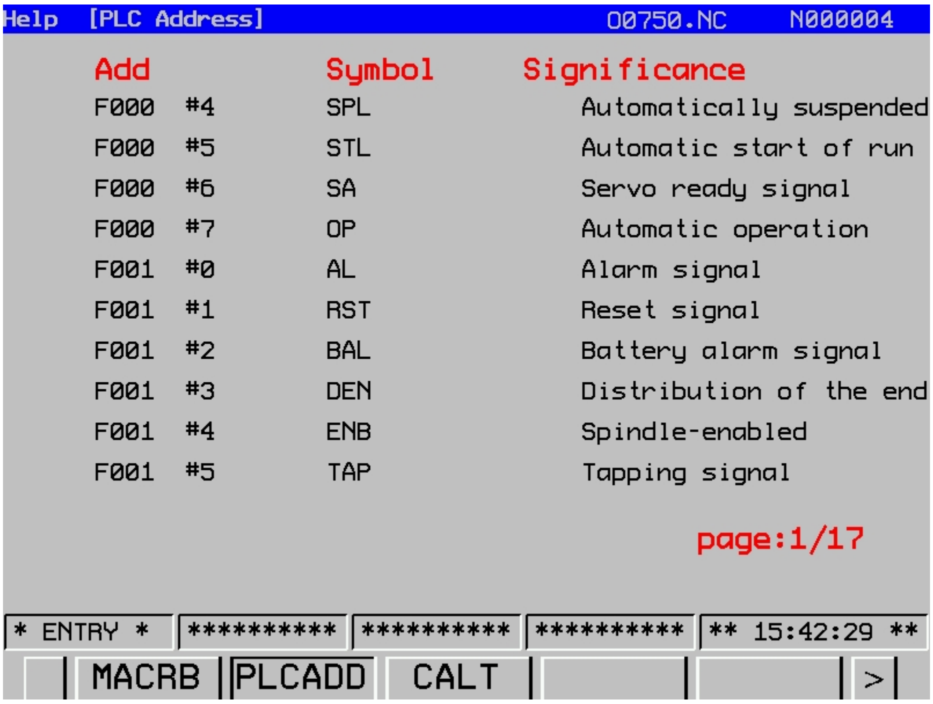


Fig. 3.7-6

PLC address, symbol and significance are described on this interface; you can search and check PLC address that you are unfamiliar with or unclear about on the interface. Totally 17 pages; you can view them by the page keys.

7. Calculator interface

At the 2nd page of **<Help>** interface, press the softkey **[CALT]** to enter the calculator address interface.

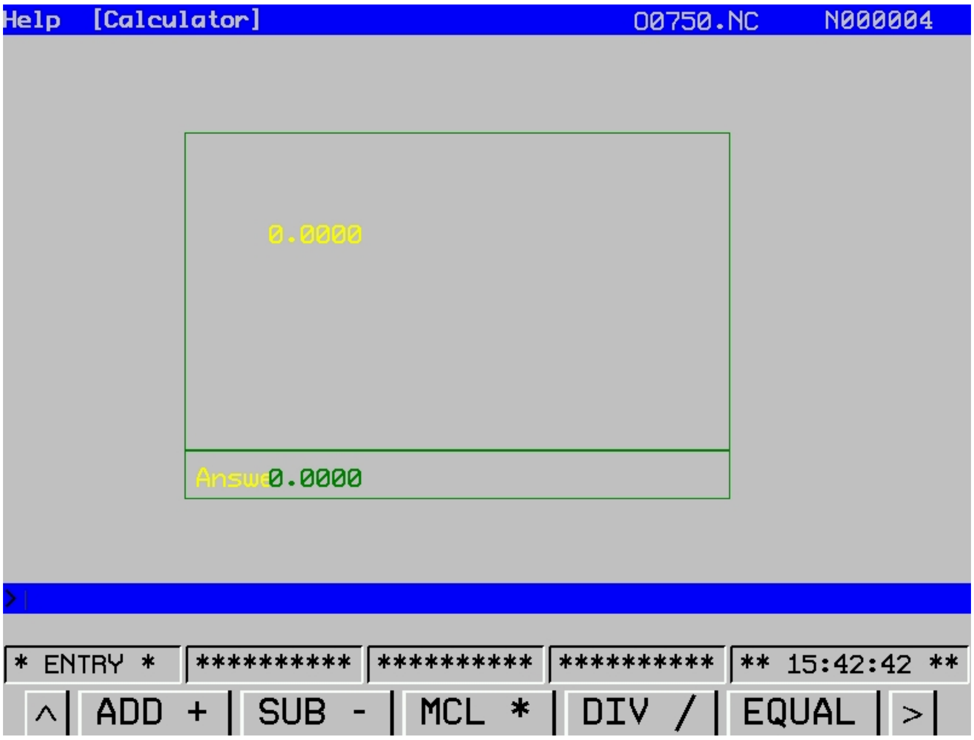


Fig. 3-7-7

On the interface, the operation formats: addition, subtraction, multiplication, division, sine, cosine and evolution are offered. Perform operation in the following two ways:

(1) Operation method for addition, subtraction, multiplication and division: Input data→press operation format key→input data→press equal key→get result.

(2) Operation method for sine, cosine and evolution: Input data→press operation format key→get result.

CHAPTER Ⅳ MANUAL OPERATION

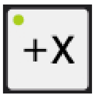
Press 

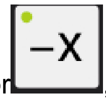
to enter manual operation mode, in which it mainly includes the manual feed, the spindle control and the machine panel control etc.

4.1 Coordinate Axis Move

In the mode of the manual operation, the five-axis can be operated at the manual feedrate or the manual rapid traverse rate.

4.1.1 Manual Feed

Press the feed axis or direction selection key 

or 

, the direction key along with X axis moves the X axis in positive or negative, the axis movement is stopped after releasing the key. In this case, the feedrate override can be adjusted to change the feedrate and the operation is similar as other axes. This system simultaneously supports the manual five-axis movement, and the zero return also can be performed by five-axis.

Note: The manual consecution feedrate for each axis is determined by parameter **N1232;** and the manual rapid traverse rate setting is depended on **N1233.**

4.1.2 Manual Rapid Traverse Move

Press 

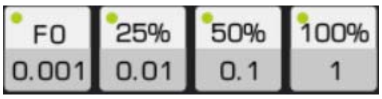
key and enter the manual rapid traverse state after the indicator is lighted up, and then press the key of manual feed axis, each axis operation moves at the rapid traverse rate. The manual rapid traverse is disabled in the manual single step mode.

Note 1: The manual rapid traverse rate is set by **N1233**.

Note 2: The manual rapid traverse move set by bit parameter **N01200#0** is valid before the reference position returns till the power is turned on.

Note 3: The feedrate is performed in manual rapid traverse, and the time constant and the acceleration/deceleration mode are same as rapid traverse rate specified with G00 program commands.

4.1.3 Manual Feed and Manual Rapid Traverse Rate Selection

When the consecution operation is performed, the manual rapid traverse rate can be selected by the 

after pressing the key of rapid operation. Four gears rapid feedrate are available: F0, 25％, 50％ and 100％. (The manual rapid traverse rate is set by **N1233**, F0 speed is set by the data parameter **No1231**). The movement speed can be selected by the feedrate override knob without performing the rapid operation key.

Note 1: The rapid feedrate selection is valid for the following traverse speed

(1) G00 rapid feed

(2) Rapid feed in canned cycle

(3) Rapid feed in the command G28

(4) Manual rapid feed

Example: When the rapid feedrate is 6m/min. If the override is 50%, the speed is 3m/min.

4.2 Spindle Control

4.2.1 Spindle Rotation CW (M04)

: S rotation speed can be specified in MDI mode, the spindle rotates CW by pressing this key in the mode of manual/MPG/single.

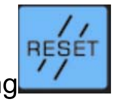
4.2.2 Spindle Rotation CCW (M03)

: S rotation speed can be specified in MDI mode, the spindle rotates CCW by pressing this key in the mode of manual/MPG/single.

4.2.3 Spindle Stop (M05)

: The spindle stops in the mode of manual/MPG/single by pressing this key.

4.2.4 Spindle Exact Stop

: The spindle accurately stops after it rotates to a fixed angle in the modes of manual and MPG by controlling this key. The spindle exact stop should be released by pressing


or performing spindle stop, spindle rotation.

4.3 Other Manual Operations

4.3.1 Coolant Control

: It is a compound key. The coolant is shifted between on and off. The indicator is ON when the power is turned; otherwise, it is OFF.

Note:

1. Initial state of coolant is no coolant output.
2. Coolant key on the panel returns to zero point in Edit, Auto, MDI, Reference return, MPG, Single step, Manual modes. It can be performed correctly when program running or Single step, Skip, Machine lock, M.S.T lock, Dry run, Optional stop are valid.
3. When M08/M09 code and cooling key on the panel are working simultaneously, take the last output cooling state signal.

4.3.2 Lubricating Control

: It is a compound key. The lubricating function is shifted between on and off. The indicator is ON when the power is turned on; otherwise, it is OFF.

4.3.3 Peck Control

: It is a compound key. The peck is shifted between on and off. The indicator is ON when the power is turned on; otherwise, it is OFF.

CHAPTER Ⅴ SINGLE STEP OPERATION

5.1 Single Step Feed

Enter single mode by pressing 

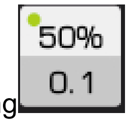
, in the single step feed mode, the machine moves based on the defined step length in the system each time.

5.1.1 The Selection of Movement Amount

The movement increment can be selected by pressing any key of 

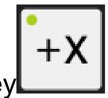
; the indicator lights up and the corresponding movement increment is then selected.

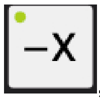
Note: the movement amount on the keys is mm in metric system and ×1/10 in inch system.

Example: The single step length is 0.100 in metric unit by pressing 

. The corresponding machine axis moves for 0.1mm by pressing a move key each time. The single step length is 0.100×1/10=0.0100 in inch unit. The corresponding machine axis moves 0.01inch by pressing the move key each time.

5.1.2 The Selection of Move Axis and Move Direction Key

Press the feed axis and direction key 

 or 

, and the X axis direction key can move in positive or negative direction along with X axis; when the key is pressed for one time, the corresponding axis moves the distance defined by the system single step, and the manual federate is set by parameter No:1232. Because the distance is short, it is not limited by federate override and rapid movement.

5.2 Miscellaneous Control in Single Step Operation

It is same as the manual operation mode, and; refer to the section 4.2 and 4.3 of this operation manual for details.

CHAPTER Ⅵ MPG OPERATION

6.1 MPG Feed

Enter MPG mode by pressing  , in MPG feed mode, the MPG controls the machine movement and the machine feed is accurately adjusted.

The MPG move steps:

The “mode selection” switch is set on the “MPG” position

(1) Select move axis

(2) Rotate the external hand unit of MPG

CW + direction

CCW - direction

(The direction described varies from one machine manufacture to another)

(3) Movement amount: Some panel has the following selection buttons: ×10 indicates that the movement amount multiplies 10; ×100 indicates that the movement amount multiplies 100.

Input system	Movement amount of each grid		
	×1	×10	×100
Metric input	0.001mm	0.01mm	0.1mm
Inch input	0.0001inch	0.001inch	0.01inch

(4) The relation between MPG scale and machine movement amount is as follows:

	The movement amount on MPG of each scale			
MPG increment (mm)	0.001	0.01	0.1	1
Machine movement amount (mm)	0.001	0.01	0.1	1

The numbers displayed on the above table vary from the mechanical drive; refer to the manual of machine manufacture for details.

Note: If the MPG is rotated up to 5 rev/s, the difference may occur between the MPG rotation amount and machine movement distance, so the MPG speed must not be too fast.

6.1.1 Selection of MPG Control Mode

The MPG control mode may be selected by parameter number N1401#4, (0: Reservoir, 1: Real-time). When the parameter is set to 0, the system controls movement amount by reservoir mode, that is to say, after MPG stops, the system continuously performs accumulated pulse that has not been executed. When the parameter is set to 1, the system adopts real-time mode, namely, the movement stops after MPG is stopped. In reservoir mode, pulse in MPG does not lose though it has not transformed to actual movement. In real-time mode, pulse in MPG loses if it has not transformed to actual movement. Generally, real-time mode is recommended.

6.3 The Miscellaneous Control in MPG Operation

It is same as the manual operation mode; refer to the sections **4.2** and **4.3** of this operation manual for details.

CHAPTER Ⅶ AUTOMATIC OPERATION

7.1 Automatic Operation

7.1.1 The Operation Procedure of Automatic Operation Program

A program can be loaded as long as in the mode of edit:

(a) Enter edit operation mode by pressing ;

(b) Enter program list page by pressing , and move the cursor to find a target program file;

(c) A target program file is loaded by pressing the softkey **[OPT]** to select the **[LOAD]**;

(d) Enter automatic mode by pressing . One line to be operated can be selected using the up/down key to enter automatic line.

Note: The current coordinate position is on the end position of the previous block operation which to be operated (If the block to be operated is an absolute programming and it is G00/G01 mode, the current coordinate position does not confirm.);

If the block to be operated is tool-change movement, it is better to confirm the current position does not interrupt or impact to the workpiece; so that the machine may result in the machine behaving unexpectedly, possibly injury to the user.

7.1.2 The Start of Automatic Operation

Press  to operate a program automatically before the program to be started is selected in terms of the section **7.1.1,** and the program operation can be checked after switching to the interface of <Position>, <Check> and <Graph>.

The program operation starts from the line where the cursor is, so it is better to check whether the cursor is on the program line to be needed before controlling the automatic operation key. If it begins from the start line on which the cursor is not located, the automatic operation program can be achieved from the starting line by pressing the  and then .

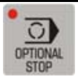
7.1.3 Automatic Operation Stop

During program automatic operation, the system provides six methods to stop the automatic operation program:

1. Program stop (M00)

The program operation dwells after the block containing M00 is performed, all modal information is totally registered. The program is continually performed after pressing .

2. Program optional stop (M01)

If  is controlled before the program is operated, the program dwells after it is executed to the block including M01, and all modal information is totally registered. The program can be continually performed after pressing . If the  does not press, it is regarded as code that the M01 does not executed.

3. Press 

The machine displays in the following status after pressing  in automatic operation:

1) Machine feed decelerates to stop;

2) The machine still stops when a dwell (G04 command) is performed;

3) The other modal information is registered;

4) The program is continually performed after pressing .

4. Press 

Program skips to the head of the program by pressing the reset key, and the reset key is enabled when the **[RESET]** is displayed on its interface. The program is performed from beginning after pressing . The reset key is pressed in auto mode during continually operation, the cursor stays in the current line. After pressing , the current program is performed. <u>Press reset key in DNC mode to clear program.</u>

5. Press <**ESP**> 

6. Mode shifting methods

When the program can be performed on the MDI interface of Auto, DNC and MDI modes, the machine can be stopped after shifting to the other modes. The details are as follows:

1) Shift to the manual, MPG or zero return mode, the machine decelerates to stop.

2) Shift to the single-step mode, the machine stops after the current block is performed.

3) When mutual shift is performed among Auto, DNC and MDI modes or shift to MDI mode, the machine stops after the current block is executed.

7.1.4 Spindle Control Speed in Automatic Operation

The spindle speed can be adjusted in Auto operation when the analog amount controls the spindle speed.

When the automatic operation is executed, the spindle speed is changed along with the spindle override varies by pressing the spindle knob, and the spindle override can be achieved **50%～120%,** totally 8-level real-time adjustment:

Spindle actual speed = program command speed × spindle override. The maximum spindle speed is determined by parameter No.5142. If it exceeds this digit speed, it is then rotated at the speed.

7.1.5 Speed Control in Automatic Operation

When the automatic operation is performed, the system can change the feedrate by modifying the override.

The federate override can be modified by the rotation knob, and it can be achieved **0％～200％,** totally 21-level real-time adjustment.

Note: In feedrate adjusting program, the programming speed set by F is modified.

The actual feedrate = the value set by F × feedrate

However, in the automatic operation, the rapid traverse speed can be selected by pressing the 

, and the rapid override can be achieved four-gear adjustment, namely, F0, 25％, 50％ and 100％.

Note 1；The calculation of the rapid traverse speed at final modification is as follows:

The maximum aggregate speed determined by axes = the maximum rapid traverse speed of axes (No:1226) ÷ vector of corresponding axis

The aggregate speed in rapid feed = the minimum value in the maximum aggregate speed determined by axes

The rapid traverse speed at the final modification = the aggregate speed in the rapid feed × rapid federate override

Note 2: Rapid override is F0, when No.1200.4=1, the system decelerates to stop.

If No.1200.4=0, the rapid federate is set by No.1231.

7.1.6 Dry Run

The program can be checked by the "dry run" before the program is automatically operated.

Enter automatic operation mode by pressing 

, then 

(In the state indication area, the indicator goes on means that the dry run is performed).

The program speed in rapid feed is dry run speed.

The program speed in cutting feed is dry run speed.

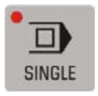
Note: 1. The dry run speed is determined by data parameter No1210 (Generally use for all axes);

2. Whether the dry run is enabled at rapid feed which is determined by bit parameter No.1200＃

7.1.7 Single Block Operation

Before the automatic operation, the program single-block operation can be selected if its operation situation is required to be checked.

Enter the automatic operation mode by pressing 

, then the 

(In state indication area, the single-block operation indicator is ON means that the single operation state is performed). When the single-block operation is executed, the system stops running after each block is completed. In this case, it is necessary to press 

again if you want to perform it continually, and the operation should be repeatedly executed till the program running completes.

Note: In G28, the single block stop can also be performed at an intermediate point;

7.1.8 All Axes Function Lock Operation

Press 

(In the state indication area, the machine lock operation indicator **ON** means that the machine lock operation state has been performed) in automatic operation mode. In this case, the machine does not move, but the position coordinate display is same as that of the machine movement, the current operation situation can be checked from **[Monitor]** interface, and then the **M, S and T** can be performed. This function is used for program checking.

Though the machine does not move and the position coordinate does not change, the CNC executes limit detection for saved stroke according to machine coordinate calculated by movement command.

7.1.9 Miscellaneous Function Lock Operation

Press  (In the state indication area, the miscellaneous function lock operation indicator **ON** means that the M.S.T function lock state has been performed) in automatic operation mode. In this case, M, S and T code commands are not performed, which is used for a program check.

Note: M00, M30, M98 and M99 are performed normally.

7.1.10 Manual Intervention

Manual intervention function is mainly used to stop and move feed axis manually in automatic operation. After manual operation, return to the beginning position of manual intervention and run continually by performing cycle start. The procedures are: press 【Feed hold】in automatic operation; Switch to Manual or MPG mode and move corresponding axis to a safe position; Perform necessary operations (such as tool changing, chip removal etc.) and then switch to Auto mode; Corresponding axis returns to automatic stop position and runs continually after pressing 【Cycle start】key.

Notes:

1. Set parameter #1950.0 to 1 before using this function, which can only be used in Auto mode.
2. Rapid traverse speed G00 is used in returning operation, manual federate override and signal block functions are valid.
3. In manual intervention or returning operation, the function is cancelled if reset is performed or alarm occurs.
4. After performing the manual intervention function, tool compensation can not be restored.
5. The returning sequence of axes is decided by actual processing requirements. It is set by parameter #1960.
6. The manual intervention is valid in group 01 (G01 G02 G03).
7. In canned cycle and special canned cycle, manual intervention is not valid.
8. The manual intervention is invalid in subprogram, macro program and G codes in group 00.
9. The manual intervention is invalid in tool length compensation.
10. Perform operation carefully based on steps of the manual. Do not try unconventional operation, otherwise, unpredictable danger may occur.

7.2 MDI Operation

The MDI operation function is added with which the command operation can be directly input.

7.2.1 MDI Program Edit

In MDI mode, after the code is input, the functions, namely, the search code, search line number as well as the selection, copy, cut and paste which can be performed similar to the editing mode. Cancel the input code in the column by pressing **【Cancel】** key. The contents where cursor locates can be replaced by pressing **【Alter】** key.

If the field input is incorrect before pressing the **【cycle start key】**, cancel the input code one by one by pressing 

; if a mistake occurs after inputting, the incorrect one where the cursor locates can be replaced by pressing 

.

7.2.2 MDI Command Operation and Stop

The MDI can be operated by pressing 

after the command is input. The command operation can be stopped by pressing 

during operation. At the end of the program without M30, the cursor does not skip to the top of the program after the operation is executed.

Note: MDI operation must be performed in the MDI mode.

7.3 Conversion of Operation Modes

The automatic operation may stop immediately after shifting to Manual, MPG and machine zero, the feed hold indicator goes on. In the automatic operation state, only when the current line is enabled before shifting to the MDI, DNC and Edit mode.

The MDI mode operation may stop immediately after shifting to the Manual, MPG and machine zero modes, the feed hold indicator goes on. The current line is performed before shifting to the Auto, DNC or Edit mode.

CHAPTER Ⅷ ZERO RETURN OPERATION

8.1 Machine Zero Return

8.1.1 Machine Zero Point Concept

Machine coordinate system is a fixed one of machine, of which its origin is called as machine zero point, and it is also referred to as a **Reference point**; generally, it is installed at the maximum stroke along with the X-axis, Y-axis, and Z-axis positive direction. This fixed origin is confirmed after the machine is designed, manufactured, and debugged. Normally, the machine zero point can not be recognized in the CNC with incremental position detection device till the CNC power is turned on, the machine zero point return is required. The machine zero point is memorized after it is set once in the CNC with absolute position detection device. Machine zero return is unnecessary though power-on again.

8.1.2 The Operation Procedures of Machine Zero Return

1. Automatic zero return

1) Enter the mechanical zero return operation mode by pressing 

, the "zero return" can be displayed on the LCD screen, in this case.

2) The X, Y, Z as well as the 4th or 5th axis, which is to be returned the machine zero point, is selected, and the zero direction is determined by bit parameter **No.1004＃5.** (This system supports five-axis zero return simultaneously)

3) The machine moves along with its zero point direction, the machine rapidly moves before decelerating (The move speed is set by data parameter **No.1235**), the machine detaches to the deceleration switch at FL speed (it is set by data parameter **No.1234**), and then it moves to its zero point (it is also called as reference position) at the second FL speed (it is set by data parameter **No.1236**). The coordinate axis stops and zero return indicator is power-on when the machine zero point is returned.

Note 1: Never attempt to use machine zero return operation if your CNC machine does not install it;

Note 2: The corresponding indicator lights up when the machine zero point return is completed

Note 3: The indicator is power-off after the machine zero point is returned if operator moves out a corresponding axis from the machine zero point;

Note 4: Refer to the manual issued by the manufacturer for the operation of machine zero return varies from one machine manufacture to another.

8.1.3 The Debugging Method of Zero Return

1) The related parameter of zero return:

Zero return direction setting (1004#)
Movement amount per revolution for each axis (1060#)
FL speed of reference position return for each axis (1234#)
Reference position return speed for each axis (1235#)
The 2nd FL speed of reference position return for each axis (1236#)
Mechanical zero return acceleration (1444#)

2) Zero return schematic chart

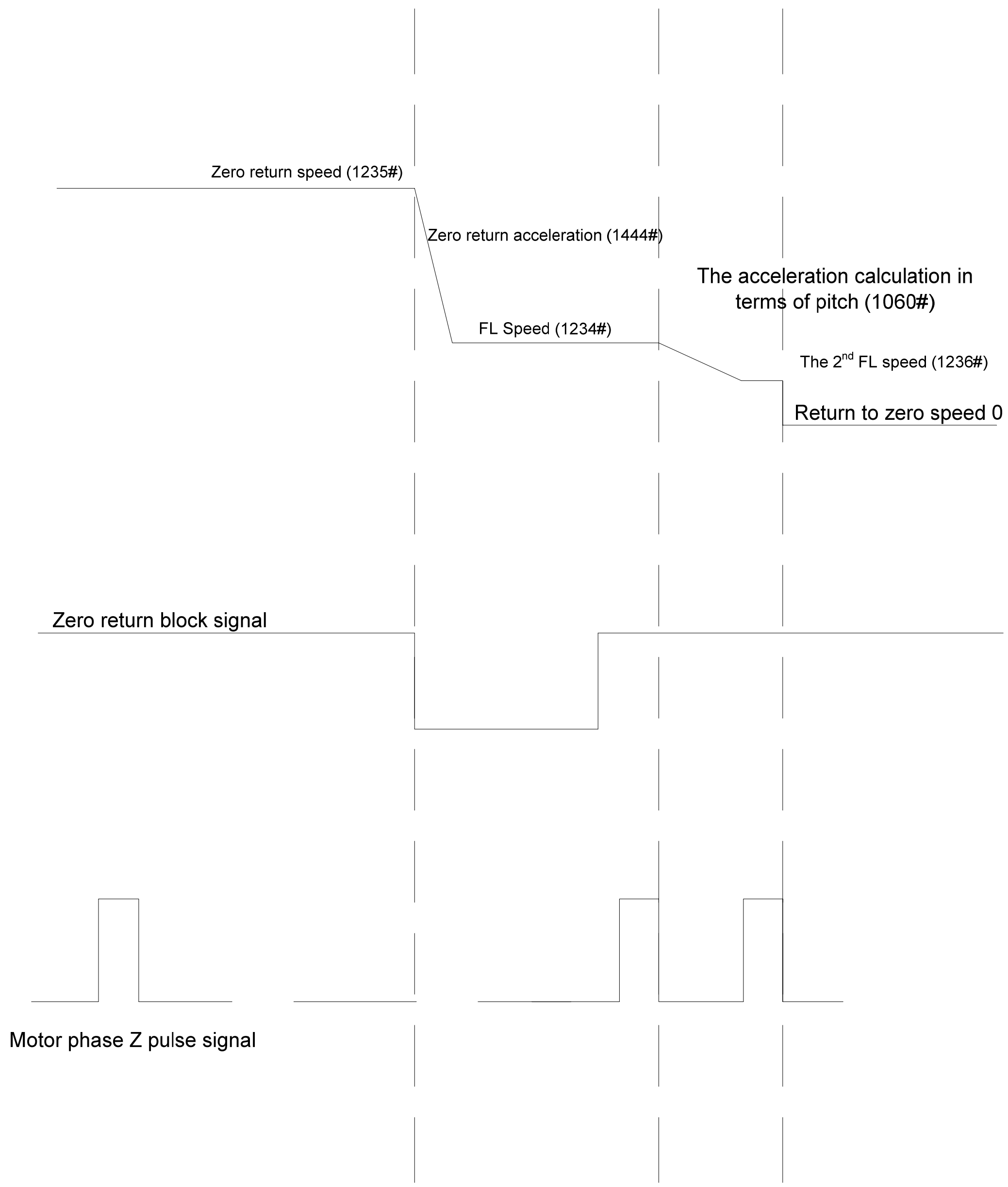


Fig. 8-1-1

3) The adjust steps of zero return parameter.

A. Confirm zero return direction (1004#) in terms of machine condition.

B. Confirm the movement amount (1060#) per revolution for each axis based upon machine condition.

C. Set zero return speed and zero return acceleration

Reference position speed return for each axis (1235#) (default: 3000, 3000, 3000, 2000, 2000, and 2000)

FL speed of reference position return for each axis (1234#) (default: 300, 300, 300, 75, 75, and 75)

The second FL speed of reference position return for each axis (1236#) (default: 7, 7, 7, 2, 2, and 2)

Acceleration speed mechanical zero return (1444#) (default: 0.3, 0.3, 0.3, 150, 150, and 150)

D. Confirm whether the zero return block signal for each axis is normal.

Enter PLC diagnosis interface, and select the X resource.

The machine moves manually when machine passes the zero return block position. View whether the zero return block signal (X9.0~X9.4) input has a corresponding change.

E. Zero return operation for each axis is performed separately

View whether the reference position speed return (1235#), FL speed (1234#) and the 2nd FL speed (1236#) for each axis is held more than 2 seconds.

If the reference position return speed (1235#) is held less than 2 seconds, it is essential to move this axis along with zero return negatively depart from the reference position. The zero return acceleration (1444#) parameter should be increased if it is arrived to a movement terminal.

If the FL speed of reference position return (1234#) is held less than 2 seconds, it is necessary to increase the zero return acceleration (1444#) parameter. If the acceleration increase does not valid, the block length is short. In this case, attempt to reduce both the reference position return speed (1235#) and FL speed (1234#).

If the 2nd FL speed (1236#) of reference position return is held less than 2 seconds, confirm whether the motor move amount parameter per revolution is correct firstly; if it is correct, the phase Z signal may be abnormal.

CHAPTER Ⅸ SYSTEM COMMUNICATION

This system can be communicated with PC by the series terminal port and Ethernet, as well as read the USB device directly. Refer to the operating explanation for details:

9.1 Series Terminal Port Communication

GSK25i serial terminal port communication software is window interface, which is used for DNC machining from PC port to CNC port. This software can be applied to Win98, WinMe, WinXP and Win2K.

9.1.1 Program Start

The “25i_DNC” program in desktop shortcut mode is performed directly; the interface is displayed after the program starts, refer to the following figure (9-1-1):

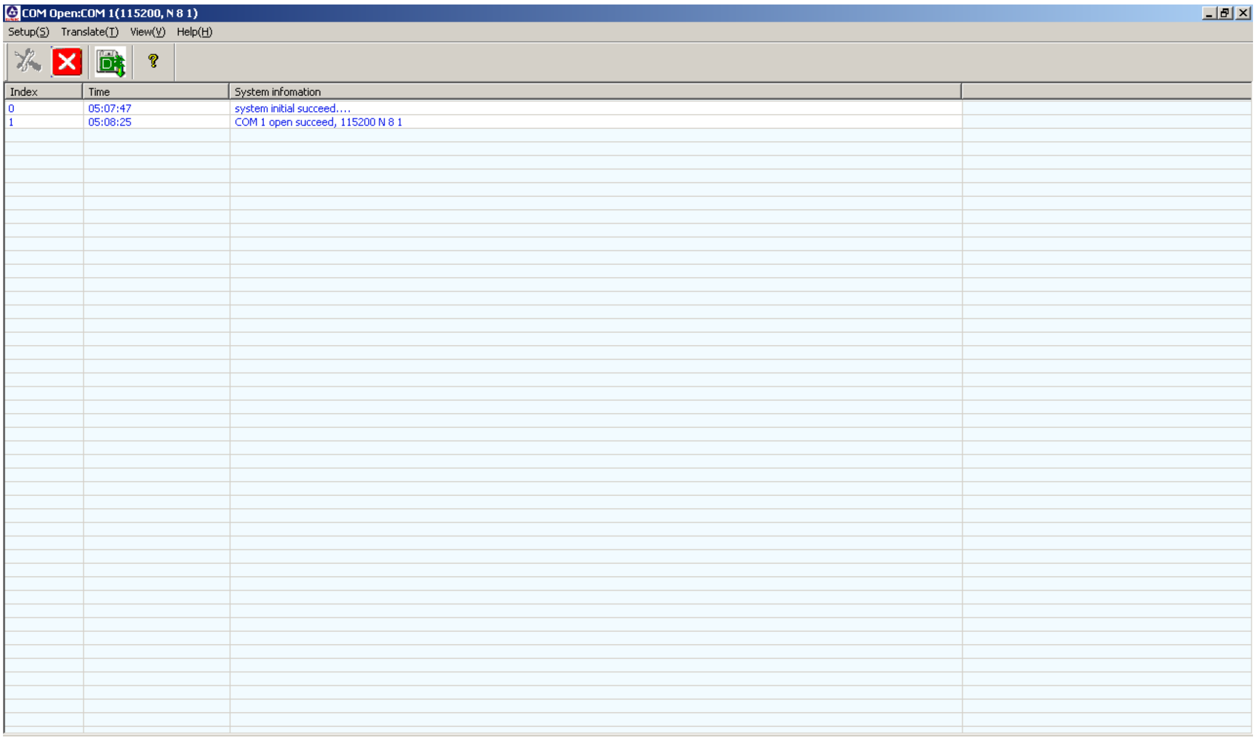


Fig. 9-1-1

9.1.2 Software Usage

1 Connection to CNC system

a> GSK25iCNC and PC are connected by cable.

b> Correctly set CNC parameter and IP address (IP address of CNC parameter 130# should be the same as the one set by transmission software and they should in the same network segment). Network communication function of CNC end is used.

c> Select Connect to CNC on the system menu Connection setting , then enter communication setting interface. Refer to the following graph. Connect to the CNC system by selecting proper connection mode.

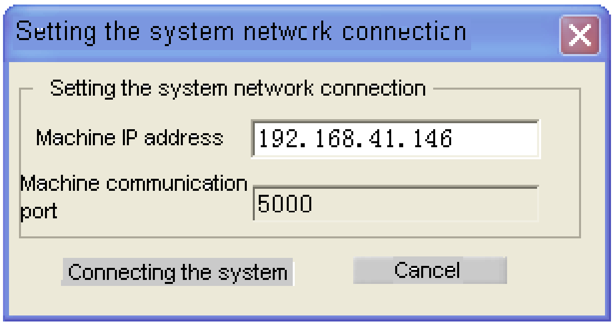


Fig. 9-1-2

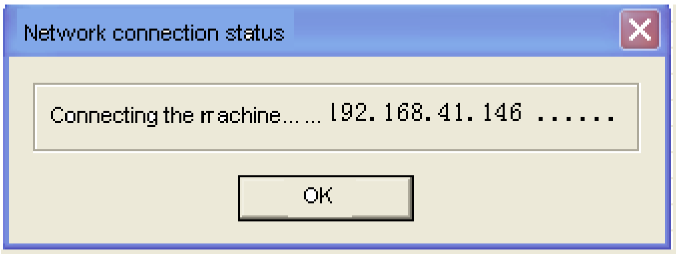


Fig. 9-1-3

d> Normal connection appears on the title column after connecting the system.

e> Cut off the connection. If DNC transmission is performing when executing Cut off the connection or Exit, the system may execute reset operation firstly, then cut off the connection to stop transmission avoiding serious results. When net cable is disconnected and IO channel is in network mode, NC code is cleared by 【Reset】operation in transmission, namely, NC transmission shall be done again after reset.

2. Network DNC

a> As long as IO channel is correctly set in CNC (set parameter 10# to 4), on-line processing and upload or down load through DNC can be performed. Refer to GSK25i system parameter manual.

b> Click the button of “Open file” on PC software, and select the NC file of DNC machining to be performed. After confirming, the file content may display on its software and it is read only, the file path name may display at the left side of the button. When file name reaches or exceeds 28 characters (character number includes suffix .NC). DNC software displays: file name is too long.

c> Click “DNC” transmission and the network data transmission begins to perform. Transmission is allowed when the system is in DNC mode at non-alarm state, or prompting information.

d> Press the button of “Cycle start” on CNC panel and the machining begins to perform.

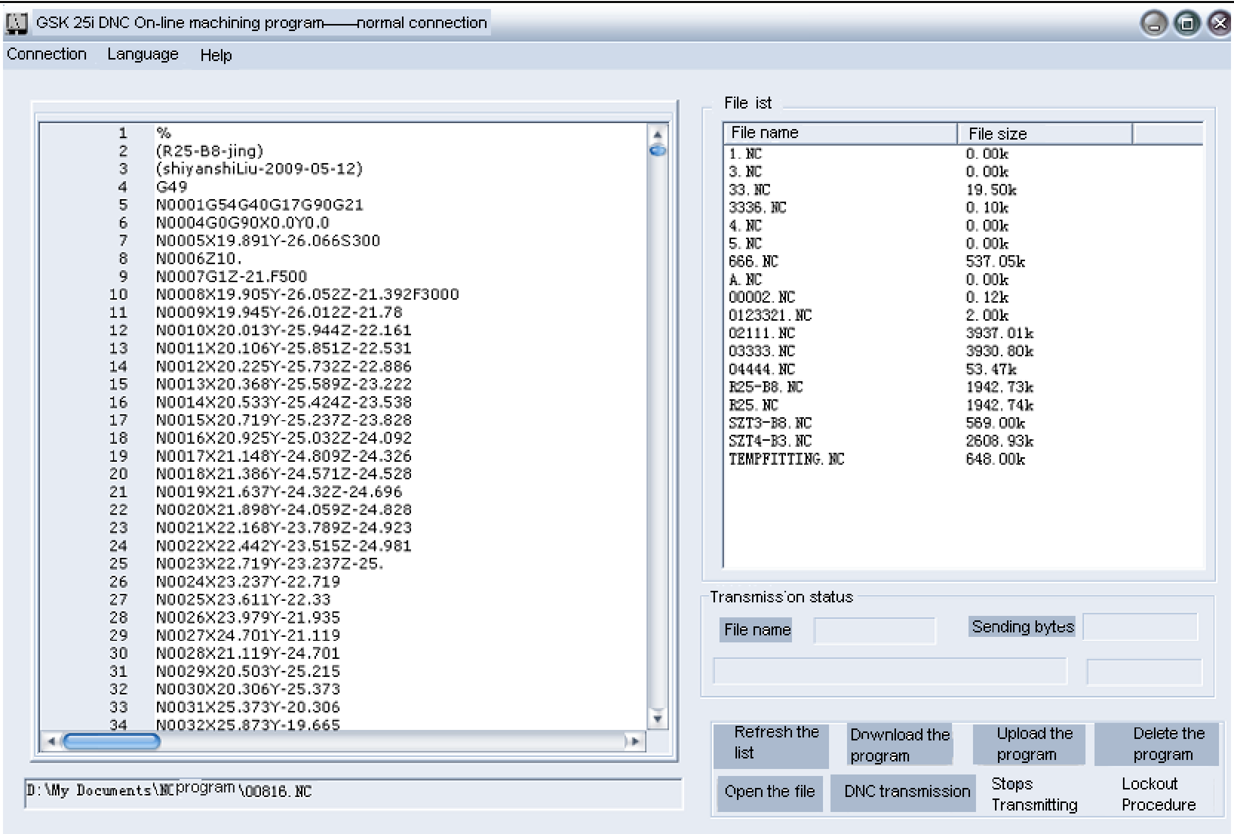


Fig. 9-1-4

Notes:

1. If current uploaded G code is deleted in transmission, PC software does not response and system alarms "Current file can not be deleted".

2. Parameters #10 and 130# can not be changed in network DNC.

3. If DNC is stopped in the process of network DNC, transmit other files after machine reset.

4. DNC operation stops when codes M99 and M30 occur. Contents on the screen are cleared.

5. Main program calls subprogram in DNC, P address is ignored when subprogram contains M99P.

6. Program transmission and deletion in network DNC are invalid when transmission software is using for file download or upload.

7. In network DNC,【Download】,【Upload】,【Deletion】 operations can not be done for the program loaded in 【Edit】mode. Otherwise, alarm occurs on PC : the program is uploaded. Transmission is not allowed.

3. File transmission

a> Obtain file list

Click the button of Refreshing list and the file list on the CNC system can be gained in the case of the software and CNC terminal are well connected. File list refreshing is not allowed in network DNC operation.

b> Upload file

Based upon the normal connection between the software and CNC, select the NC file to be uploaded by clicking "upload program", and then the file can be uploaded after confirming.

Note:

1. The name of the uploaded file should not the same as the one opened in the transmission software.

2. If transmission speed is slow when a big file is transmitted, wait a moment and it becomes smooth.

3. File upload is not allowed in network DNC operation.

c> Download file

Based upon the normal connection between the software and CNC, select the file to be downloaded in machine NC file list and then the file can be downloaded by clicking the button of **Download program**. The file download is not performed during network DNC operation.

Warning:

During communication between PC and CNC system (including file upload, file download, CNC transmission):

1）IP address can not be changed, otherwise, system alarm occurs. The operation is not allowed in the current mode.

2） Do not pull out the cable, otherwise, serious results may occur.

3）The file loaded by the current CNC can not be deleted by the transmission software, and CNC system alarm “The current file can not be deleted”

4） File list operation is not allowed during file transmission.

9.1.3 DNC Processing Function

Realize DNC machining function by communication between CNC and USB. Procedures are as follows:

1. Set IO channel (No. 10) to 3 based on the parameter manual.

2. Upload external memory in Edit mode, then upload program in DNC mode to perform the operation.

Warning:

1. The difference between DNC and Auto mode is: single-upload is used in Auto mode. Only NC programs within 8M can be loaded for the system limitation, while NC program up to 200M can be performed in DNC.

2. Skip command is not allowed in DNC mode.

3. A DNC communication software connects to one CNC only in a PC machine. Do not connect several DNC communication softwares in PC machine to multiple CNC systems by using concentrator or exchanger.

4. After NC program in USB is uploaded, in order to make CNC system clean relative resources, please download external memory before pulling out USB to avoid serious results.

9.2 USB disk Communication

9.2.1 Transmit between USB disk and CNC

Some authority requirements about transmitting between USB disk and CNC, and operation details please refer to Table 9-1.

Table 9-1

Conditions		Operations: CNC to USB disk	Operations: USB disk to CNC
Single program transmit（Program name <O8000）	Authority Requirements	No	No
	CNC work mode	DNC or EDIT mode necessarily	DNC or EDIT mode necessarily
	USB disk transmission requirement	USB disk loaded	USB disk loaded
	Program name is unique	Transmit directly	Transmit directly
	Program name is already used	Popup prompt to choose replace or cancel	Popup prompt to choose replace or cancel
Single program transmitting（Program name O8000～O8999）	Authority Requirements	Terminal management or higher level	Terminal management or higher level
	CNC work mode	DNC or EDIT mode necessarily	DNC or EDIT mode necessarily
	Parameter setting	Para 1610.0=0	Para 1610.0=0
	USB disk transmission requirement	USB disk loaded	USB disk loaded
	Program name is unique	Transmit directly	Transmit directly
	Program name is already used	Popup prompt to choose replace or cancel	Popup prompt to choose replace or cancel
Single program（Program name O9000-O9999）	Authority Requirements	Terminal management or higher level	Terminal management or higher level
	CNC work mode	DNC or EDIT mode necessarily	DNC or EDIT mode necessarily
	Parameter setting	Para 1610．4=0	Para 1610．4=0
	USB disk transmission requirement	USB disk loaded	USB disk loaded
	Program name is unique	Transmit directly	Transmit directly
	Program name is already used	Popup prompt to choose replace or cancel	Popup prompt to choose replace or cancel

From table 9-1 it can be learned that CNC transmits all the part programs, however, when the program name is equal or more than 8000, related parameters must be set before transmitting and during the transmitting if the program name has already been used, CNC will pop up prompt to choose, replace or cancel.

Note: 1. If USB disk is not correctly unloaded, the operations to USB disk are possible invalid.

2. During loading, reading and writing, it is not allowed to insert or pull out USB disk; otherwise, it will affect CNC performing normally.

9.2.2 Ladder files transmit between USB disk and CNC

Ladder files transmitting between current USB disk catalog and current CNC catalog is shown as the following table 9-2:

Table 9-2

Conditions		Operations	
		CNC to USB disk	USB disk to CNC
Authority requirements		Tool manufacture password or higher level	Tool manufacture password or higher level
CNC work mode		Any mode	Any mode
USB disk transmission requirement		USB disk loaded	USB disk loaded
Transmit ladder files	Program name is unique	Transmit directly	Transmit directly
	Program name is already used	Replace with no prompt	Replace with no prompt

From table 9-1 it can be learned that tool manufacture password or higher level is asked to transmit between USB disk catalog and CNC current catalog.

Ladder files transmitting can directly perform when the program name is unique but if the program name has already been used then CNC will replace the origin without prompt.

Note：1. If USB disk is not correctly unloaded, the operations to USB disk are possible invalid.

2. During loading, reading and writing, it is not allowed to insert or pull out USB disk; otherwise it will affect CNC performing normally.

9.2.3 Screenshot function

It should be in EDIT mode that load the USB disk, select “load external storage”, and the interface will display “SD card loaded”. If it displays “Can’t carry out in this status”, then you should check the operation mode. After USB disk loaded, press “Replace” key about 8 sec, then the interface will display “Successes”.

When unload USB disk, select “unload external storage”, and the interface will display “SD card unloaded”. Image screenshot is saved in file “25i-screen”.

Note：1. If USB disk is not correctly unloaded then the operations to USB disk is possible invalid.

APPENDIX

Appendix Ⅰ G CODE PROGRAMMING RULES

1. G codes from different groups can be commanded in the same block. And if in one block several G codes from a group are commanded then only the last commanded one is executed.
Example: N10 G00 G01 X10.; G01 valid
N11 G00 G02 I5.; G02 valid N12 G90 X10. G91 X20.; G91 valid

2. If modal G and non-modal G are in the same block, then the non-modal G is prior to be executed.

3. In G02 and G03, when address I, J, K and R are commanded, arc R address commanded is prior.
Example: G02 I10. R20; R valid, I invalid

4. When many address instructions from X,Y,Z,A,B,C,I,J,K,R,U,V,W,P,L,H,D,Q,E,O,N,S are commanded then, the last one is valid..
Example: N12 G90 X10. G91 X20.; G91valid X20.valid, but X10 in head is invalid.

5. Corresponding drill data is needed at the beginning of canned cycle, without, no drilling but positioning to the initial point (Example as following).
Example: G73 X100. Y200.; Z R Q Set no drilling parameter, so CNC does not drill but rapid position to（100. 200.）
G80;

6. The establish and cancel of cutter compensation is valid in the block which contains（G00 /G01）command while block containing G02/ G03 will lead to #34 alarm.

7. Macro skip command and cycle command should be written in a line alone; otherwise, alarm #129 occurs.

8. M codes like M00 M01 M05 M09 M30 and M02 will be executed later than other codes when they are in the same line.
Example: G83 X100. Z-50. R5. Q2. M5 M9; in this block M05 M09 is executed after drilling is finished.

9. When G28, G30 or G27 in the coordinate rotate/polar coordinate/scaling/programmable mirror image, CNC will alarm; in G51.1, G51, G68 modal, commands G27～G30,G52～G59 and G92 are illegal.

10. In AUTO, when running the program to M30, then the running will stop and return to the beginning of the block; to M02, then the running will stop and do not return to the beginning of the block. In MDI, meeting either of them, the running will be back to the beginning. %means program end and do not return to the beginning of the block.

11. When 25i CNC alarms or resets, no matter the current block runs to the end or not, the cursor will stay at the current block.

12. Specifically stress that if the appearance, weariness, workpiece coordinate system, macro variable or partly parameters, are manually amended when CNC is running the program, will not take into effect immediately. So it can't too much to emphasize that after modifying it is better to reset and move cursor to corresponding G code and run again.
The current coordinate system is rewritten in Manual mode or by G10, the absolute coordinates and relative coordinates will not change and refresh at once, but it refreshes during running.

13. Macro variable will still display null when calculation results overflow at the same time the CNC will alarm that the data is overflow. Macro variable 500—999 are the global variables which initially are 0.

14. About skip / when lighten the skip signal, it must be done 5 blocks before the block going to skip, otherwise the signal might possible be missed.

15. When call the tool change program, G16 G68 G51 and G51.1 are suggested to be cancele

Appendix Ⅱ ALARM LIST

Alarm list

System alarm (PS alarm)			
Alarm NO.	2	Alarm message	After rewriting the parameters, it can be continuously operated after restarting.
Analyzing	It occurs after rewriting some system parameters. such as 200-206, 800, 811, 1000, 1004, 2170-2172, 2800 and 2806.		
Troubleshooting	It can be continuously operated after restarting.		
Alarm NO.	3	Alarm message	Too many bits in the digit
Analyzing	The data in the program exceed the range of the system allowable value. For example, the maximum value of the coordinate value is X999999.9999; the system alarms if it exceeds the value.		
Troubleshooting	Check the data in the program, and change the digit with too many bits into the proper value.		
Alarm NO.	4	Alarm message	Fail to find the address
Analyzing	At the head of the block, the digits or the illegal characters are input.		
Troubleshooting	Check the corresponding block and modify the program.		
Alarm NO.	5	Alarm message	Without data after the address
Analyzing	In the program, the corresponding data aren't written in after the address/command. For example, G, X, Y, Z, M command occurs independently.		
Troubleshooting	Check each address/command in the program; complete the data if some commands miss some data.		
Alarm NO.	6	Alarm message	Illegally use the negative sign
Analyzing	Many negative signs are input or the negative sign is input in the position in which it can't be used.		
Troubleshooting	The program should be modified.		
Alarm NO.	7	Alarm message	Illegally use the decimal point

Analyzing	The redundant decimal point is in the program or the decimal point is used in the position in which it can't be used.		
Troubleshooting	Check the program to delete the redundant decimal point.		
Alarm NO.	9	Alarm message	Input the illegal character address
Analyzing	The illegal character address can't be used.		
Troubleshooting	Check the program and rewrite the corresponding address.		
Alarm NO.	10	Alarm message	Incorrect G codes
Analyzing	In the program, the system specifies G codes which don't exist or can't be used.		
Troubleshooting	Check the program and rewrite the wrong G codes into the correct ones.		
Alarm NO	11	Alarm message	Without the feedrate command
Analyzing	During cutting feed, the feedrate isn't specified or the feedrate isn't proper.		
Troubleshooting	The program should be modified.		
Alarm NO.	12	Alarm message	Address P is repeatedly used and the program is rewritten.
Analyzing	In one block, two or more P addresses occur.		
Troubleshooting	The address P commands are divided into different blocks which can use address P.		
Alarm NO.	13	Alarm message	The single direction positioning can't be specified in the tool compensation mode.
Analyzing	In the block setting the tool compensation includes the single direction positioning command G60 meanwhile.		
Troubleshooting	The program should be modified because G60 can't be written with the tool compensation command in the same line.		
Alarm NO.	14	Alarm message	G10 command format is wrong.
Analyzing	The system parameter (G10) is rewritten on-line because the programming format is wrong.		
Troubleshooting	Modify the program and change G10 format.		
Alarm NO.	15	Alarm	Too many axes are commanded.

		message	
Analyzing	In the program, the system has specified the axis which doesn't exist or not set.		
Troubleshooting	Check the program to delete the redundant axis command or check the system parameter 1020 (check whether the system parameter 800 setting is correct or not), and check whether the corresponding axis name is right or not.		
Alarm NO.	16	Alarm message	It's not allowed to rewrite the parameters of this group in G10.
Analyzing	The parameters which can be rewritten include: 1, 130, 1020-1021, 1031-1053, 1605-1642, 1801-1930, 1933-2034, 2112, 2113 and 2600-2653，and the system alarms if the parameters except for the above ones are specified in the program.		
Troubleshooting	Modify the program to delete the parameter number which can't be rewritten by G10 command.		
Alarm NO.	18	Alarm message	The wrong plane is selected during commanding the plane arc in the length compensation.
Analyzing	In the program, when the length compensation is set, the wrong plane on which the arc path exists is chosen.		
Troubleshooting	The program is modified to select the correct plane in which the arc command is used.		
Alarm NO.	19	Alarm message	Too many axes commanded.
Analyzing	The commanded axis linkage number exceeds the limit in one block.		
Troubleshooting	The program should be modified.		
Alarm NO.	20	Alarm message	Exceed the radius tolerance
Analyzing	In the arc interpolation, the distance from the start position to the end one on radius R can't compose arc or the distance from the start position to the arc center is different with that from the end position to the arc center.		
Troubleshooting	The program should be modified.		
Alarm NO.	21	Alarm message	The illegal plane axis is commanded.
Analyzing	In the arc interpolation, the axis not on the planes (G17 G18 G19) is specified.		
Troublesho	The program should be modified.		

oting			
Alarm NO.	22	Alarm message	Without arc radius
Analyzing	In the program, the arc radius isn't specified in the position in which it should be used.		
Troubleshooting	In the program, the arc radius command is added.		
Alarm NO.	24	Alarm message	The input data exceeds the system specified range.
Analyzing	G10 input data exceeds the system range.		
Troubleshooting	The program should be modified.		
Alarm number	25	Alarm message	The pitch error compensation number is illegal or doesn't exist or the pitch error compensation value isn't specified.
Analyzing	When the pitch error compensation is input with G10, the specified pitch error compensation number doesn't exist or is illegal, and the pitch error compensation value isn't specified.		
Troubleshooting	The program should be modified.		
Alarm NO.	26	Alarm message	G51 can't share the same block with other commands.
Analyzing	G51 isn't allowed to share the same block with G10 and G65.		
Troubleshooting	The program should be modified and written in separate lines.		
Alarm NO.	27	Alarm message	The parameter number rewritten by G10 is illegal or doesn't exist.
Analyzing	The parameters specified by G10 don't exist in the system or parameter numbers aren't correct.		
Troubleshooting.	The program should be modified.		
Alarm NO.	28	Alarm message	Illegal plane selection
Analyzing	The plane switch can't be operated in canned cycle mode.		
Troubleshooting	Modify the program: switch the plane after cancelling the canned cycle or rewrite #2000.0 bit parameter.		

Alarm NO.	29	Alarm message	Illegal offset value
Analyzing	The compensation value specified by H is too big or doesn't exist.		
Troubleshooting	The program should be modified.		
Alarm NO.	30	Alarm message	Illegal compensation number
Analyzing	The compensation number specified by D/H code is too big or doesn't exist. Moreover, the value specified by the additional workpiece coordinate system number commanded by P code exceeds the system range.		
Troubleshooting	The program should be modified.		
Alarm NO.	31	Alarm message	Command illegal P in G10.
Analyzing	The data specified by address P aren't correct in G10 command.		
Troubleshooting	Modify the program and check whether the relative data numbers which are rewritten on-line are correct or not.		
Alarm NO.	32	Alarm message	Illegal compensation value in G10
Analyzing	When the offset value is set with G10 or the system variable is rewritten into the offset value, the offset value exceeds the range specified by the system.		
Troubleshooting	The program should be modified.		
Alarm NO.	33	Alarm message	The tool compensation doesn't obtain an intersection.
Analyzing	In the tool compensation, the end position coordinate of the current block is not in the line of the next block.		
Troubleshooting	Modify the program or check the tool compensation value.		
Alarm NO.	34	Alarm message	When the arc is commanded, the tool can't be started or the tool compensation is cancelled.
Analyzing	When the arc is commanded, the tool compensation can't be set or cancelled.		
Troubleshooting	The program should be modified.		
Alarm number	36	Alarm message	G31 and G37 can't be commanded in tool compensation mode.

Appendix

Analyzing	G31 and G37 commands are not allowed to use in tool compensation mode.		
Troubleshooting	Check whether G31 or G37 command exists in the tool compensation program; if it exists, the program should be modified.		
Alarm NO.	37	Alarm message	The plane is switched during the tool compensating
Analyzing	The planes (G17 G18 G19) are not allowed to switch during the tool compensating.		
Troubleshooting	Check whether the plane is switched or not in the tool compensation program; if it is switched, the program should be modified.		
Alarm NO.	38	Alarm message	There is interference in the arc block
Analyzing	Overcutting occurs in the cutter compensation mode because the arc start position or the end one is same as the arc center.		
Troubleshooting	The program should be modified.		
Alarm NO.	39	Alarm message	The arc end position is not in arc after tool compensation.
Analyzing	The arc is commanded after the tool compensation; if the end position is not in the arc, whether the system alarms is set by parameter 1810.		
Troubleshooting	The program should be modified.		
Alarm NO.	40	Alarm message	The tool compensation amount is rewritten in the arc interpolation.
Analyzing	The radius compensation amount isn't allowed to rewrite in the cutter compensation mode.		
Troubleshooting	Check whether D value is rewritten the compensation amount in the program.		
Alarm NO.	41	Alarm message	There is interference in CRC.
Analyzing	Overcutting occurs in the cutter compensation mode.		
Troubleshooting	The program should be modified.		
Alarm NO.	42	Alarm message	G45-G48 aren't allowed to command during tool compensating
Analyzing	G45-G48 are commanded in the tool compensation mode.		
Troubleshooting	The program is modified to delete G45-G48 commands.		
Alarm NO.	43	Alarm	Radius value excess-error

Appendix

		message	
Analyzing	If the end position is not on the arc, whether the system alarms is set by parameter 1810.		
Troubleshooting	The program should be modified.		
Alarm NO.	44	Alarm message	In the canned cycle, the commands G27－G30 and G53 are not allowed to command.
Analyzing	In the canned cycle, commands G27-G30 or G53 exist.		
Troubleshooting	The program should be modified.		
Alarm NO.	45	Alarm message	Fail to find address Q in the canned cycle.
Analyzing	In the canned cycle G73/G83, the cutting depth (Q) each time isn't specified.		
Troubleshooting	The program should be modified.		
Alarm NO.	46	Alarm message	The illegal reference position return command
Analyzing	In the 2nd, the 3rd and the 4th reference position return commands, the commands except for P2, P3 and P4 are specified.		
Troubleshooting	The program should be modified.		
Alarm NO.	47	Alarm message	G10 can't be used with the canned cycle meanwhile
Analyzing	G10 command is used in the canned cycle.		
Troubleshooting	Delete G10 command in the canned cycle or the canned cycle must be cancelled before executing G10.		
Alarm NO.	48	Alarm message	The switch command format of metric/inch systems is not correct.
Analyzing	The switch between the metric/inch systems can only be executed at the head of the program. It's not allowed to switch during executing the program or the subprogram.		
Troubleshooting	The program should be modified.		
Alarm NO.	51	Alarm message	Wrongly move after CHF / CNR.
Analyzing	During chamfering, the values of R and C are specified wrongly.		

Appendix

Troubleshooting	The program should be modified.		
Alarm NO.	53	Alarm message	The type can't be judged or doesn't exist during switching the type.
Analyzing	Fail to judge the four types（L_L， L_C， C_L， C_C）specified by the system.		
Troubleshooting	The program should be modified.		
Alarm NO.	54	Alarm message	After the tool compensation is set, there are no relative plane movement commands in the consecutive 8 blocks.
Analyzing	In the cutter compensation, no movement commands are specified in the relative planes in more than 8 blocks.		
Troubleshooting	The program should be modified.		
Alarm NO.	55	Alarm message	The block number is less than 2 from setting to canceling the tool compensation.
Analyzing	After setting the tool compensation, the specified move commands can't form two blocks, the tool compensation is cancelled.		
Troubleshooting	The program should be modified.		
Alarm NO.	64	Alarm message	Illegal M code command
Analyzing	The specified M code exceeds the range or too many M codes are specified in one block.		
Troubleshooting	The program should be modified.		
Alarm NO.	65	Alarm message	The block is too long.
Analyzing	The block exceeds the maximum unit number 32.		
Troubleshooting	The program should be modified.		
Alarm NO.	66	Alarm message	The character string of some unit is too long
Analyzing	The character string of one unit exceeds the maximum character number.		
Troubleshooting	The program should be modified.		
Alarm NO.	67	Alarm	Illegal sequence number

Appendix

		message	
Analyzing	N sequence number exceeds the system range.		
Troubleshooting	The program should be modified.		
Alarm NO.	68	Alarm message	P/X dwell time is illegal or overtime.
Analyzing	The dwell time exceeds the system range.		
Troubleshooting	The program should be modified.		
Alarm NO.	72	Alarm message	The program calling or M99 skip is wrong, and the code should be rewritten.
Analyzing	In the same block, there are too many called subprogram commands or M99 is used in DNC.		
Troubleshooting	The program should be modified.		
Alarm NO.	74	Alarm message	Illegal program number
Analyzing	The initial letter of the program number isn't '0" or exceeds the maximum range.		
Troubleshooting	The program should be modified.		
Alarm NO.	75	Alarm message	Fail to operate, without the authority.
Analyzing	Because the authority of rewriting the system parameter is not enough, please input the password.		
Troubleshooting	Input the password.		
Alarm NO.	76	Alarm message	Address P isn't defined or illegal.
Analyzing	The block of M98, G65 or G66 is lack of address P.		
Troubleshooting	The program should be modified.		
Alarm NO.	77	Alarm message	The subprogram nesting is wrong.
Analyzing	The subprogram calling times are too many.		
Troubleshooting	The program should be modified.		

Alarm NO.	78	Alarm message	Fail to find the corresponding program number or the sequence number.
Analyzing	In M98, M99, G65 or G66 block, fail to find the program number and the sequence number specified by address P. Fail to find the sequence number specified by GOTO statement or the called program is edit in the background editing mode.		
Troubleshooting	Modify the program or end the background editing.		
Alarm NO.	79	Alarm message	The turning machine and the milling one can't be directly switched.
Analyzing	The turning machine and the milling one can be switched during the system setting.		
Troubleshooting	The program should be modified.		
Alarm NO.	81	Alarm message	Fail to find the compensation number in G37.
Analyzing	Fail to specify the tool compensation number before executing G37 command.		
Troubleshooting	The program should be modified.		
Alarm NO.	82	Alarm message	It's not allowed to specify H code in G37.
Analyzing	H code and G37 command can't be specified in one line.		
Troubleshooting	The program should be modified.		
Alarm NO.	83	Alarm message	Illegal axis command in G37.
Analyzing	In automatic tool length measuring, the invalid axis or the incremental value is specified.		
Troubleshooting	The program should be modified.		
Alarm NO.	84	Alarm message	No arrival signal in G37.
Analyzing	In automatic tool compensation function, the measuring position arrival signal is output out of the range specified by the parameters.		
Troubleshooting	The setting and operation should be correct.		
Alarm NO.	86	Alarm message	Errors in the 2nd stored stroke detection border specified by G22

Analyzing	The negative limit value is bigger than the positive one in the 2nd stroke detection.		
Troubleshooting	The program should be modified.		
Alarm NO.	90	Alarm message	Illegal G107 command.
Analyzing	Setting or cancelling the cylinder interpolation condition isn't correct.		
Troubleshooting	The program should be modified.		
Alarm NO.	91	Alarm message	The codes not allowed to use are specified in G107.
Analyzing	In the cylinder interpolation mode, any G codes below can't be commanded: 1. Positioning G codes, such as G28, G73, G74, G76 and G81-G89, include the codes specified during the rapid movement cycle. 2. Setting G codes of the coordinate system: G52 and G92. 3. Selecting G codes of the coordinate system: G53 and G54-G59		
Troubleshooting	The program should be modified.		
Alarm NO.	92	Alarm message	G107 doesn't end normally or the tool compensation isn't cancelled normally.
Analyzing	G107 cylinder interpolation doesn't end normally.		
Troubleshooting	The program should be modified.		
Alarm NO.	93	Alarm message	The canned cycle command can't be specified in G05.
Analyzing	The canned cycle command is used in the program in which G50 command is valid.		
Troubleshooting	The program should be modified.		
Alarm NO.	94	Alarm message	It's not allowed to call the program in G66 mode.
Analyzing	It's not allowed to call the program in G66 mode.		
Troubleshooting	The program should be modified.		
Alarm NO.	100	Alarm message	The positioning spindle address and other axis movement command are in one block.
Analyzing	The positioning spindle address set by the parameters can't be with the other axis movement address in one block.		

Troubleshooting	The program should be modified.		
Alarm NO.	101	Alarm message	The specified data exceed the valid range.
Analyzing	The relative data exceed the range specified by the system.		
Troubleshooting	The program should be modified.		
Alarm NO.	102	Alarm message	The thread retraction length is longer than the axis thread machining one.
Analyzing	The retraction length is longer than the thread machining one.		
Troubleshooting	The program should be modified.		
Alarm NO.	103	Alarm message	Illegal thread command
Analyzing	The thread tooth number in inch system is 0 or the gear number is too big.		
Troubleshooting	The program should be modified.		
Alarm NO.	104	Alarm message	G73 program start position exists the interference.
Analyzing	The interference is in G73 machining path.		
Troubleshooting	The program should be modified.		
Alarm NO.	106	Alarm message	R specified by the canned cycle G90 G92 G94, the end position and the start one can’t close.
Analyzing	The programmed path isn’t closed.		
Troubleshooting	The program should be modified.		
Alarm NO.	107	Alarm message	The direct drawing dimension programming command format is wrong.
Analyzing	The drawing dimension programming format is wrong.		
Troubleshooting	The program should be modified.		
Alarm NO.	111	Alarm message	Calculated data overflow

Analyzing	The calculation result is out of the range specified by the system.		
Troubleshooting	The program should be modified.		
Alarm NO.	112	Alarm message	Divided by zero
Analyzing	Division by zero is specified (including tan90°)		
Troubleshooting	The program should be modified.		
Alarm NO.	113	Alarm message	Improper macro command.
Analyzing	A function which can't be used in custom macro is commanded.		
Troubleshooting	The program should be modified.		
Alarm NO.	114	Alarm message	Error in macro expression format
Analyzing	Error is in macro expression format.		
Troubleshooting	The program should be modified.		
Alarm NO.	115	Alarm message	Illegal variable number is specified.
Analyzing	The macro variable 3000 substitution expression exceeds the range.		
Troubleshooting	The program should be modified.		
Alarm NO.	116	Alarm message	Without operation number in macro expression
Analyzing	The valid operation number should be filled in the macro expression.		
Troubleshooting	The program should be modified.		
Alarm NO.	118	Alarm message	Macro parenthesis nesting error
Analyzing	The nesting of bracket in the macro exceeds the upper limit.		
Troubleshooting	The program should be modified.		
Alarm NO.	119	Alarm	Illegal macro variable number

		message	
Analyzing	In the user macro, the values which can't be used as the variable number are used.		
Troubleshooting	The program should be modified.		
Alarm NO.	123	Alarm message	The macro command is used in DNC.
Analyzing	There exists the macro in the program of DNC machining.		
Troubleshooting	The program should be modified.		
Alarm NO.	124	Alarm message	DO-END doesn't correspond to each other.
Analyzing	In macro statement, DO-END doesn't correspond to each other.		
Troubleshooting	The program should be modified.		
Alarm NO.	126	Alarm message	Illegal cycle number
Analyzing	The cycle number behind DO-END is wrong.		
Troubleshooting	The program should be modified.		
Alarm NO.	127	Alarm message	NC command and the macro command in one block
Analyzing	NC command and macro command statements are co-existed.		
Troubleshooting	The program should be modified.		
Alarm NO.	128	Alarm message	Illegal macro program sequence number
Analyzing	The sequence number in the branch command exceeds the range specified by the system or can't be searched.		
Troubleshooting	The program should be modified.		
Alarm NO.	129	Alarm message	Macro skip command can't be with the other commands in one line.
Analyzing	Macro skip command and other commands are in one block.		
Troubleshooting	The program should be modified.		

Alarm NO.	135	Alarm message	Illegal angle command.
Analyzing	The index table indexing positioning angle was commanded in other than an integral multiple of the value of the minimum angle.		
Troubleshooting	The program should be modified.		
Alarm NO.	136	Alarm message	Illegal axis command
Analyzing	In index table indexing, another control axis was commanded together with B axis.		
Troubleshooting	The program should be modified.		
Alarm NO.	140	Alarm message	The wrong block address specified in multiple cycles
Analyzing	The starting block number is bigger than the end one or the specified block number doesn't exist.		
Troubleshooting	The program should be modified.		
Alarm NO.	141	Alarm message	Incorrect G (or M) code is specified in multiple cycles.
Analyzing	The G codes not allowed to specify are commanded in multiple cycles.		
Troubleshooting	The program should be modified.		
Alarm NO.	142	Alarm message	Too many blocks are specified in multiple cycles.
Analyzing	In multiple cycles, the specified block number exceeds the range specified by the system.		
Troubleshooting	The program should be modified.		
Alarm NO.	143	Alarm message	The coordinate monotonous error in multiple cycles
Analyzing	X or Z monotonous error (increase or decrease in single direction) is in multiple cycles.		
Troubleshooting	The program should be modified.		
Alarm NO.	144	Alarm message	In multiple cycles, error in the 1st block format
Analyzing	In multiple cycles, the illegal codes are specified in the 1st block.		

Appendix

Troubleshooting	The program should be modified.		
Alarm NO.	145	Alarm message	In multiple cycles, the code format is wrong.
Analyzing	In multiple cycles, the machining path enters the abnormal state.		
Troubleshooting	The program should be modified.		
Alarm NO.	146	Alarm message	The machining path graphic is wrong.
Analyzing	In multiple cycles, the machining path graphic is wrong.		
Troubleshooting	The program should be modified.		
Alarm NO.	147	Alarm message	The specified arc can't be roughed.
Analyzing	The arc command is not allowed to be specified during roughing.		
Troubleshooting	The program should be modified.		
Alarm NO.	148	Alarm message	P/Q value (0 is specified) isn't specified in G74/G75.
Analyzing	The specified P/Q is illegal in G74/G75.		
Troubleshooting	The program should be modified.		
Alarm NO.	149	Alarm message	In G74/G75, the retraction amount is bigger than the feeding one.
Analyzing	In G74/G75, the retraction amount is bigger than the feeding one.		
Troubleshooting	The program should be modified.		
Alarm NO.	154	Alarm message	G51.1,G51 and G68 can't be in the block.
Analyzing	G51.1, G51 and G68 can't be in the block.		
Troubleshooting	The program should be modified.		
Alarm NO.	155	Alarm message	G51 can only be with P parameters in G110-G137 modal.

Analyzing	In G110-G137 special canned cycle modal, G51 command can only be with P parameters.		
Troubleshooting	The program should be modified.		
Alarm NO.	156	Alarm message	G27－G30,G52－G59,G92 and G10L20 are not allowed to command in G51.1,G51 and G68 modes.
Analyzing	G27－G30,G52－G59,G92 and G10L20 are not allowed to command in G51.1,G51 and G68 modes.		
Troubleshooting	The parameters should be modified.		
Alarm NO.	157	Alarm message	G68.4 vector is 0.
Analyzing	G68.4 vector length is 0.		
Troubleshooting	The program should be modified.		
Alarm NO.	158	Alarm message	The machine type doesn't support.
Analyzing	The machine type doesn't support.		
Troubleshooting	The parameters 8010 should be rewritten.		
Alarm NO.	160	Alarm message	Level is specified wrongly.
Analyzing	NURBS interpolation level is wrong.		
Troubleshooting	P value is rewritten.		
Alarm NO.	161	Alarm message	The knot isn't specified.
Analyzing	The knot isn't specified in NURBS interpolation.		
Troubleshooting	The program should be modified.		
Alarm NO.	162	Alarm message	The knot is specified wrongly.
Analyzing	The knot range specified by NURBS interpolation is wrong.		
Troubleshooting	The program should be modified.		

Alarm NO.	163	Alarm message	Too many axis numbers are specified.
Analyzing	The redundant axes are specified in NURBS interpolation.		
Troubleshooting	The program should be modified.		
Alarm NO.	164	Alarm message	The interpolation mode not allowed to use with NURBS interpolation meanwhile is specified.
Analyzing	The interpolation mode not allowed to use with NURBS interpolation meanwhile is specified.		
Troubleshooting	The program should be modified.		
Alarm NO.	165	Alarm message	Fail to find G68.2.
Analyzing	When G53.1 is specified, G68.2, G68.3 and G68.4 modals are not found.		
Troubleshooting	The program should be modified.		
Alarm NO.	166	Alarm message	In G124/G125, E or K isn't defined or is 0.
Analyzing	In G124/G125, the drilling number E on the 1st and the 3rd borders or the drilling number K on the 2nd and the 4th borders isn't defined or is 0.		
Troubleshooting	The program should be modified.		
Alarm NO.	167	Alarm message	Program codes in one block aren't reasonable, so they should be written in separate blocks.
Analyzing	The program codes not allowed to co-exist in one block are used.		
Troubleshooting	The program should be modified.		
Alarm NO.	170	Alarm message	The canned cycle punching modes G73-G89 should be defined.
Analyzing	During G120-G12 continuous drilling cycle, the canned cycle punching modes G73-G89 aren't defined		
Troubleshooting	During G120-G125 continuous drilling cycle, the canned cycle punching modes G73-G89 should be added.		
Alarm NO.	171	Alarm message	I isn't defined or I is 0.

Appendix

Analyzing	In G110-G137special canned cycles, I isn't defined or I is 0.		
Troubleshooting	Modify the program to add 1 or I isn't 0.		
Alarm NO.	172	Alarm message	J isn't defined or J is 0.
Analyzing	In G110-G137special canned cycle, J isn't defined or J is 0.		
Troubleshooting	Modify the program: add J or J isn't 0.		
Alarm NO.	173	Alarm message	W is too small or W isn't defined.
Analyzing	In rough milling cycle, W isn't defined or W is 0.		
Troubleshooting	Modify the program: add W or W isn't 0.		
Alarm NO.	174	Alarm message	Q is too small or Q isn't defined.
Analyzing	In rough milling cycle, Q isn't defined or Q is 0.		
Troubleshooting	Modify the program: add Q or Q isn't 0.		
Alarm NO.	175	Alarm message	L is too small or L isn't defined.
Analyzing	L isn't defined or L is too small in G110-G137 special canned cycle and G126 plane milling.		
Troubleshooting	Modify the program: add L or increase L value.		
Alarm NO.	176	Alarm message	V is too small or V isn't defined.
Analyzing	In rough milling cycle, V isn't defined or V is 0.		
Troubleshooting	Modify the program: add V or V isn't 0.		
Alarm NO.	178	Alarm message	I or J is too small or the tool radius is too big.
Analyzing	In G110-G137 special canned cycle, I or J is too small or the tool radius is too big.		
Troubleshooting	Modify the program: increase I or J or change the tool.		
Alarm NO.	179	Alarm	L is too big.

Appendix

		message	
Analyzing	During rough milling, L is too big.		
Troubleshooting	Modify the program to make L smaller.		
Alarm NO.	180	Alarm message	U is less than D.
Analyzing	During rectangle milling rough cycle, U is bigger than D.		
Troubleshooting	Modify the program to make U less or equal to D.		
Alarm NO.	181	Alarm message	In the special canned cycle, the punching number is 0 or doesn't exist.
Analyzing	During G120-G125 continuous drilling, the punching number is 0 or doesn't exist.		
Troubleshooting	Modify the program: add the punching number parameter or it is not 0.		
Alarm NO.	183	Alarm message	The special canned cycle can only be executed in G17 plane.
Analyzing	G110-G137 special canned cycle and G120-G125 continuous drilling cycle can only be executed on G17 plane.		
Troubleshooting	Modify the program on G17 plane.		
Alarm NO.	185	Alarm message	The corner radius U is too big or I or J is too small.
Analyzing	In G130-G137 rectangle milling groove cycle, U is too big or I or J is too small.		
Troubleshooting	Modify the program to make U smaller or I or J is increased.		
Alarm NO.	186	Alarm message	U value is less than the tool radius.
Analyzing	In the rectangle milling finish cycle, U value is less than the tool radius.		
Troubleshooting	Modify the program to increase U or change the tool.		
Alarm NO.	187	Alarm message	I or J is too small or L is too big.
Analyzing	In G110-G137 special canned cycle, I or J is too small or L is too big.		
Troubleshooting	Modify the program to increase I or J or decrease L.		

Appendix

Alarm NO.	189	Alarm message	G68 G51 G16 and G51.1 can't be specified in the special canned cycle.
Analyzing	G68 G51 G16 and G51.1 can't be specified in the special canned cycle.		
Troubleshooting	The program should be modified.		
Alarm NO.	200	Alarm message	Alarm: The abnormal situation 0 occurs.
Analyzing	The system protective alarm.		
Troubleshooting	Operation is restarted after the alarm is cleared by resetting; or contact the technician.		
Alarm NO.	201	Alarm message	Alarm: The abnormal situation 1 occurs.
Analyzing	The system protective alarm.		
Troubleshooting	Operation is restarted after the alarm is cleared by resetting; or contact the technician.		
Alarm NO.	202	Alarm message	Alarm: The abnormal situation 2 occurs.
Analyzing	The system protective alarm.		
Troubleshooting	Operation is restarted after the alarm is cleared by resetting; or contact the technician.		
Alarm NO.	203	Alarm message	Alarm: The abnormal situation 3 occurs.
Analyzing	The system protective alarm.		
Troubleshooting	Operation is restarted after the alarm is cleared by resetting; or contact the technician.		
Alarm NO.	204	Alarm message	Errors in setting the machine configuration parameters
Analyzing	The machine configuration parameters aren't proper.		
Troubleshooting	Check whether setting the machine configuration parameters are reasonable or not; or contact the technician.		
Alarm NO.	205	Alarm message	Alarm: The abnormal situation 5 occurs.
Analyzing	The system protective alarm.		

Troubleshooting	Operation is restarted after the alarm is cleared by resetting; or contact the technician.		
Alarm NO.	206	Alarm message	Alarm: The abnormal situation 6 occurs.
Analyzing	The system protective alarm.		
Troubleshooting	Operation is restarted after the alarm is cleared by resetting; or contact the technician.		
Alarm NO.	207	Alarm message	Alarm: The abnormal situation 7 occurs.
Analyzing	The system protective alarm.		
Troubleshooting	Operation is restarted after the alarm is cleared by resetting; or contact the technician.		
Alarm NO.	208	Alarm message	Alarm: The abnormal situation 8 occurs.
Analyzing	The system protective alarm.		
Troubleshooting	Operation is restarted after the alarm is cleared by resetting; or contact the technician.		
Alarm NO.	209	Alarm message	Alarm: The abnormal situation 9 occurs.
Analyzing	The system protective alarm.		
Troubleshooting	Operation is restarted after the alarm is cleared by resetting; or contact the technician.		
Alarm NO.	210	Alarm message	Alarm: The abnormal situation 10 occurs.
Analyzing	The system protective alarm.		
Troubleshooting	Operation is restarted after the alarm is cleared by resetting; or contact the technician.		
Alarm NO.	211	Alarm message	Alarm: The abnormal situation 11 occurs.
Analyzing	The system protective alarm.		
Troubleshooting	Operation is restarted after the alarm is cleared by resetting; or contact the technician.		
Alarm NO.	212	Alarm message	Alarm: The abnormal situation 12 occurs.

Analyzing	The system protective alarm.		
Troubleshooting	Operation is restarted after the alarm is cleared by resetting; or contact the technician.		
Alarm NO.	213	Alarm message	Alarm: The abnormal situation 13 occurs.
Analyzing	The system protective alarm.		
Troubleshooting	Operation is restarted after the alarm is cleared by resetting; or contact the technician.		
Alarm NO.	214	Alarm message	Alarm: The abnormal situation 14 occurs.
Analyzing	The system protective alarm.		
Troubleshooting	Operation is restarted after the alarm is cleared by resetting; or contact the technician.		
Alarm NO.	215	Alarm message	Alarm: The abnormal situation 15 occurs.
Analyzing	The system protective alarm.		
Troubleshooting	Operation is restarted after the alarm is cleared by resetting; or contact the technician.		
Alarm NO.	216	Alarm message	Alarm: The abnormal situation 16 occurs.
Analyzing	The system protective alarm.		
Troubleshooting	Operation is restarted after the alarm is cleared by resetting; or contact the technician.		
Alarm NO.	217	Alarm message	Alarm: The abnormal situation 17 occurs.
Analyzing	The system protective alarm.		
Troubleshooting	Operation is restarted after the alarm is cleared by resetting; or contact the technician.		
Alarm NO.	218	Alarm message	Alarm: The abnormal situation 18 occurs.
Analyzing	The system protective alarm.		
Troubleshooting	Operation is restarted after the alarm is cleared by resetting; or contact the technician.		
Alarm NO.	219	Alarm	Alarm: The abnormal situation 19 occurs.

		message	
Analyzing	The system protective alarm.		
Troubleshooting	Operation is restarted after the alarm is cleared by resetting; or contact the technician.		
Alarm NO.	220	Alarm message	Alarm: The abnormal situation 20 occurs.
Analyzing	The system protective alarm.		
Troubleshooting	Operation is restarted after the alarm is cleared by resetting; or contact the technician.		
Alarm NO.	221	Alarm message	G code transmitting wrong
Analyzing	The system protective alarm.		
Troubleshooting	Operation is restarted after the alarm is cleared by resetting; or contact the technician.		
Alarm NO.	222	Alarm message	Alarm: Error in sending the plane.
Analyzing	The system protective alarm.		
Troubleshooting	Operation is restarted after the alarm is cleared by resetting; or contact the technician.		
Alarm NO.	223	Alarm message	Alarm: The abnormal situation 23 occurs.
Analyzing	The system protective alarm.		
Troubleshooting	Operation is restarted after the alarm is cleared by resetting; or contact the technician.		
Alarm NO.	224	Alarm message	Alarm: The abnormal situation 24 occurs.
Analyzing	The system protective alarm.		
Troubleshooting	Operation is restarted after the alarm is cleared by resetting; or contact the technician.		
Alarm NO.	225	Alarm message	Alarm: The abnormal situation 25 occurs.
Analyzing	The system protective alarm.		
Trouble shooting	Operation is restarted after the alarm is cleared by resetting; or contact the technician.		

Alarm NO.	226	Alarm message	Fail to return the reference position
Analyzing	During the program running, the alarm occurs because the machine with the block zero return gets zero lost after switching off, emergency stop or servo alarm.		
Trouble shooting	Return to the reference position, again.		
Alarm NO.	227	Alarm message	Alarm: The abnormal situation 27 occurs.
Analyzing	The system protective alarm.		
Trouble shooting	Operation is restarted after the alarm is cleared by resetting; or contact the technician.		
Alarm NO.	228	Alarm message	Alarm: The abnormal situation 28 occurs.
Analyzing	The system protective alarm.		
Trouble shooting	Operation is restarted after the alarm is cleared by resetting; or contact the technician.		
Alarm NO.	229	Alarm message	Alarm: The abnormal situation 29 occurs.
Analyzing	The system protective alarm.		
Trouble shooting	Operation is restarted after the alarm is cleared by resetting; or contact the technician.		
Alarm NO.	230	Alarm message	The rigid tapping VPO signal is invalid.
Analyzing	During rigid tapping, the spindle servo position mode is cancelled.		
Troubleshooting	Check the spindle servo.		
Alarm NO.	231	Alarm message	The machine lock is cancelled.
Analyzing	In the machine lock state, run the program or move the axis, and then cancel the machine lock.		
Troubleshooting	The machine with the block zero return should be operated zero return, again; the machine without the block zero return is directly reset.		
Alarm NO.	232	Alarm message	Alarm: Emergency stop
Analyzing	In emergency stop, press the emergency stop button, the machine is forbidden to operate.		
Troubleshooting	After troubleshooting, release the emergency stop button.		

Appendix

Alarm NO.	233	Alarm message	Error in executing positioning.
Analyzing	The system protective alarm.		
Troubleshooting	Operation is restarted after the alarm is cleared by resetting; or contact the technician.		
Alarm NO.	234	Alarm message	Errors in coding and interpolation communication
Analyzing	The system protective alarm.		
Troubleshooting	Operation is restarted after the alarm is cleared by resetting; or contact the technician.		
Alarm NO.	235	Alarm message	Alarm: The abnormal situation 35 occurs.
Analyzing	The system protective alarm.		
Troubleshooting	Operation is restarted after the alarm is cleared by resetting; or contact the technician.		
Alarm NO.	236	Alarm message	Tapping spindle speed is zero.
Analyzing	Before tapping starts, the spindle speed isn't commanded.		
Troubleshooting	Check the program to confirm whether the spindle speed is commanded or not.		
Alarm NO.	237	Alarm message	Alarm: The abnormal situation 37 occurs.
Analyzing	The system protective alarm.		
Troubleshooting	Operation is restarted after the alarm is cleared by resetting; or contact the technician.		
Alarm NO.	238	Alarm message	Fail to detect G27 reference position return
Analyzing	The reference position return detection position isn't in the reference position.		
Troubleshooting	Check whether the program commanded position is complied with the reference position.		
Alarm NO.	239	Alarm message	Running time is too long, and it exceeds the range of the system calculator.
Analyzing	The system protective alarm.		

Appendix

Troubleshooting	Operation is restarted after the alarm is cleared by resetting; or contact the technician.		
Alarm NO.	240	Alarm message	The 2nd stroke detection forbidden area
Analyzing	Enter the 2nd stroke detection forbidden area from the system current position.		
Troubleshooting	Check the program to confirm whether the system commands to enter the 2nd stroke detection forbidden area		
Alarm NO.	241	Alarm message	The 3rd stroke detection forbidden area
Analyzing	Enter the 3rd stroke detection forbidden area from the system current position.		
Troubleshooting	Check the program to confirm whether the system commands to enter the 3rd stroke detection forbidden area		
Alarm NO.	242	Alarm message	The feedrate is zero
Analyzing	In feeding per revolution mode, the feedrate isn't commanded.		
Troubleshooting	Check the program to confirm whether the system commands the feedrate.		
Alarm NO.	243	Alarm message	The spindle speed is zero.
Analyzing	In feeding per revolution mode, the spindle speed isn't commanded.		
Troubleshooting	Check the program to confirm whether the spindle speed is commanded.		
Alarm NO.	244	Alarm message	Illegal operation is executed in G88 boring
Analyzing	In G88 mode, it doesn't return into the original operation mode for running after switching into the operation mode.		
Troubleshooting	Return to the original operation mode and then run.		
Alarm NO.	245	Alarm message	Alarm: The abnormal situation 45 occurs.
Analyzing	The system protective alarm.		
Troubleshooting	Operation is restarted after the alarm is cleared by resetting; or contact the technician.		
Alarm NO.	246	Alarm message	Alarm: The abnormal situation 46 occurs.

Analyzing	The system protective alarm.		
Troubleshooting	Operation is restarted after the alarm is cleared by resetting; or contact the technician.		
Alarm NO.	247	Alarm message	Alarm: The abnormal situation 47 occurs.
Analyzing	The system protective alarm.		
Troubleshooting	Operation is restarted after the alarm is cleared by resetting; or contact the technician.		
Alarm NO.	248	Alarm message	Alarm: The abnormal situation 48 occurs.
Analyzing	The system protective alarm.		
Troubleshooting	Operation is restarted after the alarm is cleared by resetting; or contact the technician.		
Alarm NO.	249	Alarm message	Alarm: The abnormal situation 49 occurs.
Analyzing	The system protective alarm.		
Troubleshooting	Operation is restarted after the alarm is cleared by resetting; or contact the technician.		
Alarm NO.	250	Alarm message	Alarm: The abnormal situation 50 occurs.
Analyzing	The system protective alarm.		
Troubleshooting	Operation is restarted after the alarm is cleared by resetting; or contact the technician.		
Alarm NO.	251	Alarm message	Alarm: The abnormal situation 51 occurs.
Analyzing	The system protective alarm.		
Trouble shooting	Operation is restarted after the alarm is cleared by resetting; or contact the technician.		
Alarm NO.	278	Alarm message	Exceeding the spindle maximum speed
Analyzing	During rigid tapping, the speed of the spindle exceeds the maximum speed set by the parameters（#2140-#2143）.		
Troubleshooting	Reduce the spindle speed during tapping or rewrite the value of the parameters （#2140-#2143）.		
Alarm NO.	279	Alarm	The positioning spindle can't be with the

Appendix

		message	other movement commands in the same block.
Analyzing	During the spindle positioning, the other axes can't be with the movement command meanwhile.		
Troubleshooting	The spindle positioning and the movement commands of the other axes are divided into two blocks.		
Alarm NO.	301	Alarm message	The servo system communication is abnormal.
Analyzing	The errors exist in the connection between the system and the slave equipment port, or problems in the network cable, or in the hardware interface.		
Trouble shooting	In the connected internet, the internet indicator should keep normal after power supply; otherwise, troubleshooting should be operated.		
Alarm NO.	303	Alarm message	The servo axis number should be corresponded to the logic address error.
Analyzing	The slave logic address set by parameter #1022 doesn't correspond with the actual connected logic address.		
Trouble shooting	Rewrite #1022 parameter value.		
Servo and bit control alarm (PV alarm)			
Alarm NO.	1	Alarm message	Servo exceeding the speed
Analyzing	The servo drive suddenly receives a bigger commanded value due to the interference; the speed exceeds the value set by parameter #4223.		
Troubleshooting	Check whether the earth wire and network cable is well connected or not; rewrite the value set by 4223.		
Alarm NO.	2	Alarm message	Servo main circuit is over voltage
Analyzing	The fault in the power bottom board; the brake resistance capacity is not enough; the main return circuit voltage is too high.		
Troubleshooting	Change the servo drive; replace with the brake resistance with bigger power; check whether the main return circuit voltage is in the servo allowable voltage range.		
Alarm NO.	3	Alarm message	The servo main circuit is less voltage.
Analyzing	The fault in the power bottom board; the power supply capacity of the main return circuit is not enough; the mechanical part gets stuck or the motor instantly outputs the bigger torque which causes the busbar voltage dropped.		

Appendix

Troubleshooting	Change the servo drive; check whether the main return circuit and the transformer capacity are enough or not; check the mechanical device.		
Alarm NO.	4	Alarm message	Servo position excess-error
Analyzing	The position differential value between the commanded and the feedback one exceeds the maximum value set by the parameters, and it is divided into two types of the dynamic excess-error and the static one.		
Troubleshooting	In the normal situation, the system alarms: Excess-error occurs, so the parameters values of #4111 or #4112 should be increased properly; in the abnormal situation, the alarm occurs, and it should be analyzed though the diagnosis parameters; about the details, please refer to the diagnosis introduction.		
Alarm NO.	6	Alarm message	Fail to shake hand of the servo communication
Analyzing	The network cable is poor connected; the connection between the drive and the system cabinet earth is poor.		
Troubleshooting	Pull and insert the network cable, again; the drive and the system cabinet earth connection should be good; the alarm is cleared through resetting or restarting.		
Alarm NO.	8	Alarm message	Multi-ring data of the encoder is abnormal
Analyzing	The encoder wire is pulled out more than half an hour; the battery voltage is less than 3.6V.		
Troubleshooting	After inserting the encoder, again, and the column of “APZ” in the system parameter 4001 is changed into 0, and then changed into 1, and the alarm is cleared through resetting; the battery is changed, again, and the alarm is cleared based on the above description.		
Alarm NO.	9	Alarm message	Servo encoder communication is wrong.
Analyzing	The encoder wire is cut off or the plug welding is poor, which causes the servo drive communication wrong.		
Troubleshooting	Check whether the encoder wire is cut or not; check whether the plug welding point is with the void-welding; check whether the connection between the end and the servo drive is normal or not.		
Alarm NO.	11	Alarm message	Servo IPM module fault
Analyzing	The mechanical part is stuck; the drive UVW is connected reversely; the bigger command is received instantly; 4286 parameter setting isn’t proper; IPM module gets breakdown.		
Troubleshooting	Check the mechanical device; correctly connect UVW phase wire; check		

Appendix

oting	whether the earth wire, the encoder wire and the network cable are connected well or not; the emergency stop occurs due to high speed, just power on and restart; 4286 is increased, and about the detailed setting value, adjust the parameters; change the servo drive.		
Alarm NO.	12	Alarm message	Servo over current
Analyzing	The mechanical part is stuck; the drive UVW phase is connecter reversely; the bigger command is received instantly; the parameter 4276 setting is wrong.		
Troubleshooting	Check the mechanical device; connect UVW phase correctly; check whether the earth wire, the encoder wire and the network cable are connected well or not; the emergency stop occurs due to high speed, just power on and restart; parameter 4276 is increased, about the details, please refer to the parameter adjusting.		
Alarm NO.	13	Alarm message	The servo encoder feedback data abnormal
Analyzing	The grid is with bigger interference; the connection is poor.		
Troubleshooting	The servo drive power supply is filtered with the isolator transformer and the grid isolation; check the encoder wire and the earth wire connection.		
Alarm NO.	14	Alarm message	The servo brake fault
Analyzing	The brake resistance power is not enough; the power bottom board circuit fault.		
Troubleshooting	Replace with the brake resistance with bigger power; change the servo drive.		
Alarm NO.	15	Alarm message	Alarm: The motor polar pair
Analyzing	Setting the motor type code is wrong.		
Troubleshooting	In the system, the servo drive default parameters are restored, read the motor type code and the polar pair, again and then restarted after power off.		
Alarm NO.	16	Alarm message	Alarm: The servo main return circuit power off
Analyzing	The main return circuit isn't powered on or the voltage isn't enough; the power bottom board circuit fault.		
Troubleshooting	Check the main return circuit voltage; change the servo drive.		
Alarm NO.	18	Alarm message	Error in the servo motor type code

Analyzing	The motor type isn't written in; and the encoder wire isn't connected.		
Troubleshooting	Rewrite the motor type; connect the encoder wire.		
Alarm NO.	19	Alarm message	The servo encoder disconnection due to the error
Analyzing	The encoder communication is interfered; and the connection between the encoder wire and the servo drive is poor.		
Troubleshooting	Check the earth wire, and ensure the earth wire connection is well; check whether the encoder is wore or not, and pull and insert the encoder wire, again.		
Alarm NO.	21	Alarm message	Alarm: AC is lack of the phase.
Analyzing	The main return circuit is only connected the voltage in single phase.		
Troubleshooting	The three-phase voltage is connected based on the servo drive requirements.		
Alarm NO.	22	Alarm message	Servo initialization failure
Analyzing	The servo drive DSP doesn't match with FPGA software; FPGA fault.		
Troubleshooting	Burn DSP and FGPA softwares, again; change the servo drive.		
Alarm NO.	23（only use for GH2030）	Alarm message	Brake feedback alarm
Analyzing	The power bottom board gets malfunction.		
Troubleshooting	Change the servo drive.		
Alarm NO.	100	Alarm message	The servo speed command is abnormal.
Analyzing	The commanded incremental position or the feedback incremental position is too big, or the set speed parameter isn't proper.		
Troubleshooting	The troubleshooting is operated based on the diagnosis data; please refer to the detailed introduction of the diagnosis.		
Alarm NO.	102	Alarm message	The servo axis doesn't exist.
Analyzing	There isn't corresponding servo axis number.		
Troubleshooting	The corresponding axis number is shielded by parameter #4000.		

Alarm NO.	104	Alarm message	The synchronous mode command is illegal
Analyzing	During moving, the servo axis synchronous control mode is changed.		
Troubleshooting	Before the axis moves, confirm the synchronous control mode of the servo axis.		
Alarm NO.	105	Alarm message	The synchronous error of the position offset is too big
Analyzing	During the servo axis synchronous control, the offset differential value between the drive axis and the slave axis is bigger than the set synchronous error value.		
Troubleshooting	Correct the set synchronous error value; or confirm the consistency of the drive axis and the slave axis set by the parameters.		
Alarm NO.	106	Alarm message	The synchronous error of the machine coordinate is too big.
Analyzing	During the servo axis synchronous control, the machine coordinate differential value of the drive axis and the slave axis is bigger than the machine coordinate synchronous error value.		
Troubleshooting	Correct the set machine coordinate synchronous error value, or confirm the compliance of the drive axis and the slave axis set by the parameters.		
Alarm NO.	108	Alarm message	The calibration error in the feeding synchronous control
Analyzing	In the servo axis synchronous control, detect whether there exists the synchronous calibration error in the system initialization process.		
Trouble shooting	Correct the synchronous control calibration setting value.		
Alarm NO.	109	Alarm message	Alarm: Exceed torque differential value limit
Analyzing	Under the servo axis synchronous control, the torque differential value exceeds the set parameter value.		
Troubleshooting	Rewrite the parameter value 4025, or exclude the reasons which cause the abnormal torque differential value.		
Alarm NO.	110	Alarm message	NC and PMC axis control commands conflict
Analyzing	When PLC axis control function is valid, the program commands the corresponding axis movement.		
Troubleshooting	The program should be modified.		
Alarm NO.	111	Alarm message	PMC control axis can't be changed.

Analyzing	When PLC axis control selection is valid and moved, PLC axis control selection signal is cancelled.		
Troubleshooting	Check PLC ladder diagram.		
Alarm NO.	112	Alarm message	Setting the servo zero overtime
Analyzing	Overtime occurs when the servo zero is set.		
Troubleshooting	Reset the servo zero, or check whether the servo version is the newest one.		
Alarm NO.	113	Alarm message	System position data illegal.
Analyzing	The system alarms when the input commanded position data are illegal.		
Troubleshooting	The main reason is the abnormal situations occurs; and exclude the reasons which cause the input commanded data illegal.		
Alarm NO.	114	Alarm message	The incremental grating axis doesn't return the reference position
Analyzing	When the incremental grating equipment is used, the grating is valid while the zero isn't set, and the system alarms during automatic running.		
Troubleshooting	Directly set zero position or zero return is operated in Manual mode.		
Alarm NO.	115	Alarm message	The grating scale axis equipment doesn't exist.
Analyzing	When the grating axis selection is valid, the corresponding axis grating equipment doesn't exist.		
Troubleshooting	Cancel the axis grating setting and select the parameters.		
Alarm NO.	116	Alarm message	The differential value between the grating coordinate and the servo one exceeds the allowable value
Analyzing	The protective alarm, the difference between the grating closed-loop value and the servo feedback coordinate value exceeds the allowable parameter value.		
Troubleshooting	Rewrite the value of parameter 4115 and confirm whether the grating direction is complied with the servo movement one, or with the gear ratio, rewrite the corresponding parameter value of 4001#RDIR or 4212/4213.		
Alarm NO.	117	Alarm message	Grating scale type wrong
Analyzing	At present, only Heidenhain absolute linear grating and rotating grating can be supported; and the system alarms if the other types occur.		

Appendix

Troubleshooting	Cancel the grating axis selection valid parameters, or confirm the grating equipment hardware problem.		
Alarm NO.	118	Alarm message	Position interface unit and grating communication alarm
Analyzing	The alarm occurs in the position interface slave unit and the grating equipment communication		
Troubleshooting	Confirm the problem of the hardware connection.		
Alarm NO.	119	Alarm message	When the servo enable is disconnected, the system alarms due to abnormally running.
Analyzing	During the system running, the servo axis is cut off.		
Troubleshooting	The positioning causes the servo disconnected.		
Alarm NO.	120	Alarm message	Pulse servo alarm
Analyzing	When the pulse servo selection is valid, the alarm of the pulse servo occurs.		
Troubleshooting	Check whether ISC parameter is 0 or not, please refer to the diagnosis introduction.		
Alarm NO.	121	Alarm message	Position PID internal operation abnormal
Analyzing	The system PID internal operation is abnormal.		
Troubleshooting	The protective alarm, it is caused by the abnormal calculated output value and the program should be comprehensively diagnosed.		
Alarm NO.	130	Alarm message	Zero return abnormal
Analyzing	The correct position can't be returned during the nth axis zero return.		
Troubleshooting	Correctly input the nth screw pitch parameter or adjust zero return block position.		
Alarm NO.	131	Alarm message	Positive software limit
Analyzing	The machine coordinate exceeds or is equal to the set value of the nth axis software limit set by the system parameter 1081.		
Troubleshooting	Move the nth axis in the negative direction, and leave the position set by the parameter 1081.		
Alarm NO.	132	Alarm message	Negative software limit

Analyzing	The machine coordinate exceeds or is equal to the set value of the nth axis software limit set by the system parameter 1081.		
Troubleshooting	Move the nth axis in positive direction and leave the position set by the parameter 1081.		
Alarm NO.	133	Alarm message	Position hardware overtravel
Analyzing	The nth axis positive limit switch is pressing the limit block.		
Troubleshooting	Move the nth axis in negative direction and leave the limit block position.		
Alarm NO.	134	Alarm message	Negative hardware overtravel
Analyzing	The nth axis negative limit switch is pressing the limit block.		
Troubleshooting	Move the nth axis in positive direction and leave the limit block position.		
Spindle alarm (PD alarm)			
Alarm NO.	1	Alarm message	Spindle over speed
Analyzing	The motor is started instantly or running, the motor running speed exceeds the alarm speed set by the drive.		
Troubleshooting	1. Power on, again；2. Check the connection circuit and the parameter setting.		
Alarm NO.	2	Alarm message	Main circuit over voltage
Analyzing	The drive main circuit over voltage alarm		
Troubleshooting	1. Check the parameter setting; 2. Check the brake resistance connection and increase the deceleration time; 3. Change the drive.		
Alarm NO.	3	Alarm message	Main circuit less voltage
Analyzing	Alarm: The drive main circuit is less voltage.		
Troubleshooting	1. Check the power supply；2. Reduce the frequency of running start/stop, increase the acceleration time；3. Change the drive。		
Alarm NO.	6	Alarm message	Speed regulator saturation for a long time
Analyzing	During the motor running, the alarm occurs due to too big load in a short time.		

Appendix

Troubleshooting	1. Increase the speed gain of the drive；2. Increase the integral separate point parameters.		
Alarm NO.	9	Alarm message	The encoder fault
Analyzing	1. Encoder connection wrong; 2. Encoder damaged; 3. Poor encoder cable; 4. Encoder cable too long which causes the encoder power supply voltage too low.		
Troubleshooting	1. Check connection; 2. Change the motor; 3. Change the cable; 4. Shorten the cable。		
Alarm NO.	10	Alarm message	Control power supply less voltage
Analyzing	The input control power supply is less voltage.		
Troubleshooting	1. Check the input power supply; 2. Change the drive unit.		
Alarm NO.	11	Alarm message	IPM module fault
Analyzing	IPM module alarm		
Troubleshooting	1. Check the parameter setting; 2. Change the drive; 3. Increase the acceleration and deceleration time; 4. Check the motor drive wire connection。		
Alarm NO.	12	Alarm message	Overcurrent
Analyzing	The drive over current alarm: Transient current is too big.		
Troubleshooting	1. Check the parameter setting; 2. Change the drive; 3. Check the motor drive wire connection.		
Alarm NO.	13	Alarm message	Overload
Analyzing	Alarm: Overload is too big.		
Troubleshooting	1. Check the parameter setting; 2. Check the mechanical part: The load is too big or the mechanical part gets stuck; 3. Change the drive。		
Alarm NO.	14	Alarm message	Brake fault
Analyzing	The brake time is too long.		
Troubleshooting	1. Check the brake resistance connection; 2. Change the drive.		

Alarm NO.	15	Alarm message	Encoder counting wrong
Analyzing	The encoder counting is wrong.		
Troubleshooting	1. Change the encoder wire; 2. Check the drive encoder pulse number setting; 3. Change the drive; 4. Change the motor.		
Alarm NO.	20	Alarm message	EEPROM mistake
Analyzing	EEPROM alarm		
Troubleshooting	1. Restore the motor default parameters; 2. Change the drive.		
Alarm NO.	21	Alarm message	CPLD mistake
Analyzing	CPLD alarm。		
Troubleshooting	Change the drive.		
Alarm NO.	30	Alarm message	Encoder Z pulse lost
Analyzing	The encoder Z pulse gets lost		
Troubleshooting	1. Check the encoder wire; 2. Change the drive; 3. Change the motor.		
Alarm NO.	60	Alarm message	Spindle closed-loop excess-error alarm
Analyzing	In the analog spindle closed-loop function and rigid tapping, the difference between the commanded position and the feedback one exceeds the set tapping difference.		
Troubleshooting	Correct the rigid tapping static/dynamic excess-error allowable value and check whether the spindle closed-loop set parameters are proper or not.		
Alarm NO.	61	Alarm message	Spindle alarm
Analyzing	When the spindle alarm function is valid, the spindle alarms.		
Troubleshooting	Find the reasons of the spindle alarm, or check whether the set spindle alarm level parameters are proper or not.		
Alarm NO.	62	Alarm message	Spindle speed abnormal
Analyzing	When the spindle speed fluctuation check function is valid, the spindle speed fluctuation range exceeds the set value.		

Appendix

Troubleshooting	Check whether the set speed fluctuation detection parameters are proper or not, and find the reasons of the spindle speed fluctuation.		
Alarm NO.	63	Alarm message	Exceed the allowable range of the rigid tapping synchronous error width
Analyzing	When the analog spindle closed-loop is valid and in the rigid tapping, the spindle and the tapping axis synchronous error exceeds the set allowable range.		
Troubleshooting	Correct the synchronous error setting range or check whether the spindle and the tapping axis parameters are set properly or not.		