✧ **The user manual is for GSK980TC3 Series Bus Turning CNC System, divided into Programming and Operation.**

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

Preface

Dear users,

It is our pleasure for your patronage and purchase of this machining center CNC system of GSK980TC3 Series Bus Turning CNC system produced by GSK CNC Equipment Co., Ltd.

This book is "Programming and Operation Manual", which introduces the programming and operation of GSK980TC3 Series Bus CNC Turning System (software version V1.4) in detail.

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

Warnings

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

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

Declaration!

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

Warning!

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

Caution!

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

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

Safety notes

- **Transportation and storage**
  - Do not pile up the packing boxes over 6 layers.
  - Never climb the packing box, neither stand on it, nor place heavy objects on it.
  - Do not move or drag the product by the cables connected to it.
  - Avoid impact or scratch to the panel and screen.
  - Packing box should be protected from dampness, insolation and drench.
- Open-package inspection
  - Confirm the product is the one you purchased after opening the package.
  - Check whether the product is damaged during transportation.
  - Confirm all the elements are complete without damage by referring to the list.
  - If there is incorrect product type, incomplete accessories or damage, please contact us in time.
- **Connection**
  - Only qualified personnel can connect and inspect the system.
  - The system must be earthed. The earth resistance should not be greater than 0.1Ω, and a neutral wire (zero wire) cannot be used as an earth wire.
  - The connection must be correct and secured. Otherwise, the product may be damaged or unexpected results may occur.
  - Connect the surge absorbing diode to the product in the specified direction; otherwise the product may be damaged.
  - Turn off the power before inserting or unplugging a plug, or opening the electric cabinet.
- **Troubleshooting**
  - Turn off the power supply before troubleshooting or replacing components.
  - Overhaul the system when there is a short circuit or overload, and do not restart it until the trouble is removed.
  - Do not turn ON/OFF the product frequently, and the ON/OFF interval should be 1 minute at least.

Safety responsibility

Manufacturer Responsibility

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

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

——Be responsible for the provided information and advice.

User Responsibility

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

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

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

This user manual shall be kept by the end user.

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

Ⅰ Programming

This part gives an introduction to the specification, product portfolio, parameter configuration, code codes as well as program format of GSK980TC3 Series Bus Turning CNC System.

Ⅱ Operation

This part gives an introduction to the operation items of GSK980TC3 Series Bus Turning CNC System.

Appendix

This part gives an introduction to the delivery standard parameters and alarm list of GSK980TC3 Series Bus Turning CNC System.

Contents

Ⅰ Programming

Chapter 1 Overview

1 Introduction

With GSK-Link Ethernet bus, MPG trial-cut, Cs axis control, GSK980TC3 Series Bus Turning CNC System is a new CNC system developed by GSK CNC Equipment Co., Ltd., which greatly improves machining speed, precision, surface roughness. Its band-new designed human-machine interface characterizes friendly beauty and easily use; its connection is convenient, which can meet the applied requirements of popularized CNC turning.

- Standard equipped GE servo unit, optional to bus I/O
- Using 8.4 inch TrueColor LCD，supporting Chinese and English
- The least control precision 0.1um，maximum traverse speed 60m/min
- The adaptive servo spindle realizing the spindle orientation, CS axis control
- Single-head/multi-head metric/inch straight thread, taper thread and end thread
- MPG trial-cut, MPG interrupt function
- Supporting RS232 communication
- Providing 12-stage time limit stop setting
- Supporting the servo tool turret, 4-cutter spacing electric tool post, hydraulic tool post

1.2 Technical Specifications

Motion control	Controlled axes: X, Z, Y, C; C axis is taken as Cs axis; optional to 5 axes and 3 link;
	Interpolation: positioning (G00), linear（G01）, circular（G02, G03）
	Maximum programmable dimensions:metric：-99999.999mm～99999.999mm，least code increment：0.001mm Inch:-9999.9999inch～9999.9999inch，least code increment：0.0001inch
	Maximum feedrate: linear 15000mm/min Feedrate override: 0～200％ divided into 12 to realize real-time adjustment
	Maximum rapid traverse speed: 60000 mm/min， Rapid override：F0, 25％, 50％, 100％ to real-time adjustment
	Feed per rev：0.01 mm/r～500mm/r（need to install a spindle encoder 1024P/r or 1200P/r）
	Acceleration/deceleration mode: front acceleration/deceleration（linear, S type）, post acceleration/deceleration（linear, exponential type）
	Electronic gear: frequency multiplying 1～65536，frequency division 1～65536
	MPG feed：0.001, 0.01, 0.1mm；single step feed：0.001, 0.01, 0.1, 1mm
Display interface	* Using a color 8.4 inch LCD with resolution ratio 800×600 * Displaying all machining path
G Function	* Using G code system A including 39 G codes with a fixed cycle code and compound cycle code * Supporting a statement macro program（macro B） * Supporting 5-level subprogram call, and using a user macro program to call
Thread function	* Common thread（following the spindle） * Single-head/multi/head metric/inch straight thread, taper thread, end thread, constant pitch thread and variable pitch thread * Thread run-out length, angle and speed characteristics can be set by parameters * Thread pitch：0.001mm~500mm(metric) 0.06 tooth/inch~25400 tooth/inch (inch)
Compensation function	* Pitch error compensation：compensation interval, compensation origin can be set. Select a one-way thread compensation or bidirection thread compensation * Backlash compensation: can set a fixed frequency or speed-up/down method, support G0 and G1 to use different backlash compensation * Tool compensation: 99 groups tool length compensation and tool nose radius compensation
T Tool function	* Adaptive tool post: set up to 16-cutter spacing electric tool post, LIO SHING company's tool post (12-cutter spacing), DIAMOND company's tool post (encoder or count type) * Toolsetting mode: MDI/automatic absolute tool change or manually relative too change, cutter spacing CW, locking CCW * Toolsetting mode: fixed point toolsetting, trial-cut toolseting, machine zero return toolsetting * Cutter spacing signal input mode: direct input
S Spindle function	* S2 digit（I/O gears control）/ S5 digit（analog output） * Spindle encoder: encoder lines can be set（100 p/r～5000p/r） * Drive ratio between encoder and spindle：（1～255）:（1～255） * Spindle override：50%~120% divided into 8 levels to real-time tuning * 2-channel 0V~10V analog voltage output, supporting double-spindle control
M Miscellaneous function	* Specify with M and 2-digit. M function can be customized * System's interior M codes(they cannot be defined again): end of program M02, M30；program stop M00；optional stop M01；subprogram call M98；end of subprogram M99 *Cooling ON/OFF * lubricating ON/OFF * chuck clamping/releasing in MDI/Auto mode, tailstock forward/backward
Program edit	* Program capacity：57MB, 400 subprograms * Format：relative/absolute compound programming * Subprogram：can be edited, supporting 5-layer subprogram nesting * Program preview *background edit

Operation function	*Mode selection: Edit, Auto, MDI, Zero return, JOG, Single step, MPG *Motion control: Single block, skip, dry run, miscellaneous lock, program restart, MPG interrupt, single step interrupt, MPG interference, machine lock, interlock, feed hold, cycle start, emergency stop, external reset signal, external power supply ON/OFF
PLC function	* PLC processing speed：1us/step；up to 8000 steps；10 basic code，35 functional codes； * I/O input/output:32/32，extensible IO *Select 1~4 PMC axis
Safety function	● Emergency stop ● Hardware stroke limit ● Data backup and recover
Communication function	● RS232：two-way transmission of part programs and parameters, supporting PLC programs ● USB：U-disk file operation, U-disk file's direct machining, supporting PLC programs, system software U-disk upgrade
Adaptive component	* Switching power supply：RS-PB2（provided with a whole set, and installed） * Drive unit：GSK GE series（including incremental and absolute） * Tool post' controller：GSK TB tool post controller

1.3 Product Module Definition

GSK980	T	(C)	(a)	—	(H)	(1)	(1)
1	2	3	4		5	6	

No	Code explanation	
①	Main part property of product model：GSK980TC3 series	
②	Function（machining object）allocation：represented with the capital English letters T-turning machine	
③	Its series' extension: represented with the capital English letters. None: initial version	
④	Its subfamily extension(or improved model): represented with the lower case letters a,b,c……or digital. None: initial version	
⑤	Structural style or special machine style	Structural style：separately represented with the capital English letters U, H, V and B. U-combined， H-horizontal， V-vertical， B-box model special machine style：represented with the capital English P
⑥	LCD dimension（structure） or special machine code	LCD dimension：represented with one Arabic digit 1～9. 1 means 8.4 inch，2 means 10.4 inch，3～9 means…， special machine code：represented with 2-digit Arabic digit 01～99

Example:

◆ **GSK980TC3-U1**：show 980TC3 series，a combined structure, 8.4-inch LCD

◆ **GSK980TC3-P01**：show 980TC3 series， No. 01 special machine

Chapter 2 Programming Fundamentals

2.1 Controllable Axis

Table 2-1-1

Item	GSK990MC
Basic controllable axes	3（X, Y, Z）
Total extended controllable axes	Up to 5 (including Cs axis)

2.2 Axis Name

Name of 3 basic axes is defaulted to X, Z, C.
P005 sets the controllable axis quantity and **P175-P179** sets each additional axis' name, such as A, B, C's axis name.

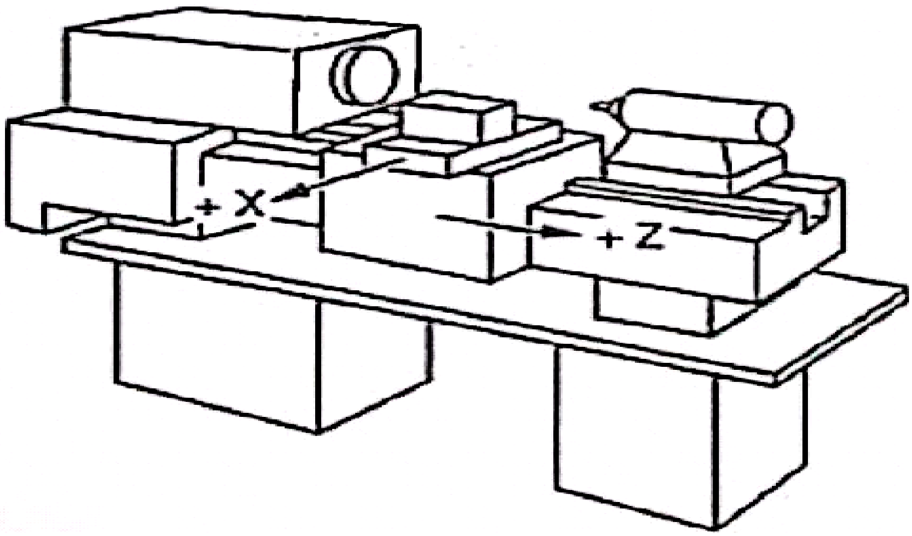


Fig. 2-2-1 Turning machine's CNC axis diagram

The system uses a rectangular coordinate system composed of X, Z axis to execute the positioning and interpolation motion. X axis is the level plane's front and back direction and Z axis is its left and right direction; negative directions of them approach to the workpiece and positive ones are away from it, which is shown in Fig 2-2-1.

The system supports the front tool post and rear tool post function. From the front of machine, the too post in front of the workpiece is called front tool post, and the too post behind of the workpiece is called rear tool post. Fig. 2-2-2 is a coordinate system of the front tool post and Fig. 2-2-3 is a rear toolpost one. they show exactly the opposite of X axes, but the same of Z axes from figures. In the manual, it introduces programming application with the front tool post coordinate system in the following figures and examples.

Ⅰ Programming

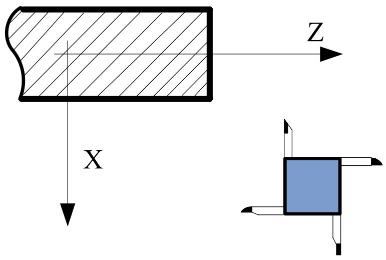


Fig.2-2-2 Front tool post coordinate system

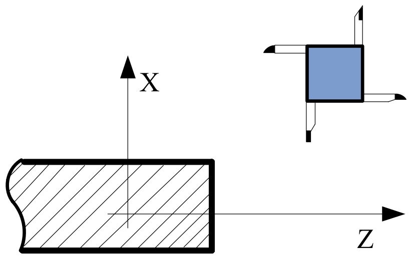


Fig.2-2-3 Rear tool post coordinate system

2.3 Axis Display

GSK980TC3 can set each axis' linear axis or rotary axis by No.8#0~No.8#4; set each axis' display or hide No.58#0~No.58#4.

When the additional axis is set to the rotary, the rotary axis' unit is displayed to deg. When it is set to the linear, its display is the same that of the three basic axis (X, Z, Y), and its unit is mm. The following is the axis display when X, Z is a linear axis, and C is a rotary axis.

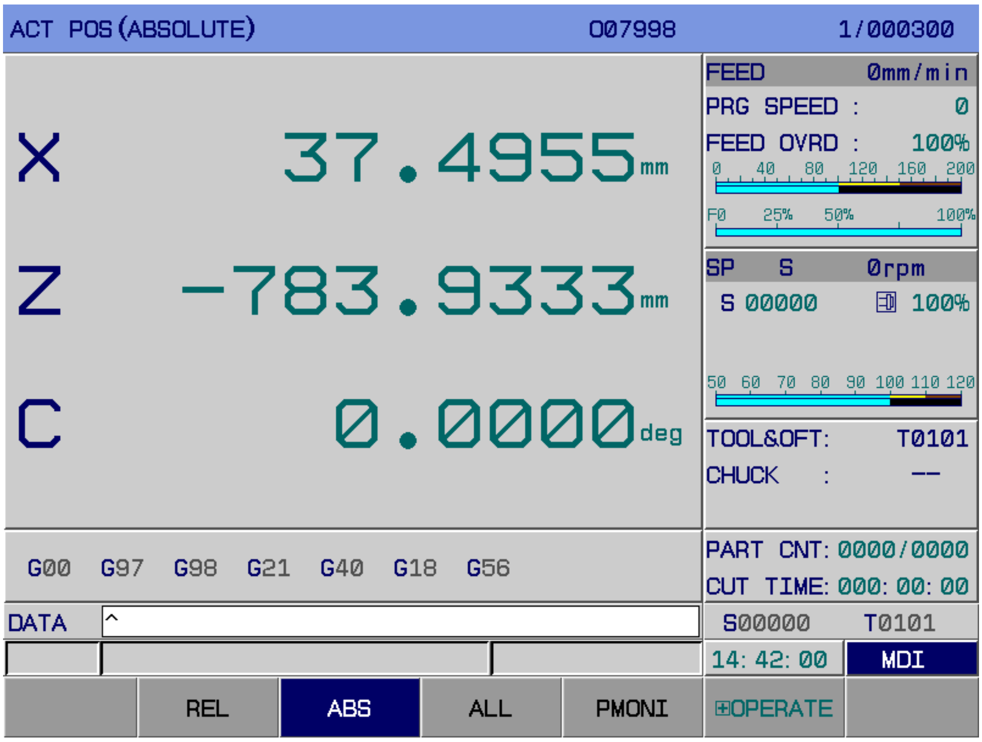


Fig. 2-3-1

2.4 Machine Zero and Machine Coordinate System

A special point used to the machining reference on the machine is called a machine zero. The machine tool manufacturers have set a machine zero on each machine. Generally, it is located at the max. stroke of X, Z's positive direction. After it is set, it is not moved or changed. A coordinate system using the machine zero as an origin setting is called machine coordinate system.

The machine zero's position cannot be confirmed when the CNC is turned on. Generally, automatic or manual machine zero return is executed to create a machine coordinate system. After the machine returns to the machine zero, the CNC automatically creates a machine coordinate system with the machine zero as an origin.

Note: when there is no zero block installed on the machine, the CNC's machine zero function (such as G28) must not be performed.

2.5 Workpiece Coordinate System

When the system machines a workpiece, the used coordinate system is called a workpiece coordinate system(called a part coordinate system). A workpiece coordinate system is set in advance by the CNC (set a workpiece coordinate system).

Generally, a part's all machining programs set one public workpiece coordinate system(select a workpiece coordinate system).

Changing the workpiece coordinate system is executed by moving its origin (changing the workpiece coordinate system's position).

A workpiece coordinate system's reference position should meet conditions as possible, such as easy programming, few dimension conversion, few machining errors. In general conditions, the reference point should be on the dimension marking's reference or positioning's reference. For the turning machine programming, the reference point is on the intersection point between the workpiece axis and chuck's end (Fig. 2-5-1) or the workpiece's end (Fig. 2-5-1).

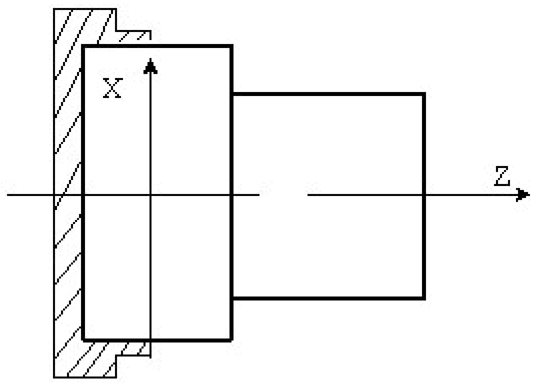


Fig. 2-5-1 the reference point on the chuck's end

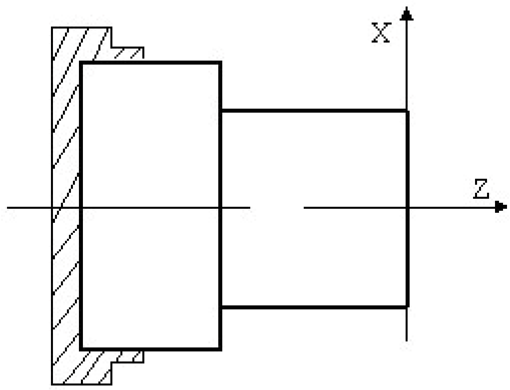


Fig. 2-5-2 the reference point on the workpiece's end

Use the following three methods to set a workpiece coordinate system :

1. Use G50 to set a workpiece coordinate system, and refer to Section 4.2.8
2. Use G54~G59 to set a workpiece coordinate system, and refer to Ⅱ 3.4.2.1
3. Use G54Pn to set a workpiece coordinate system, and refer to Ⅱ 3.4.2.2

2.6 Absolute Value Programming and Incremental Value Programming

A code axis' movement amount method is divided into two: an absolute value code and relative value code. The absolute value code is to use an end point's coordinate value of the axis movement to perform a programming is called an absolute coordinate programming. The incremental coordinate code is to use an axis movement value to directly program, which is called an incremental coordinate programming. In the system, the absolute coordinate programming uses X, Z and the incremental value programming uses U, W.

Table 2-6-1

Absolute value code	Incremental value code	Remark
X	U	X movement code
Z	W	Z movement code

Example: separately use the absolute value, incremental value, absolute value and incremental value programming mode to compile A→B programs in Fig. **2-6-1**.

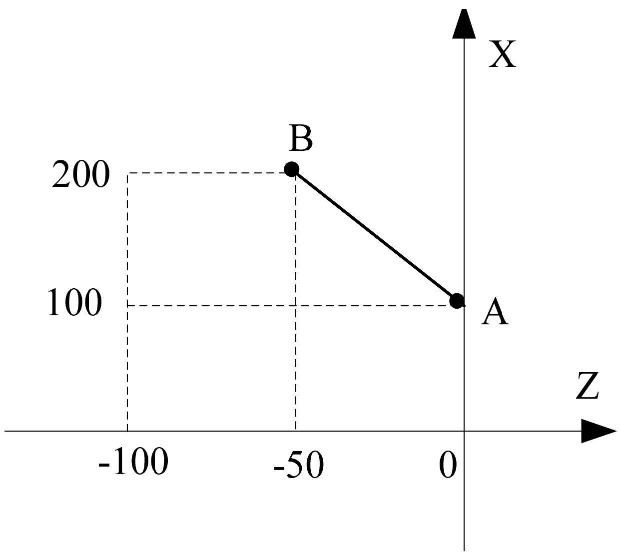


Fig. 2-6-1

Table 2-6-2

Programming mode	Absolute coordinate programming	Relative coordinate programming	Compound coordinate programming
Program	G1 X200 Z-50	G1 U100 W-50	G1 X200 W-50

Remark	Suppose the tool is on the current workpiece coordinate system's A point, and use the linear interpolation to move

Note: an alarm occurs when X/Z and U/W are in one block.

Example: G50 X10 Z20；

G1 X20 W30 U20 Z30；

2. 7 Diameter and Raduis Method to Programming

The outer appearance of the workpiece which is machined by CNC turning machine is rotating body and X dimension can be specified by two methods: diameter and radius methods. It can be set by bit parameter **NO: 39#2**.

When **NO: 39#2** is 0, it is specified by the radius to programming.

When **NO: 39#2** is 1, it is specified by the diameter to programming.

Table 2-7-1 Diameter and radius specification

Item	Specified by the diameter value	Specified by the radius value
Z axis code	Be irrelevant with the diameter value, radius value	
X axis code	Specified by the diameter value	Specified by the radius value
Increment codes of address U	Specified by the diameter value	Specified by the radius value
Setting coordinate system (G50)	Specified by the diameter value	Specified by the radius value
Tool offset X axis value	It is consistent with X axis dimension method and is set by NO:39#2	
Radius code of circular interpolation （R, I , K ）	Specified by the radius value	Specified by the radius value
Feedrate of X axis direction	Radius change (mm/min, mm/r)	
Display X axis position	Display the diameter value	Display the radius value

Note 1: In the following content, it is normally specified by the diameter without special explanation.

Note 2: The meaning of the tool offset value using the diameter/radius is the outer diameter of the work piece changes through the diameter or the radius value when the tool offset value changes. For example: if the diameter specifies and compensation amount is 10mm, then the outer diameter value of cutting work piece changes into 10mm; and if the radius specifies and compensation amount is 10mm, then diameter value of cutting work piece diameter changes into 20mm.

2. 8 Mode, Non-Mode and Initial State

The mode means once the function and status of corresponding fields are executed, it will be valid until the new function and status are executed again. That is to say, the following blocks use the same function and status, and then the fields are not required to input again.

Example:

G0 X100 Z100；（Rapid position to X100 Z100）

X120 Z30；（Rapid position to X120 Z30，G0 is a modal code and can be omitted not to input）

G1 X50 Z50 F300；（Linear interpolation to X50 Z50，feedrate: 300mm/min G0→G1）

X100；（Linear interpolation to X100 Z50，feedrate 300 mm/min, G1, Z50, F300 are modal codes and can be omitted not to input）

G0 X0 Z0；（Rapid position to X0 Z0 G1→G0）

Non-mode means once the function and status of corresponding fields are executed, it will be valid for one time. If the same function and status are required to use again, they should be executed, again. That is to say, the following blocks use the same function and status, and then the fields should be input again.

The initial state is the default function and status after the system is turned on, that is to say, if the corresponding function and state are not specified after power on, it will execute the function in initial mode and status. The initial states are G00, G18,G21, G40, G54, G97,G98.

Example:

O0001；

G0 X100 Z100；（Rapid position in X 100 Z100, G0 is the initial state of the system. When the system does not code other mode, the system executes movement in the initial state G00 mode）

G1 X0 Z0 F100；（Linear interpolation to X0 Z0 at the feedrate 100mm/min. Because G98 is the feed per minute mode and G98 is the system initial state when power on.）

Chapter 3 Structure of an Part Program

3.1 Structure of a Program

A program consists of many blocks, and a block is composed of words. Each block is separated by a code for end of block (ISO uses LF, EIA uses CR). Using a character "； " means a code for end of block.

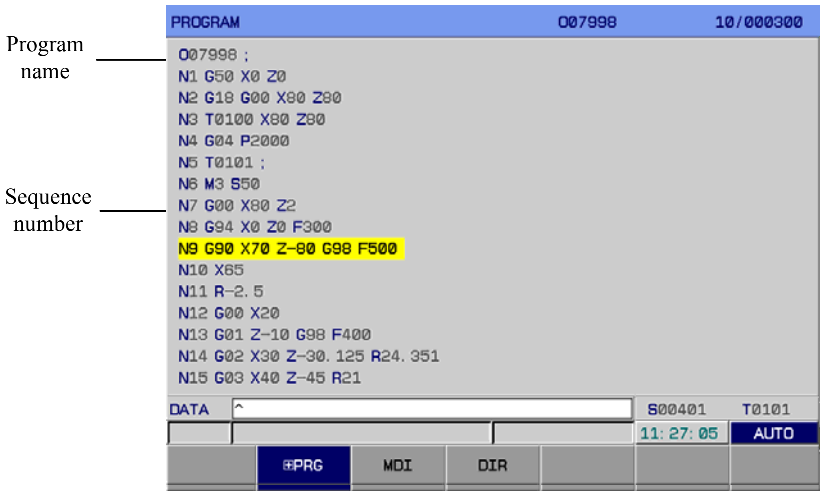


Fig. 3-1-1 structure of a program

A group of codes for controlling the CNC machine to finish workpiece machining is called a program. After the compiled program is input to the CNC system, the system makes the tool move along a straight line or an arc, or rotate or stop the spindle. Please edit these codes according to the actual movement sequence of the machine tool in the program. Structure of the program is shown in Fig. **3-1-1.**

3.1.1 Program Name

In the system, the system's memory can store many programs. In order to mutually differentiate these programs, each program begins with an address O followed by a five-digit number, which is shown in Fig. 3-1-1-1.

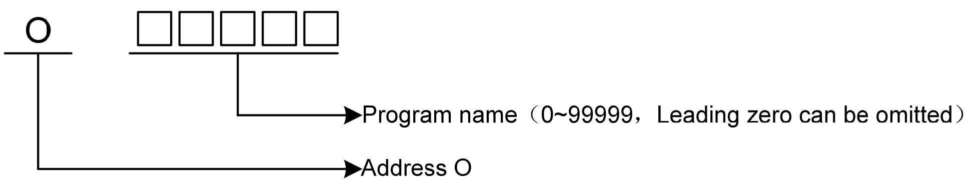


Fig.3-1-1-1 Structure of a program name

3.1.2 Sequence Number and Blocks

A program is composed of many codes and one code unit is called a block (see Fig. 3-1-1). The block is separated through end code (see Fig.3-1-1), it adopts the field "；" as the end code.

Address N with a 4-digit sequence number behind it can be used at the beginning of the block (see Fig. 3-1-1), and the leading zero can be omitted. Sequence numbers (whether the sequence number is inserted is set by Parameter NO: 0 # 5, or set the number in the setting page directly. See Section 3.3.1 in Operation) can be specified in a random order, and the intervals between them can be unequal (set by Data Parameter P210). They can be specified in all blocks, or just in some important blocks. However, the numbers should be arranged in ascending order according to general machining sequence.

Note: The N code is not taken as a line number when it and G10 are in the same block.

3.1.3 Word

A word (See 3-1-3-1) is an element that composes a block. It consists of an address and its following digits (with sign +or - before the digits sometimes).

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

An address is one of the English letters（A～Z）. It specifies the meaning of its following digits. In the system, the used addresses and their meanings as well as their ranges are shown in Fig. 3-1-3-1.

Sometimes, an address may have different meanings based on different preparatory functions.

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

Table 3-1-3-1

Address	Range	Meaning
A, B, C	Set by P175~179	Address of axis name
F	0.001～99999.999（mm/min）	Feedrate per minute
	0.001～500(mm/r)	Feedrate per turn
G	00～99	Preparation function
H	01～99	Operator in G65
I	-9999.9999～9999.9999（mm）	X vector between arc center and start point
	0.06～25400（tooth/inch）	Inch thread tooth quantity
K	-9999.9999～9999.9999（mm）	Z vector between arc center and start point
L	1～9999	Times of recalling a subprogram
,L	-999999.99～999999.99（mm）	Chamfering
M	Set by P204	Miscellaneous function output, program executed flow, subprogram call
N	0～99999	Serial number
O	0～99999	Program name
P	0～99999.9999（ms）	Pause time
	1～99999	Calling subprogram number
	0.001～99999.999（mm）	X cycle movement amount in G74, G75

Address	Range	Meaning
	0～9999	Initial block's serial number in the compound cycle code's finishing
Q	0～9999	End block's serial number in the compound cycle code's finishing
	0.001～99999.999（mm）	Z cycle movement amount in G74, G75
,R R	-999999.99～999999.99（mm）	Cornering value
	-999999.99～999999.99（mm）	Arc radius/angle displacement
	0～99999.999（mm）	G71, G72 cycle tool retraction amount
	1～9999999（times）	Roughing times in G73
	0～99999.999（mm）	Tool retraction amount after cutting in G74, G75
	0～99999.999（mm）	The tool retraction amount to end point in G74, G75
	0～99999.999（mm）	Finishing allowance in G76
	-999999.99～999999.99（mm）	Taper in G90, G92, G94
S	Set by P205	Specify the spindle speed
	00～04	Multi-gear output
T	Set by P206	Tool function
U	Set by P175~179	Address of axis name
	-999999.99～999999.99（mm）	X increment
	-999999.99～999999.99（mm）	X finishing allowance in G71, G72, G73
	0.0001～99999.999 （mm）	cutting depth in G71
	-999999.99～999999.99（mm）	X tool retraction distance in G73
V	Set by P175~179	Address of axis name
	-999999.9～999999.99（mm）	Y increment
W	Set by P175~179	Address of axis name
	-999999.99～999999.99（mm）	Z relative coordinate address
	0～99999.999（mm）	cutting depth in G72
	-999999.99～999999.99（mm）	Z finishing allowance in G71, G72, G73
	-999999.99～999999.99（mm）	Z tool retraction distance in G73
X	Set by P175~179	Address of axis name
	-999999.99～999999.99（mm）	X coordinate address
	0～9999.999（S）	Specify pause time
Y	Set by P175~179	Address of axis name
	-999999.99～999999.99（mm）	Y coordinate address
Z	Set by P175~179	Address of axis name
	-999999.99～999999.99（mm）	Z coordinate address

All described in Table **3-1-3-1** are limited values for the CNC device, but the limit for the machine tool is not described here. Therefore, users are required to refer to the manual provided by the machine tool builder besides this one, in order to get a good understanding of the programming limits before programming.

Note 1: Each word should not exceed 79 characters.

Note 2：The setting unit: ISC=0: maximum value, minimum value in MDI mode is +999999.99mm; ISC=1: maximum value, minimum value in MDI mode is +99999.999mm.

Set unit's displayed value：+999999.999mm　　+99999.9999mm

Set unit's inch：+99999.9999mm　　+9999.99999mm

3.2 General Structure of a Program

The program is divided into main program and subprogram. In general, the CNC system is actuated by the main program. If a code for calling the subprogram is executed in the main program, the CNC system acts by the subprogram. When a code for returning to the main program is executed in the subprogram, the CNC system will return to the main program and execute the following blocks. The program execution sequence is shown in Fig.3-2-1.

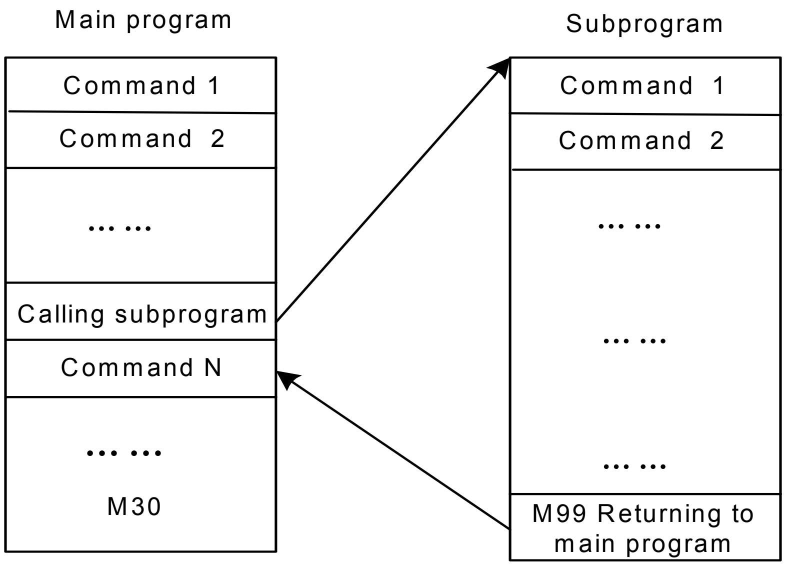


Fig. 3-2-1

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

If a program contains a fixed sequence or frequently repeated pattern, the sequence or pattern can be stored as a subprogram in the memory to simplify the program. The subprogram can be called in Auto mode, usually by M98 in the main program. Besides, the subprogram called can also call another subprogram. The subprogram called from the main program is called the one-level subprogram. Up to 4 levels subprogram can be called in a program (Fig.3-2-2). The last block of a subprogram is the Code M99 used for returning to the main program. After the return, the blocks following the subprogram calling block are executed. (If the last block of a subprogram is ended with M02 or M03, the system will also return to the main program and proceed to the next block, just as ended with M99.)

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

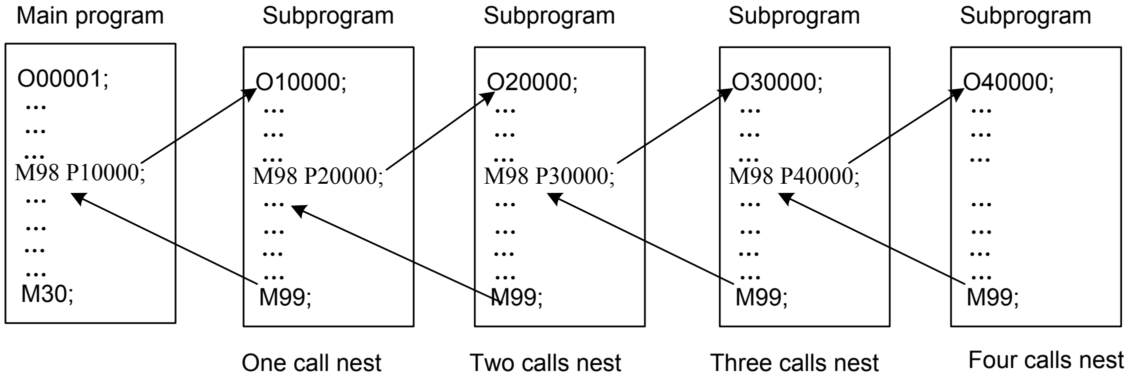


Fig. 3-2-2 Quadruple subprogram nesting

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

3.2.1 Subprogram Writing

Write a subprogram following the format below

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

Fig. 3-2-1-1

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

3.2.2 Subprogram Call

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

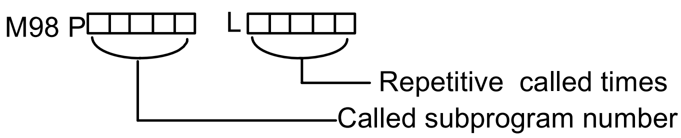


Fig. 3-2-2-1

- If no repetition count is specified, the subprogram is called just once.
  (Example) M98 P1002L5 ;(It means a subprogram with number 1002 is repeatedly called 5 times)
- Execution sequence of calling a subprogram from a main program

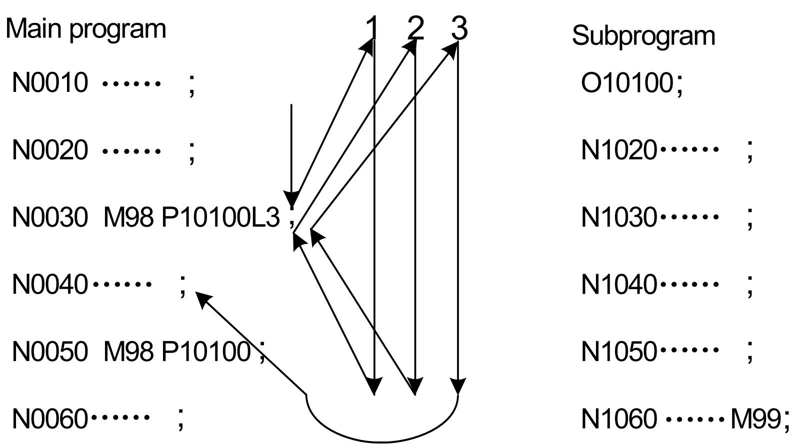


Fig. 3-2-2-2

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

Note 1: An alarm is given when no subprogram number specified with address P is detected.
Note 2: Subprograms with number 90000~99999 are the system reserved programs. When users call such kind of subprograms, the system can execute them but not display them.
Note 3: A subprogram can nest a quadruple.

3.2.3 Program End

The program begins with a program name, and ends with M02, M30 or M99 (see Fig. **3-2-2-2**). For the end code M02, M30 or M99 detected in program execution: If M02 or M03 is executed in a program, the program is terminated, and the reset state is entered; M30 can be set by bit parameter N0.33#4 to return to the program beginning, and M02 can be set by bit parameter N0.33#2 to return to the program beginning. If M99 is executed in a program, the control returns to the beginning of the program, and then executes the program repeatedly; if M99, M02 or M30 is at the end of the subprogram, the control returns to the program that calls the subprogram and goes on executing the following blocks.

Chapter 4 Preparatory Function : G Code

4.1 Categories of Preparatory Function G Code

Preparatory function is represented with G code and its following two digits, and describes its belonging block's meanings. G code is divided into two types:

Table 4-1-1

Classification	Meaning
Non-modal G code	It is valid only in the specified block
Modal G code	It is valid till other code in the same group is commanded

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

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

Table 4-1-2 G code and their functions

Group	G code （G code system A）	Function explanation	Group	G code （G code system A）	Function explanation
00	G04	Dwell, exact stop	02	G96	Constant surface speed control
	G28	Automatic return to machine zero		G97*	Constant surface speed control cancel
	G31	Skip function	05	G98	Feed per minute
	G50	Setting a coordinate system		G99*	Feed per rev
	G65	Macro code non-modal call	06	G20	Inch unit selection
	G70	Finishing cycle		G21*	Metric unit selection
	G71	Axial roughing cycle	07	G40*	Tool nose radius compensation cancel
	G72	Radial roughing cycle		G41	Tool nose radius compensation(right)
	G73	Closed-loop cutting cycle		G42	Tool nose radius compensation (left)
	G74	Axial grooving cycle	14	G54*	Workpiece coordinate system 1
	G75	Radial grooving cycle		G55	Workpiece coordinate system 2
	G76	Multiple thread cutting cycle		G56	Workpiece coordinate system 3
01	G00*	Positioning (rapid traverse)		G57	Workpiece coordinate system 4
	G01	Linear interpolation		G58	Workpiece coordinate system 5
	G02	Circular interpolation CW (clockwise)		G59	Workpiece coordinate system 6
	G03	Circular interpolation CCW (counter clockwise)	16	G17	XY plane selection
	G32	Constant pitch thread cutting		G18*	ZX plane selection
	G34	Variable pitch thread cutting		G19	YX plane selection
	G90	Axial cutting cycle			
	G92	Thread cutting cycle			
	G94	Radial cutting cycle			

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

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

Note 3: G codes in group 00 are non-modal.

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

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

Note 6: G codes are represented by group numbers respectively based on their types. Whether the G codes of each group are cleared after reset or emergency stop is determined by bit parameter NO:35#0~7 and NO:36#0~7.

4.2 G code

4.2.1 Rapid Positioning G00

Format：　G00 X(U)_ Z(W)_

Function: each axis move to the position specified by X(U), Z(W) at its separate rapid traverse speed.

X（U）：absolute (U is an incremental programming code and is the tool movement distance) coordinate value of X positioning end point;

Z（W）：absolute (W is an incremental programming code and is the tool movement distance) coordinate value of Z positioning end point;

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

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

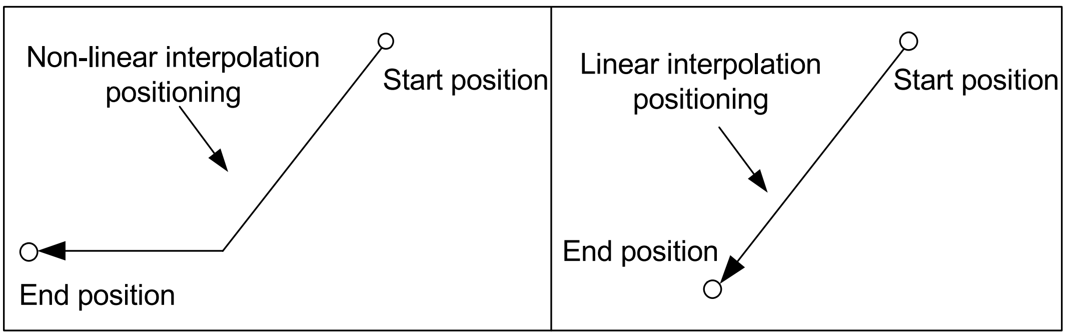


Fig. 4-2-1-1

Explanation:

1. After G00 is executed, the system changes the current tool movement mode for G00 mode. Whether the default mode is G00 (parameter value is 0) or G01 (parameter value is 1) after power-on is set by bit parameter No.031#0.

2. With no positioning parameter specified, the tool does not move and the system only changes the mode of the current tool movement for G00.
3. G00 is the same as G0.
4. The G0 speed of axes X, Y, Z, C and 4th is set by data parameters P88~P92.

Limitations:

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

Example：

G0 X0 Z10 F800； Feeding at the speed set by system parameter

G1 X20 Z50； Using the feedrate of F800

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

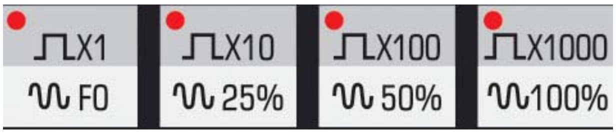


Fig. 4-2-1-2 Keys for rapid feedrate override

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

Example: the tool rapidly positions to point B from point A, which relevant dimension is shown in Fig.4-2-1-3.

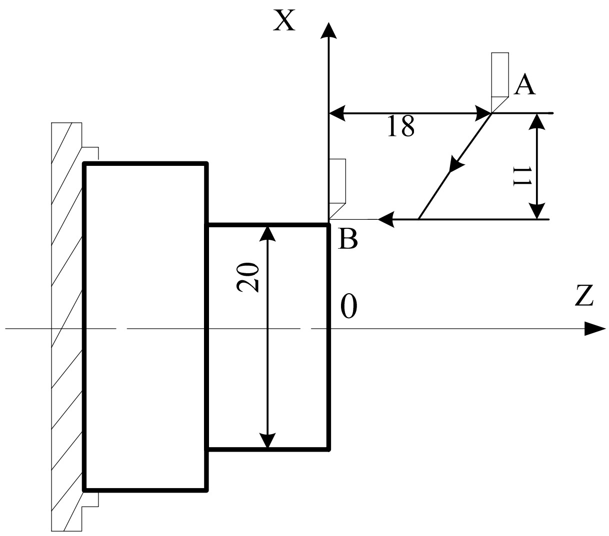


Fig. 4-2-1-3

Programming:

G0 X20 Z0；（absolute value programming, diameter programming）

G0 U-22 W-18；（incremental value programming, diameter programming）

G0 U-22 Z0；（hybrid programming，diameter programming）

Programming

4.2.2 Linear Interpolation G01

Format: **G01 X(U)_ Z(W)_ F_**

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

Explanation:

X（U）: absolute (U is an incremental programming code and is the tool movement distance) coordinate value of X interpolation end point;

Z（W）: absolute (W is an incremental programming code and is the tool movement distance) coordinate value of Z interpolation end point;

1. X_ Y_ Z_ are the coordinates of the end point. Because they are related to the coordinate system, please see Sections 2.4.
2. The feedrate specified by F keeps effective till a new F value is specified. The feedrate specified by F code is calculated by an interpolation along a straight line. If F code is not specified in a program, the default F value at system Power On is used (see data parameter P87 for details).

Code path:

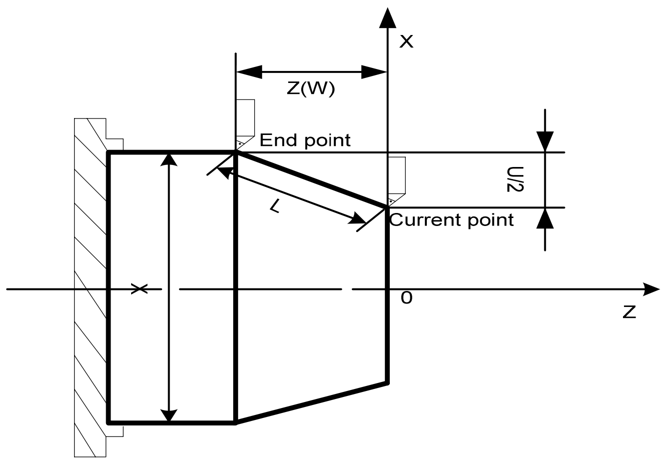
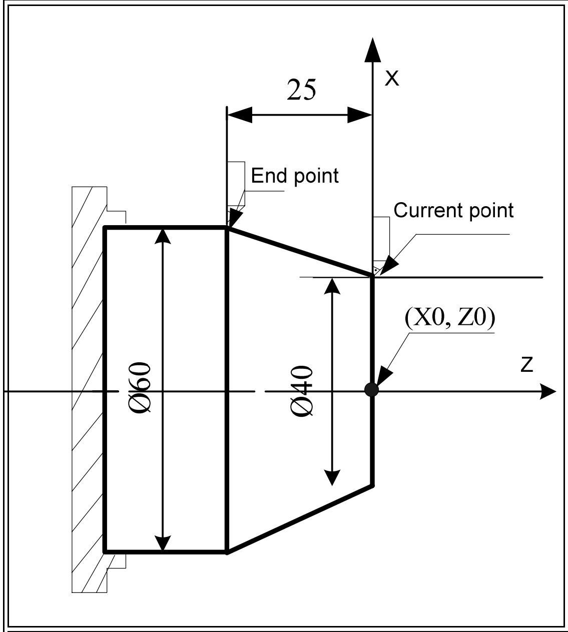


Fig. 4-2-2-1

Program example（Fig. **4-2-2-2**）



G01 X60.0 Z-25;（absolute value programming）
G01 U20.0 W-25.0;（incremental value programming）

Note: each axis' speed is shown below:
G01 Xα ZγFf ;
In the block:

X speed: $f_X = \frac{a}{L} x f$

Z speed: $f_Z = \frac{y}{L} x f$

L= $\sqrt{a^2 + b^2}$

Fig. 4-2-2-2

Note:

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

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

4.2.3 Circular (Helical) Interpolation G02/G03

G02 and G03 provision

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

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

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

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

Arc in XY plane

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

Arc in ZX plane

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

Arc in YZ plane

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

Code path:

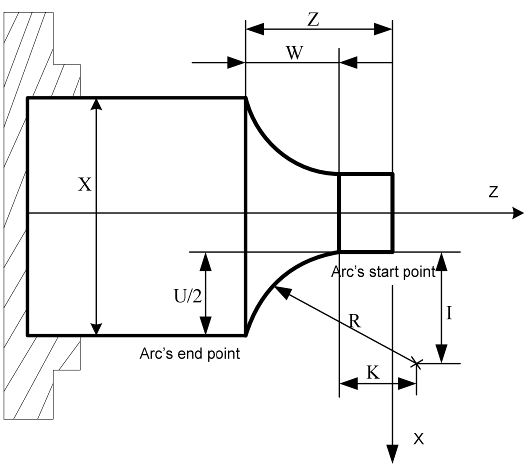


Fig. 4-2-3-1 G02 path

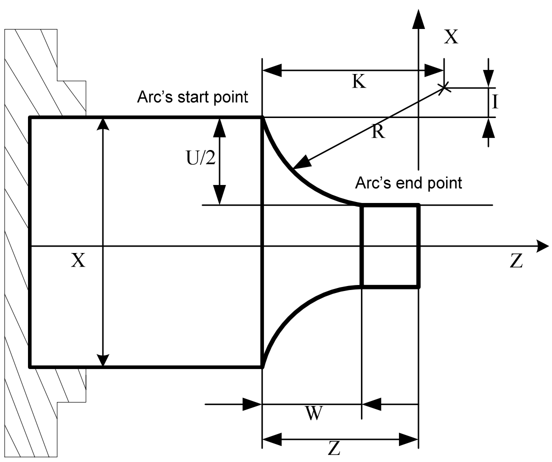


Fig. 4-2-3-2 G03 path

Table 4-2-3-1

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

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

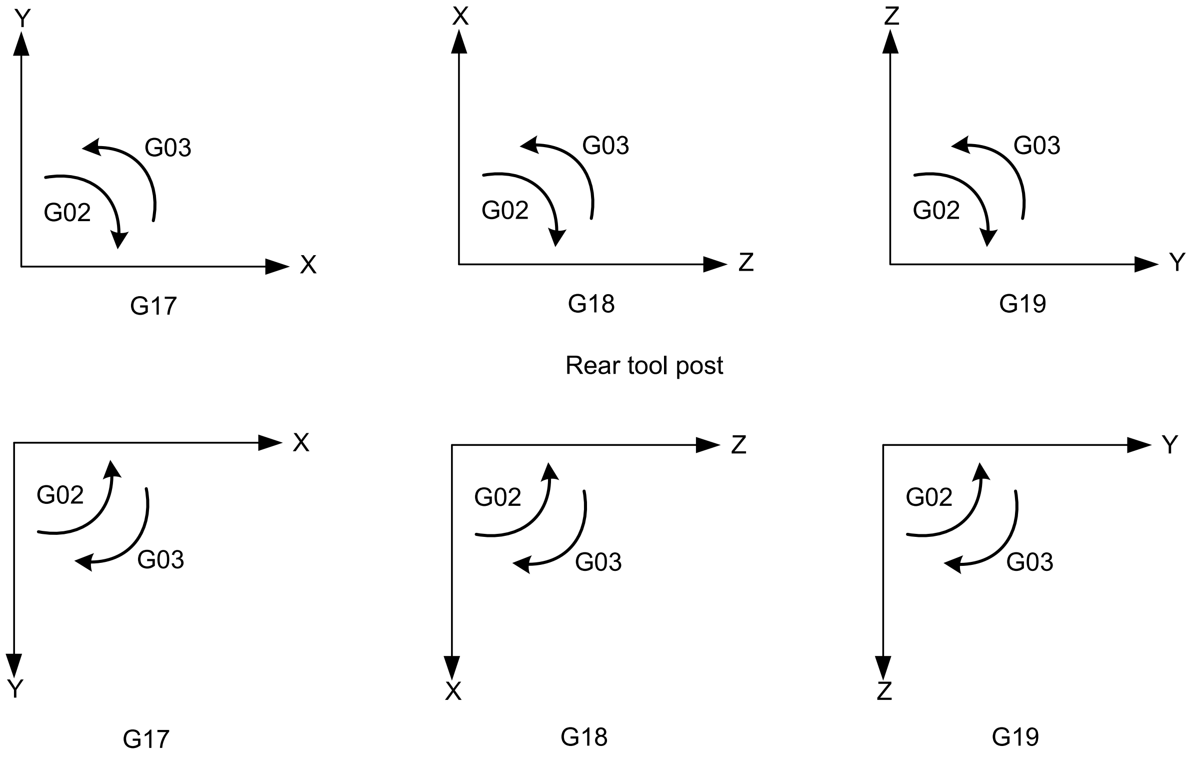


Fig. 4-2-3-3

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

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

Note: the system supports X, Y, Z axis. But, it defaults to be G18 plane and X, Z axis after power on, its command explanations are described based on the system supporting modes, and the user executes operations according to a machine structure.

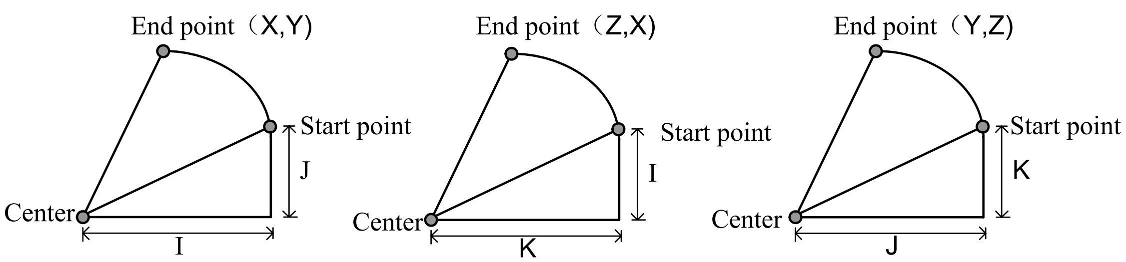


Fig. 4-2-3-4

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

G02 X_ Z_ R_ ；

G03 X_ Z_ R_ ；

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

(Example: Fig. **4-2-3-5**) ①When arc is less than 180°

G02 U60 W20 R50 F300 ；

②When arc is more than 180°

G02 U60 W20 R-50 F300 ；

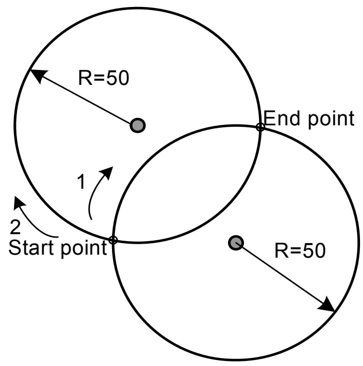


Fig. 4-2-3-5

2. The arc equal to 180° can be programmed either by I, J and K, or by R.

Example: G90 G0 X0 Y0;G2 X20 I10 F100;
Equal to G90 G0 X0 Y0;G2 X20 R10 F100

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

Note: For the arc of 180°, the arc path is not affected whether the value of R is positive or negative.

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

Example: using G02 to compile programs shown in Fig. 4-2-3-6.

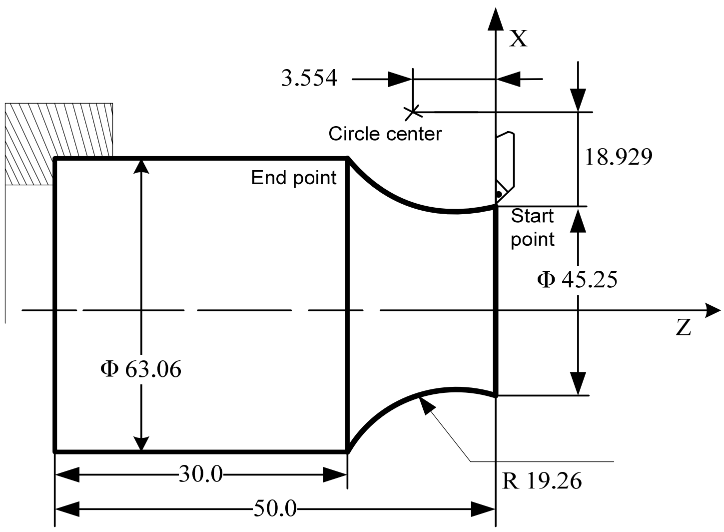


Fig. 4-2-3-6

The programs are shown below: the tool's current point is at start point):

G02 X63.06 Z-20 R19.26 F300 ； or

G02 U17.81 W-20.0 R19.26 F300 ； or

G02 X63.06 Z-20 I18.929 K-3.554 F300 ；　　or

G02 U17.81 W-20.0 I18.929 K-3.554 F300 ；

Restrictions:

1. If addresses I, J, K and R are specified simultaneously in a program, the arc specified by R takes precedence, and others are ignored.
2. If neither arc radius parameter nor the parameter from the start point to the circle center is specified, an alarm is issued in the system.
3. A full circle can only be interpolated by parameters I, J, K from start point to circle center rather than parameter R.
4. Pay attention to the setting for selecting the coordinate plane when the circular interpolation is being done.
5. If X, Y, Z are all omitted (i.e., the start point and the final point coincides), and R is specified (e.g. G02R50), the tool does not move.

4.2.4　Dwell (G04)

Format: G04 X(U)_ or P_

Function: G40 is for dwell operation. The dwell per revolution in Feed per Revolution mode G99 can be specified by bit parameter No.34#0.

Table 4-2-4-1 Value range of dwell time (with X, U code)

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

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

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

Explanation:

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

4.2.6　Workpiece Coordinate System Selection G54～G59

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

Format: G54～G59

Explanation:

1. With no code parameter.
2. The system itself is capable of setting 6 workpiece coordinate systems, any one of which can

be selected by codes G54~G59.

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

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

Example: The corresponding machine tool coordinate for G54 coordinate system origin is (10, 10)

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

When the program is executed in order, the absolute coordinates and machine coordinates of the end point I are displayed as follows:

Table 4-2-6-1

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

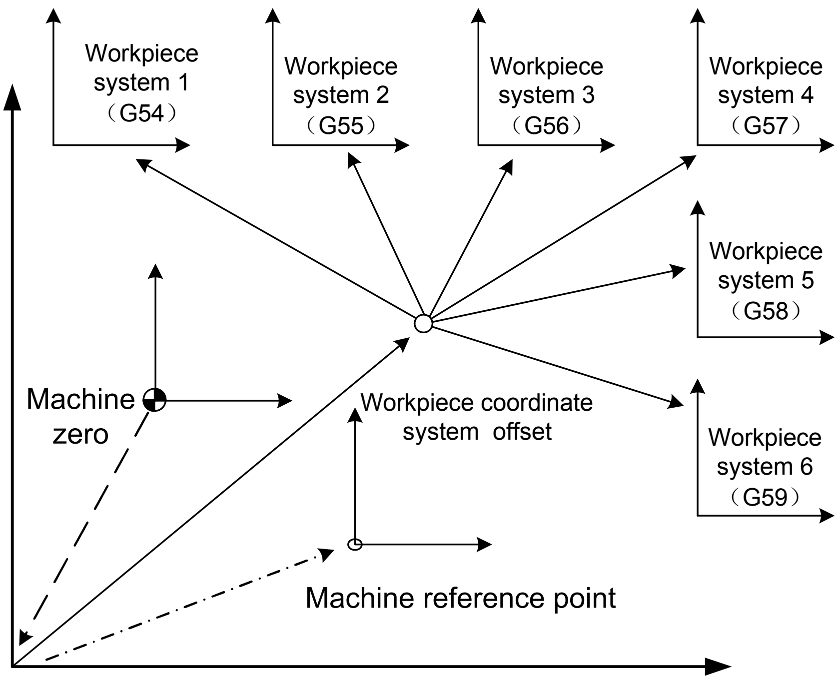


Fig. 4-2-6-1

As shown in Fig. **4-2-6-1**, after power-on, the machine returns to machine zero by manual zero return. The machine coordinate system is set up by the machine zero, which thus generates the

Ⅰ Programming

machine reference point and determines the workpiece coordinate system. The origins of these workpiece coordinate systems can be specified by inputting the coordinate offset in MDI mode or by setting data parameters **P15～P44**. These 6 workpiece coordinate systems are set up by the distances from machine zero to their respective coordinate system origins.

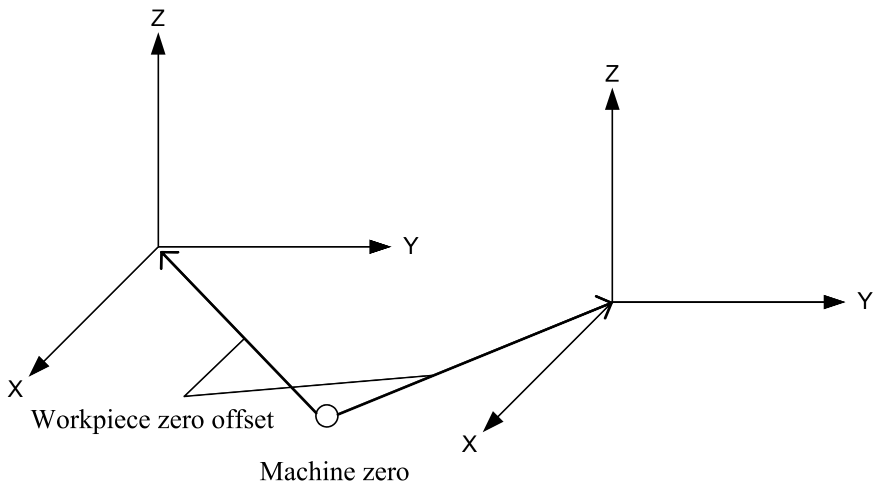


Fig. 4-2-6-2

Example： N10 G55 G00 X100 Z20；

N20 G56 X80.5 Z25.5；

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

4.2.7 Setting a Workpiece Coordinate System G50

1）Setting a workpiece coordinate system

Command format:：G50 X(U)_ Z(W)_；

Function: Set the work piece coordinate system. Two code parameters specify the absolute coordinate value of new work piece coordinate system at the tool nose on the current tool post. The code will not move the movement axis.

Explanation:

X: X absolute coordinate which current tool nose is at the work piece coordinate system;

Z: Z absolute coordinate which current tool nose is at the work piece coordinate system;

1. Once the coordinate system has been created, the positions of the following absolute value codes all represent those on the coordinate system until G50 code is used again to set the new coordinate system.
2. X direction is specified by diameter when the parameter is set to diameter programming, X is specified by radius when the parameter is set to radius programming.

Ⅰ Programming

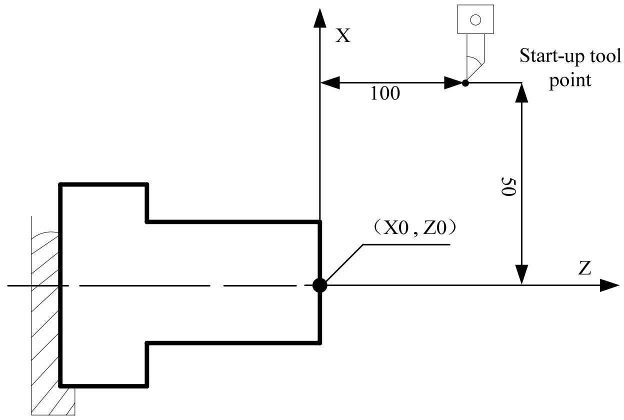


Fig. 4-2-7-1

As shown in Fig. **4-2-7-1**, after executing G50 X100 Z100 code, the work piece coordinate system is set as the figure and the position of the tool nose at the current work piece coordinate system is set. About the detailed coordinate setting methods, refer to the *tool-setting operation, Section* 4.1.5.

Note 1: G50 setting a coordinate system is executed at state of the tool offset cancel. The absolute coordinates after setting are G50's setting value; the tool offset cancel is performed at the tool offset cancel state: "T0100 G00 U0 W0" . Suppose that the current tool offset state is T0101.

An absolute coordinate display is divided into two conditions when G50 setting a coordinate system is done at state of tool offset:

A. When a tool offset is executed (a movement command exists after tool offset), absolute coordinates after setting are G50's setting values as follows:

Table 4-2-7-1

Program(executing the tool compensation by coordinate offset mode	Absolute coordinate's displayed value	No. 01 tool compensation value
G0 X0 Z0	X：0 Z：0	X：-12 Z：-23
T0101	X：12 Z：23	
G0 X0 Z0	X：0 Z：0	
G50 X20 Z20	X：20 Z：20	

B. When a tool offset is not executed (without a movement command after tool offset) including tool offset cancel and tool offset setting, absolute coordinates mirrors a tool offset value after setting as follows:

Table 4-2-7-2

Program(executing the tool compensation by coordinate offset mode	Absolute coordinate's displayed value	No. 01 tool compensation value
G0 X0 Z0	X：0 Z：0	X：-12 Z：-23
T0101	X：12 Z：23	
G0 X50 Z50	X：50 Z：50	

T0100	**X：38　Z：27**	
G50 X20 Z20	**X：8　Z：-3**	

Table 4-2-7-3

Program(executing the tool compensation by coordinate offset mode	**Absolute coordinate's displayed value**	**No. 01 tool compensation value**
G0 X0 Z0	**X：0　Z：0**	**X：-12 Z：-23**
T0101	**X：12　Z：23**	
G50 X20 Z20	**X：32　Z：43**	

2）Coordinate system translation

Command format: G50 U_ W_ ；

Function: According to the above codes, the tool nose position of the tool post in the original absolute coordinate system moves the distance specified by one parameter. That is to say, the new coordinate tool nose position relative to the original absolute coordinate system is: X+U，Z+W.

Note: When the parameter is set to the diameter programming, X direction is specified by the diameter; it is set to the radius programming, X is specified by radius.

4.2.8　Plane Selection G17/G18/G19

Format:G17/G18/G19

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

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

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

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

Example：G18 X_ Z_；　ZX plane

Prompt: The system supports the fixed cycle in only G18 currently. It's better to specify a plane at a corresponding block. In special, when many people share the same system, such can avoid mistaken programming to cause unexpected or abnormality.

4.2.9　Skip Function（G31）

Command format：　G31 X(U)_ Z(W)_ F_

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

Explanation:

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

Example:

The block after G31 is a single axis movement specified by incremental values, which is shown in Fig. 4-2-9-1 :

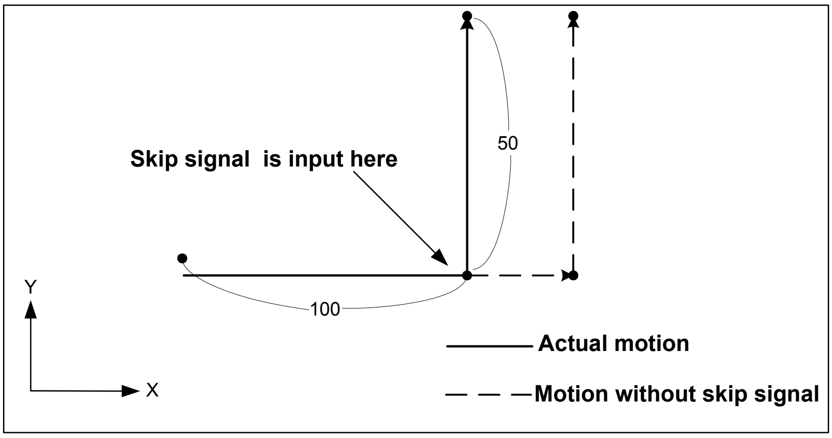


Fig. 4-2-9-1 The next block is the single-axis movement specified by incremental values

The next block after G31 is a single-axis movement specified by absolute values, which is shown in Fig. 4-2-9-2：

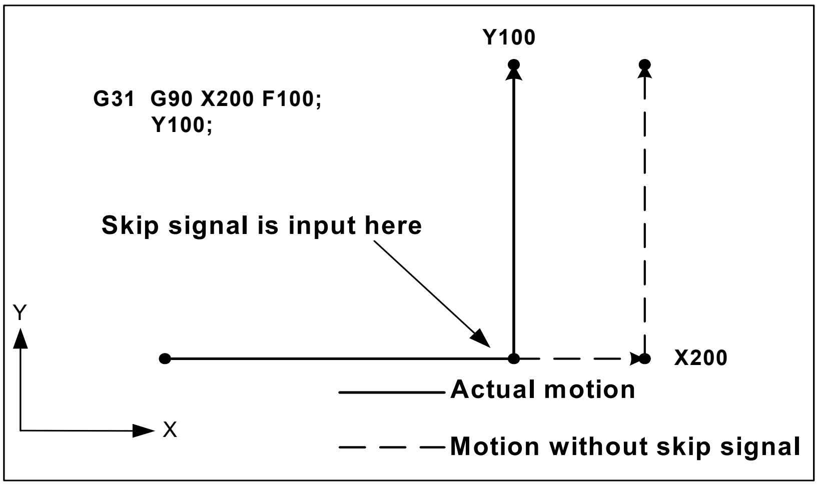


Fig. 4-2-9-2 The next block is a single-axis movement specified by absolute values

The next block after G31 is two-axis movement specified by absolute values, which is shown in Fig. 4-2-9-3:：

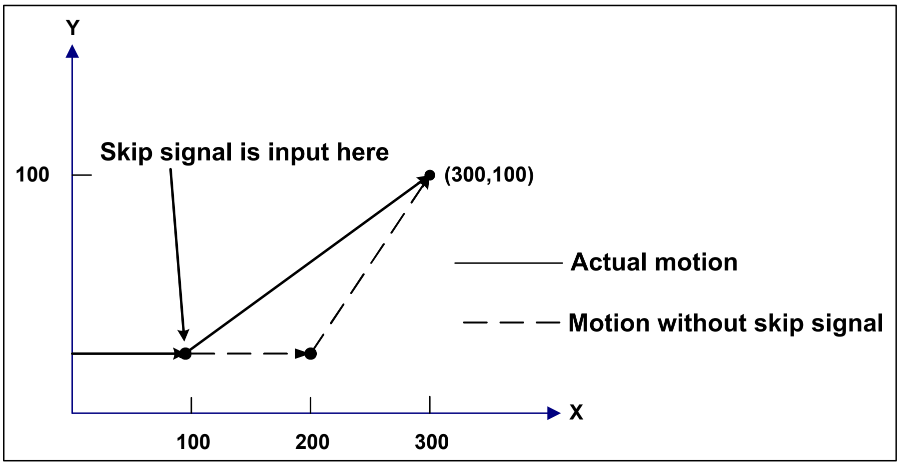


Fig. 4-2-9-3 The next block is two-axis movement specified by absolute value

Note: The setting can be done by bit parameter NO: 02#7 [skip signal SKIP, (0:1, 1:0)].

4.2.10 Inch/Metric Conversion G20/G21

Format: **G20**：inch input

G21：metric input

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

Explanation:

After inch/metric conversion, the units of the following values are changed: Feedrate specified by F code, position code, workpiece zero offset value, tool compensation value, scale unit of MPG and movement distance in incremental feeding.

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

Note：

1. When the inch input is converted to metric input or vice versa, the tool compensation value must be preset according to the least input incremental unit.
2. After inch input is converted to metric input or vice versa, for the first G28, the operation from the intermediate point is the same as that of manual reference point return.
3. When the least input incremental unit is different from the least code incremental unit, the maximum error is half of the least code unit and this error is not accumulated.
4. Program inch/metric input can be set by bit parameter N0:00#2.
5. Program inch/metric output can be set by bit parameter N0:00#1.
6. G20 or G21 must be specified in a separate block.

4.2.11 Optional Angle Chamfering/Corner Rounding

Command format: ，L_：chamfering

，R_：corner arc transition

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

Explanation：

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

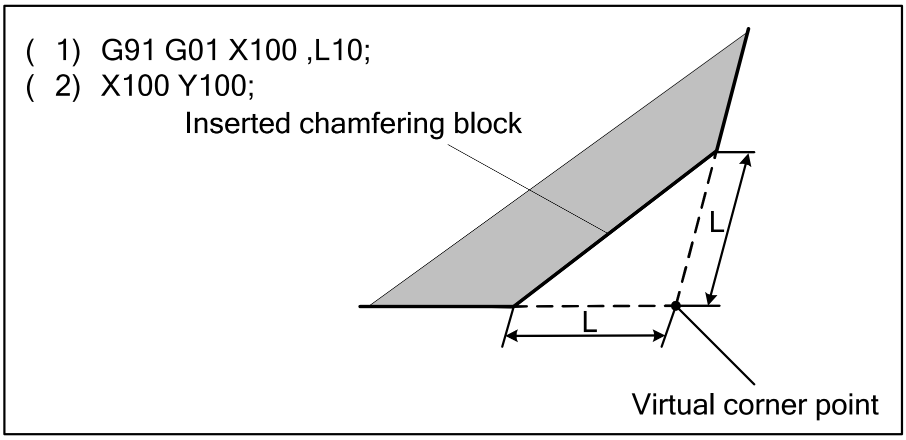


Fig. 4-2-11-1

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

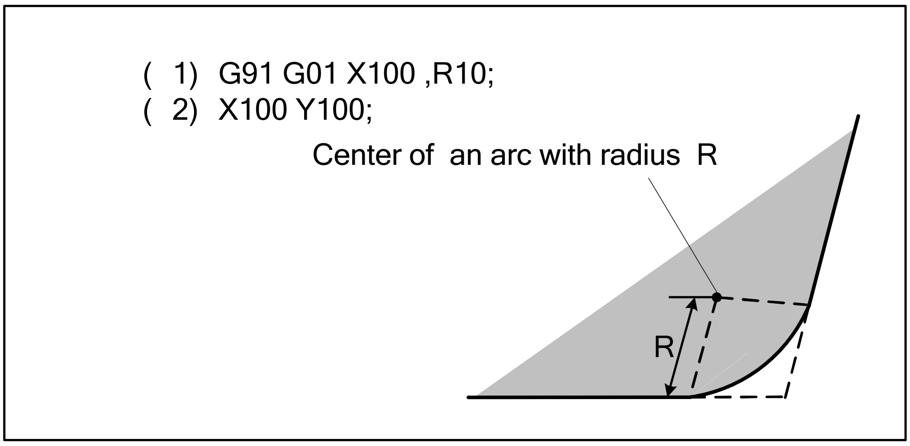


Fig. 4-2-11-2

Restrictions:

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

4.2.12 Constant Pitch Thread Cutting（G32）

Command format:G32 X(U)__Z(W)__ F（I）__ J__K__Q__；

Code function: two axes simultaneously execute the thread cutting (their path is shown in Fig.

4-2-12-1) from the start point (the position before G32 runs) to the end point specified by X(U), Z(W), Y(V). The code can execute the constant pitch straight thread and taper thread, and end thread.

Explanation:

X(U)：X absolute(U is an incremental programming code and is the tool movement distance) coordinate value of thread cutting end point;

Z(W)：Z absolute (W is an incremental programming code and is the tool movement distance) coordinate value of thread cutting end point;

F：Metric pitch. Namely it is movement amount of the tool opposite to the workpiece when the spindle rotates one rev, and its range: 0.001～500 mm, and it is a modal parameter;

I ：The tooth quantity/inch in the inch system, its range is 0.06 teeth/inch~~25400 teeth/inch, and it is a modal parameter;

J:：Stroke in the short axis in thread run-out , its range(-99999999~99999999）x the least input increment, its unit: mm with negative sign; if the short axis is X, its value is specified with the radius; J value is a modal parameter.

K：Length in the long axis in thread run-out, its range: 0~99999999x the least input increment, and its unit: mm./inch. If the long axis is X, its value is in radius without direction; K is a modal parameter.

Q：it is an initial angle defined to an offset angle between the spindle rotation one rev and starting point of thread cutting: 0～360 (unit: degree). Q is a non-modal parameter, must be defined, otherwise it is 0^0.

Q rules：

1. Its initial angle is 0 if Q is not specified;
2. For continuous thread cutting, Q specified by its following thread cutting block except for the first block is invalid, namely Q is omitted even if it is specified;
3. Multi threads formed by initial angle is not more than 65535;
4. Unit: 1^0 . Q180 is input in program if it offsets 180^0 with spindle one rev. it can be used to multi-head thread cutting. The system automatically counts the thread initial angle according to the thread head quantity.

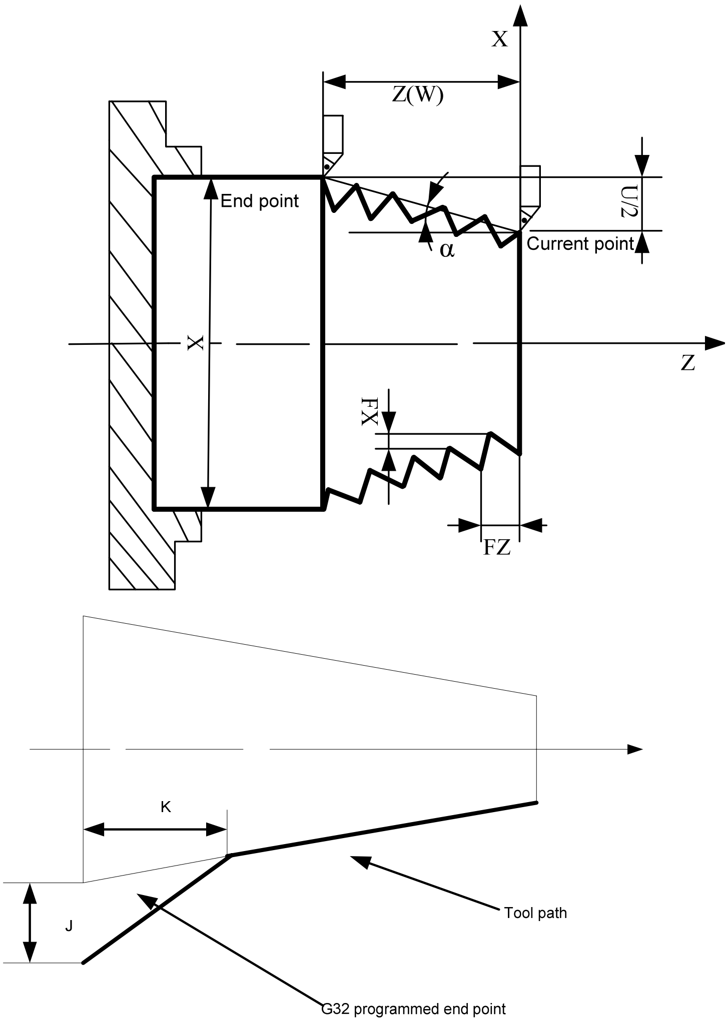


Fig. 4-2-12-1

The system uses long-short axis and its calculation method is shown below in Fig. **4-2-12-2.**

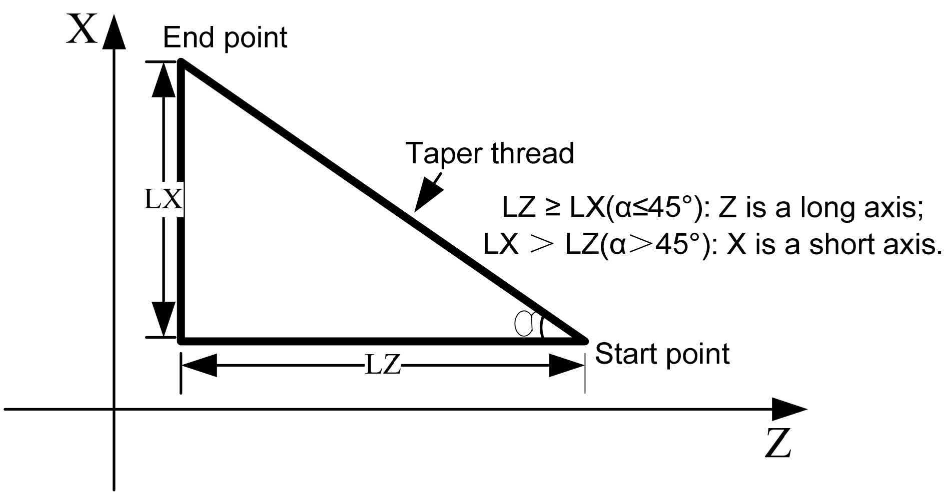


Fig. **4-2-12-2**

Notes:

1. At the beginning and ending part of thread cutting, it will result in the incorrect lead due to the lifting speed. Considering the factor, the length of commanded thread is longer than

that of the required thread, which is shown in Fig. 4-2-12-3.

2. During thread cutting, the feedrate override is not valid, and keeps at 100%.
3. During thread cutting, the spindle override is not valid, because once the spindle override is changed, it will result in the incorrect thread due to lifting speed.
4. After the feed hold is executed, the system displays “Feed hold” and the thread cutting continuously executes not to stop until the current block is executed completely; when the continuous thread cutting is executed, the program run pauses after thread cutting blocks are executed completely.
5. In Single block, the program stops run after the current block is executed. The program stops run after all blocks for thread cutting are executed.
6. When the previous block is for thread cutting and the current block is the same, the system does not test the spindle encoder signal per rev at starting when the thread is started.
7. The spindle speed must be constant. Thread errors occur when the spindle speed changes.
8. An alarm occurs when F, I are in the same block.
9. J, K are modal. T J, K must not be specified in the block and is done in the last block in continuous thread cutting. Their modes are cancelled when no thread cutting is executed;
10. There is no thread run-out when J, or J, K is omitted; K=J: a thread run-out is executed when K is omitted;
11. There is no thread run-out when J=0 or J=0, K=0;
12. When K=0 or it is omitted, J=K: a thread run-out is executed.

Example 1：using G32 compiles a program shown in Fig. **4-2-12-3**，and the thread pitch：4mm.

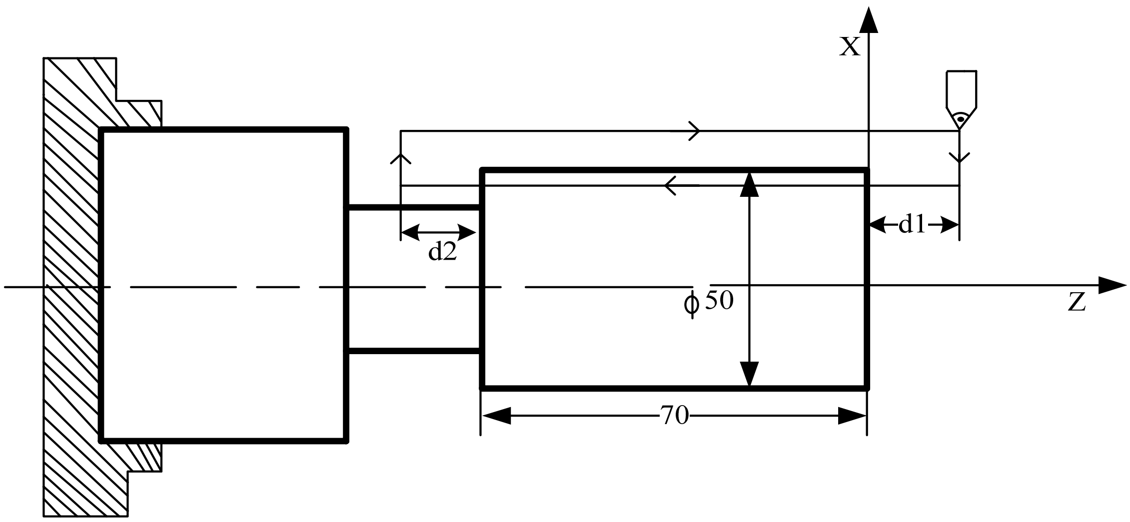


Fig. 4-2-12-3

Example: d1 = 3mm，d2 = 1.5mm, total cutting depth 1mm with two times cut-in.

G0 X100 Z50；（rapid positioning）

M03 S200；（start the spindle, the speed 200）

T0101； （call the thread tool）

G0 X49 Z3；（rapid positioning, the 1st time cut-in 1mm）

G32 W-74.5 F4.0；

G00 X55；

W74.5；
X48；　　　　（rapid positioning, the 2nd time cut-in 1mm）
G32 W-74.5 F4.0；
G00 X55
W74.5；
G0 X100 Z50 M05；
M30；

Example: Use G32 code to program shown as Fig. **4-2-12-4**. The long axis is Z axis, and the thread lead is 3mm.

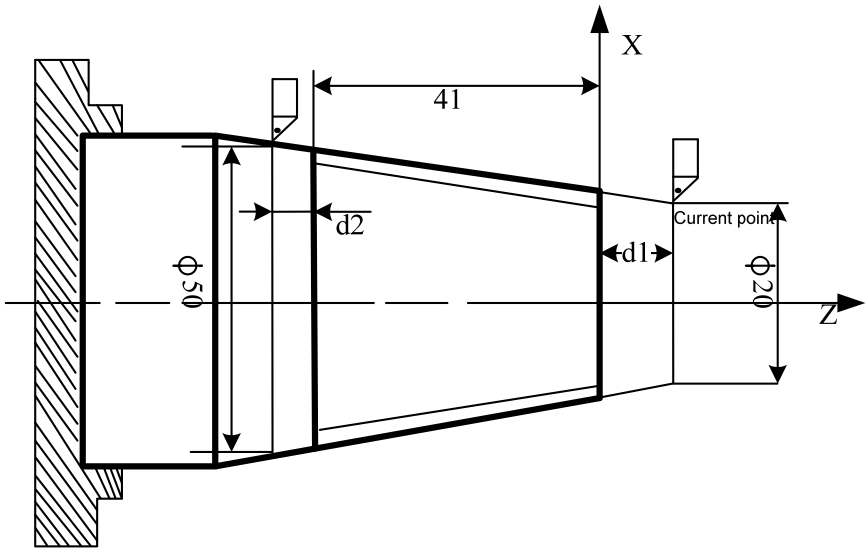


Fig. 4-2-12-4

Example: d1 = 2mm，d2 = 1mm, total cutting depth 1mm(one-sided) with two times cut-in.

G0 X100 Z50；（rapid positioning）
M03 S200；（start the spindle, the speed 200）
T0101；　（call the thread tool）
G00 X19 Z2；　　　　（rapid positioning, the 1st time cut-in 1mm）
G32 X49 Z-41 F3；
G00 X55；
Z2；
G0 X18；　　　　　（rapid positioning, the 2nd time cut-in 1mm）
G32 X48 Z-41 F3；
G0 X55；
Z2；
G0 X100 Z50 M05；
M30；

4.2.13 Variable Pitch Thread Cutting（G34）

Command format：G34 X(U)__Z(W)__ F（I）__J__ K__R___Q__；

Function：two axes simultaneously execute the thread cutting from the start point (the position before G34 runs) to the end point specified by X(U), Z(W), Y(V). The code can execute the variable pitch straight thread and taper thread, and end thread.

Explanation:

X(U)：X absolute (U is an incremental programming code and is the tool movement distance) coordinate value of thread cutting end point;

Z(W)：Z absolute (W is an incremental programming code and is the tool movement distance) coordinate value of thread cutting end point;

F：Metric pitch. It is a pitch of the thread start point, its range: 0.001～500 mm, and it is a modal parameter;

I ：The tooth quantity/inch in the inch system, its range is 0.06 teeth/inch~~25400 teeth/inch, and it is a modal parameter;

J:：Stroke in the short axis in thread run-out, with negative sign; if the short axis is X, its value is specified with the radius; J value is a modal parameter.

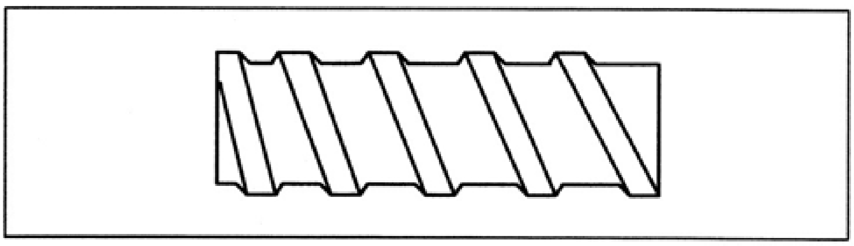
K：Length in the long axis in thread run-out, its range: 0~99999999x the least input increment, and its unit: mm./inch. If the long axis is X, its value is in radius without direction; K is a modal parameter.

R：When R value's increasing or reducing makes the pitch exceed its permissive value or the pitch reduces to 0 or a negative value, an alarm occurs; simultaneously, when the pitch change is big, acceleration/deceleration is slow during thread machining maybe cause an mistaken pitch.

Q：it is an initial angle defined to an offset angle between the spindle rotation one rev and starting point of thread cutting: 0～360 (unit: 1 degree). Q is a non-modal parameter, must be defined, otherwise it is 0^0.

Q rules：

1. Its initial angle is 0 if Q is not specified;
2. For continuous thread cutting, Q specified by its following thread cutting block except for the first block is invalid, namely Q is omitted even if it is specified;
3. Unit : 0.001^0 . Q180 is input in program if it offsets 180^0 with spindle one rev. it can be used to multi-head thread cutting. The system automatically counts the thread initial angle according to the thread head quantity.（see Example 3）



Programming

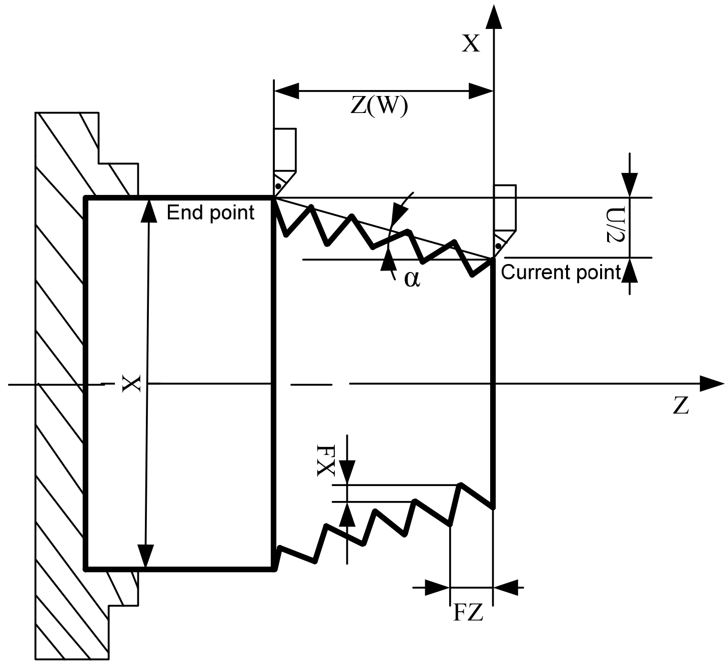
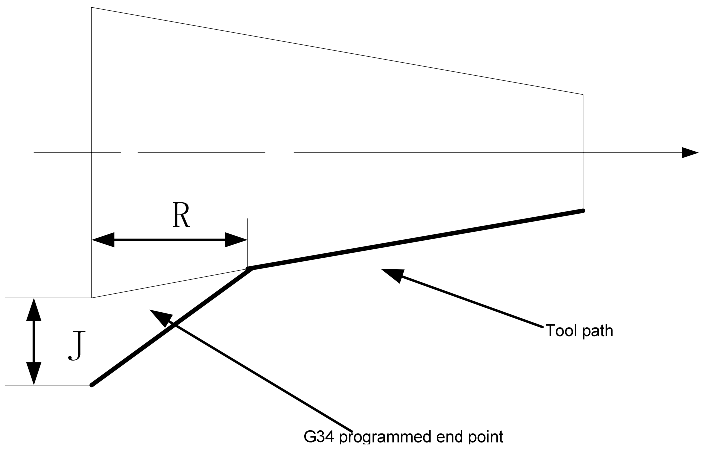


Fig. 4-2-13-1



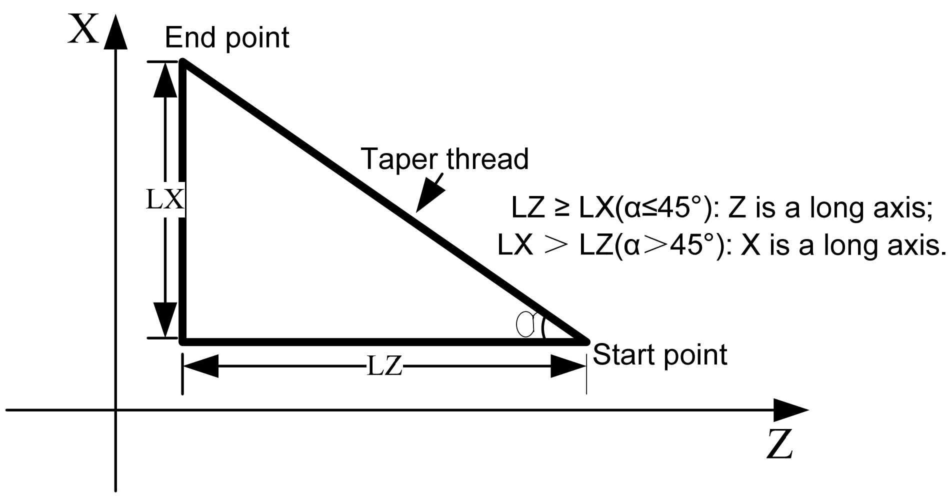


Fig. **4-2-13-2**

Notes：

1. **At the beginning and ending part of thread cutting, it will result in the incorrect lead due to the lifting speed. Considering the factor, the length of commanded thread is longer than that of the**

required thread, which is shown in Fig. 4-2-13-3.

2. During thread cutting, the feedrate override is not valid, and keeps at 100%.

3. During thread cutting, the spindle override is not valid, because once the spindle override is changed, it will result in the incorrect thread due to lifting speed.

4. After the feed hold is executed, the system displays "Feed hold" and the thread cutting continuously executes not to stop until the current block is executed completely; when the continuous thread cutting is executed, the program run pauses after thread cutting blocks are executed completely.

5. In Single block, the program stops run after the current block is executed. The program stops run after all blocks for thread cutting are executed.

6. When the previous block is for thread cutting and the current block is the same, the system does not test the spindle encoder signal per rev at starting when the thread is started.

7. The spindle speed must be constant. Thread errors occur when the spindle speed changes.

8. An alarm occurs when F, I are in the same block.

9. J, K are modal J, K must not be specified in the block and is done in the last block in continuous thread cutting. Their modes are cancelled when no thread cutting is executed;

10. There is no thread run-out when J, or J, R is omitted; R=J: a thread run-out is executed when K is omitted;

11. There is no thread run-out when J=0 or J=0, R=0;

12. When R=0 or it is omitted, J=R: a thread run-out is executed.

Example 1：using G34 compiles a program shown in Fig. **4-2-13-3**，and the thread pitch: 4mm.

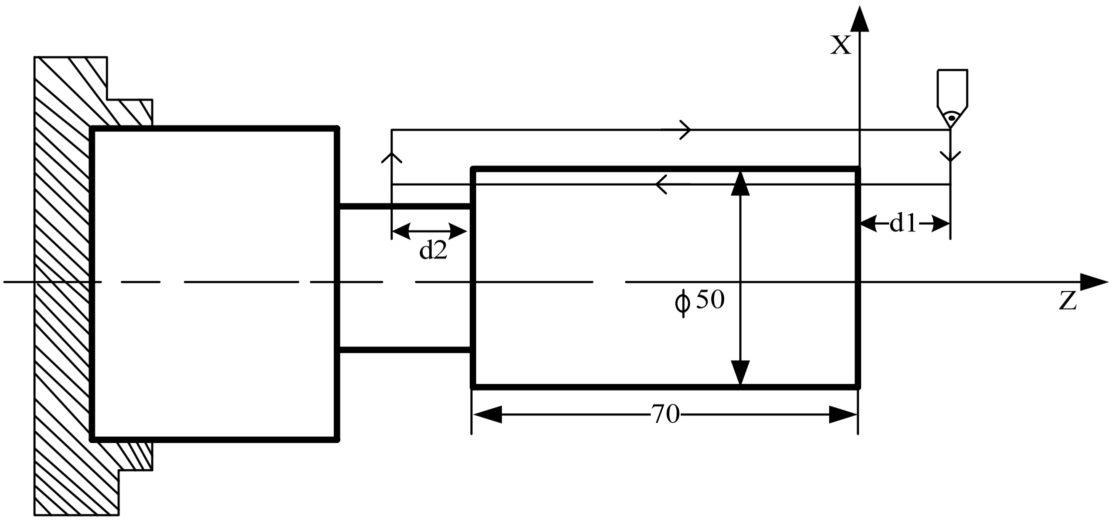


Fig. 4-2-13-3

Example: d1 = 3mm，d2 = 1.5mm, total cutting depth 1mm (one-sided) with two times cut-in.

G0 X100 Z50；（rapid positioning）

M03 S200；（start the spindle, the speed 200）

T0101；　（call the thread tool）

G0 X49 Z3；（rapid positioning, the 1st time cut-in 1mm）

G34 W-74.5 F4.0；

G00 X55；

W74.5；

X48；　　　（rapid positioning, the 2nd time cut-in 1mm）

G34 W-74.5 F4.0；

G00 X55

W74.5;
G0 X100 Z50 M05;
M30;

Example: Use G34 code to program shown as Fig. **4-2-13-4**. The long axis is Z axis, the thread lead is 3mm.

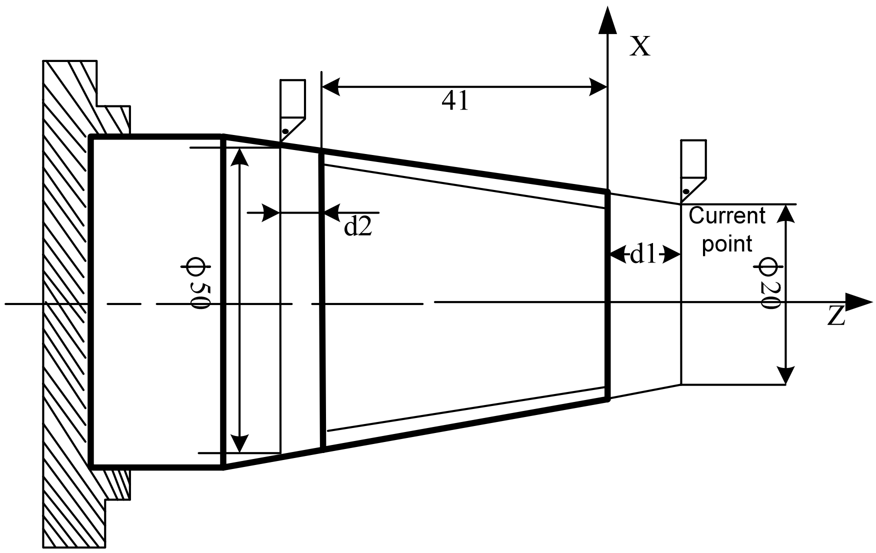


Fig. 4-2-13-4

Example: d1 = 2mm, d2 = 1mm, total cutting depth 1mm with two times cut-in.

G0 X100 Z50; (rapid positioning)
M03 S200; (start the spindle, the speed 200)
T0101; (call the thread tool)
G00 X19 Z2; (rapid positioning, the 1st time cut-in 1mm)
G34 X49 Z-41 F3;
G00 X55;
Z2;
G0 X18; (rapid positioning, the 2nd time cut-in 1mm)
G34 X48 Z-41 F3;
G0 X55;
Z2;
G0 X100 Z50 M05;
M30;

Example: Use G34 code to compile 2-head thread shown programs as Fig. **4-2-13-5** and the thread lead is 2mm.

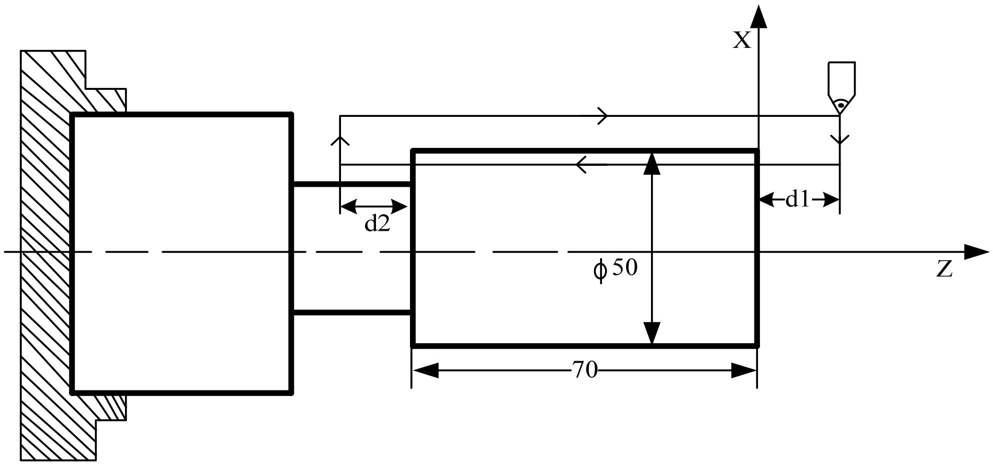


Fig. 4-2-13-5

Example: d1 = 3mm，d2 = 1.5mm, total cutting depth(one-sided) 1mm with two times cut-in.

G0 X100 Z50；（rapid positioning）

M03 S200；（start the spindle, the speed 200）

T0101；　（call the thread tool）

G0 X49 Z3；（rapid positioning, the 1st head's 1st time cut-in 1mm）

G34 W-74.5 F4.0Q0；

G00 X55；

W74.5；

G0 X49 Z3；（rapid positioning, the 2nd head's 1st time cut-in 1mm）

G34 W-74.5 F4.0Q180；

G00 X55；

W74.5；

X48；　　　　（rapid positioning, the 1st head's 2nd time cut-in 1mm）

G34 W-74.5 F4.0Q0；

G00 X55

W74.5；

G0 X48 Z3；（rapid positioning, the 2nd head's 2nd time cut-in 1mm）

G34 W-74.5 F4.0Q180；

G00 X55

W74.5；

G0 X100 Z50 M05；

M30；

4.3　Reference Point G Code

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

There are 3 codes for the reference point, as is shown in Fig. **4-3-1**. The tool can be automatically moved to the reference point via an intermediate point along a specified axis by G28; or

Ⅰ Programming

be moved automatically from the reference point to a specified point via an intermediate point along a specified axis by G28.

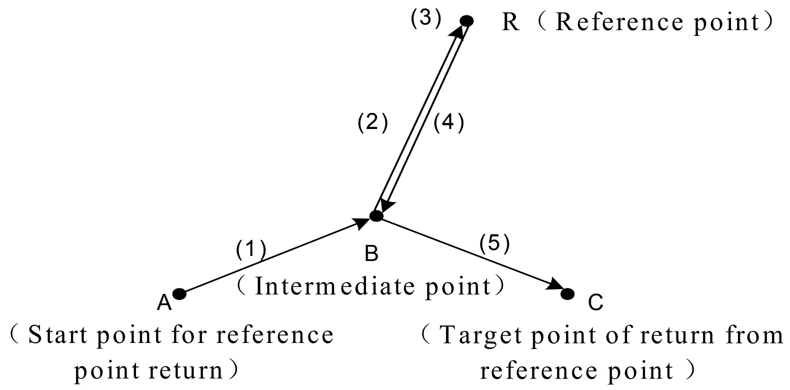


Fig. 4-3-1

4.3.1 Automatic Return to Machine Zero（G28）

Command format:：G28 X（U）_ Z（W）_ ；

Function: the axis specified by the code moves to the middle point defined by X(U), Z(W) from start point and then return to the machine zero. One or two axes can be commanded in the code.

Table **4-3-1-1**

Code	Function
G28 X（U）	X returns to machine zero and Z remains in the previous position
G28 Z（W）	Z returns to machine zero and X remains in the previous position
G28	Remain in previous position（No. 166 alarm occurs）
G28 X（U）___Z（W）	X,Z simultaneously returns to the machine zero

Explanation: the code execution process（see Fig.**4-3-1-1**）：

（1）Rapid traverse to middle point of specified axis from current position(A point→B point) ;

（2）Rapid traverse to reference point from the middle point(B point→R point) ;

（3）If the machine is not locked, LED is ON when the machine zero return is completed.

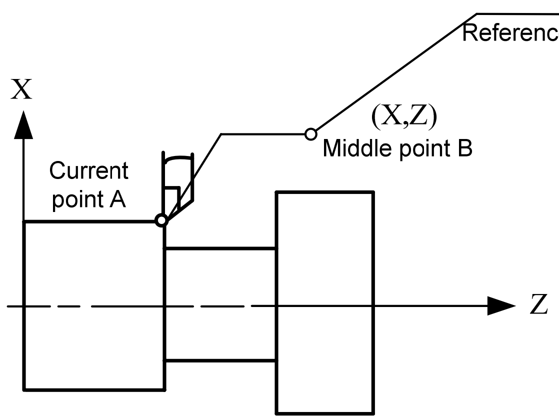


Fig. 4-3-1-1

Note 1: After the power supply is turned on, when the manual machine zero is not executed one time, and G28 is code, the motion from the middle point to the machine zero is the same as the manual machine zero return.

Note 2: The two axes position at the respectively rapid traverse speed from A to B and from B to R, and so the path is not always a straight line.

Note 3: Do not use the function and the machine zero is not installed on the machine.

4.4 Simple Canned Cycle G Code

In some special roughing, due to the large cutting amount, the same machining path should be repeated for many times. Then, the fixed cycle function can be used, that is one block can realize the machining which is normally commanded by many blocks. Moreover, during repeated cutting, just the corresponding numerical value is rewritten, which is very useful to simplify the program. The single cycle codes include the outer/inter circle cutting cycle G90, thread cutting cycle G92 and end face cutting cycle G94.

In the following explanatory figure, it is specified by diameter. When the radius specifies, replace U with U/2, and X with X/2.

4.4.1 Axial Cutting Cycle（G90）

Command format：G90 X（U）__ Z（W）__ R__ F__；

Function: Executing the code can realize the single cycle machining of the cylinder and conical faces. The cycle completes and the tool comes back to the start position. The broken line（R） means the rapid traverse and the full line（F）means the cutting feed shown in figures 4-4-1-1 and 4-4-1-2. In increment programming, the codes of the numerical value after address U depend on X direction of path1, the codes after address W are set by Z direction of path 2.

Explanation:

X, Z: Absolute coordinate value of cycle end point, unit: mm;

U, W: Coordinate of the cycle end point corresponding to the cycle start point, unit: mm;

R: Radius difference between the conical face cutting's start point and its end point, unit: mm;

F: The compound feedrate of X and Z axes in cycle, and it is a modal code.

Ⅰ Programming

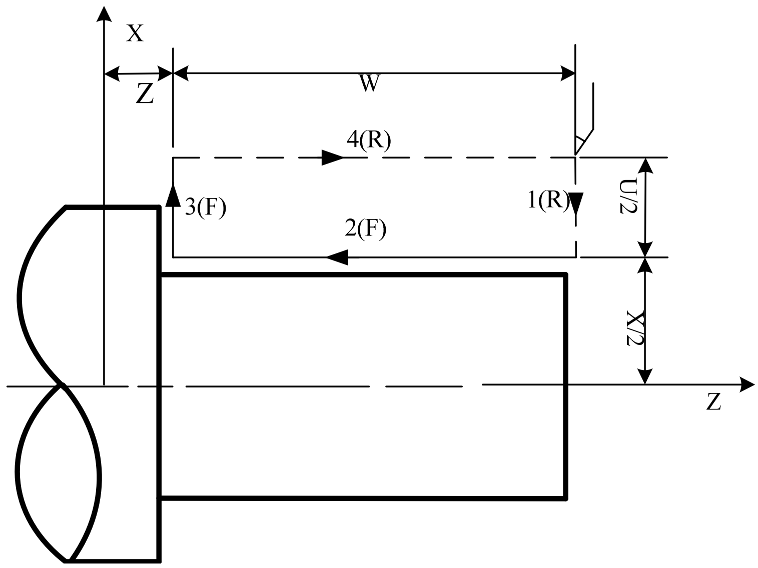


Fig. 4-4-1-1

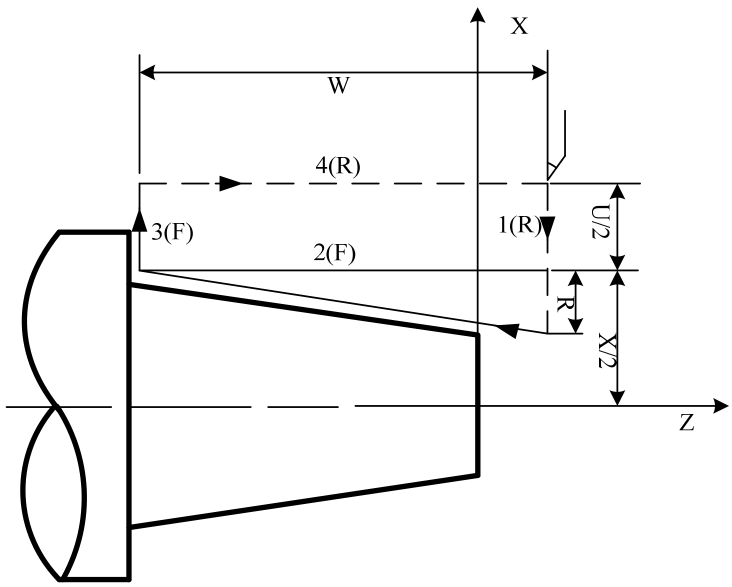


Fig. 4-4-1-2

According to the different tool start-up positions, G90 code has four paths, which is shown in Fig.**4-4-1-3.**

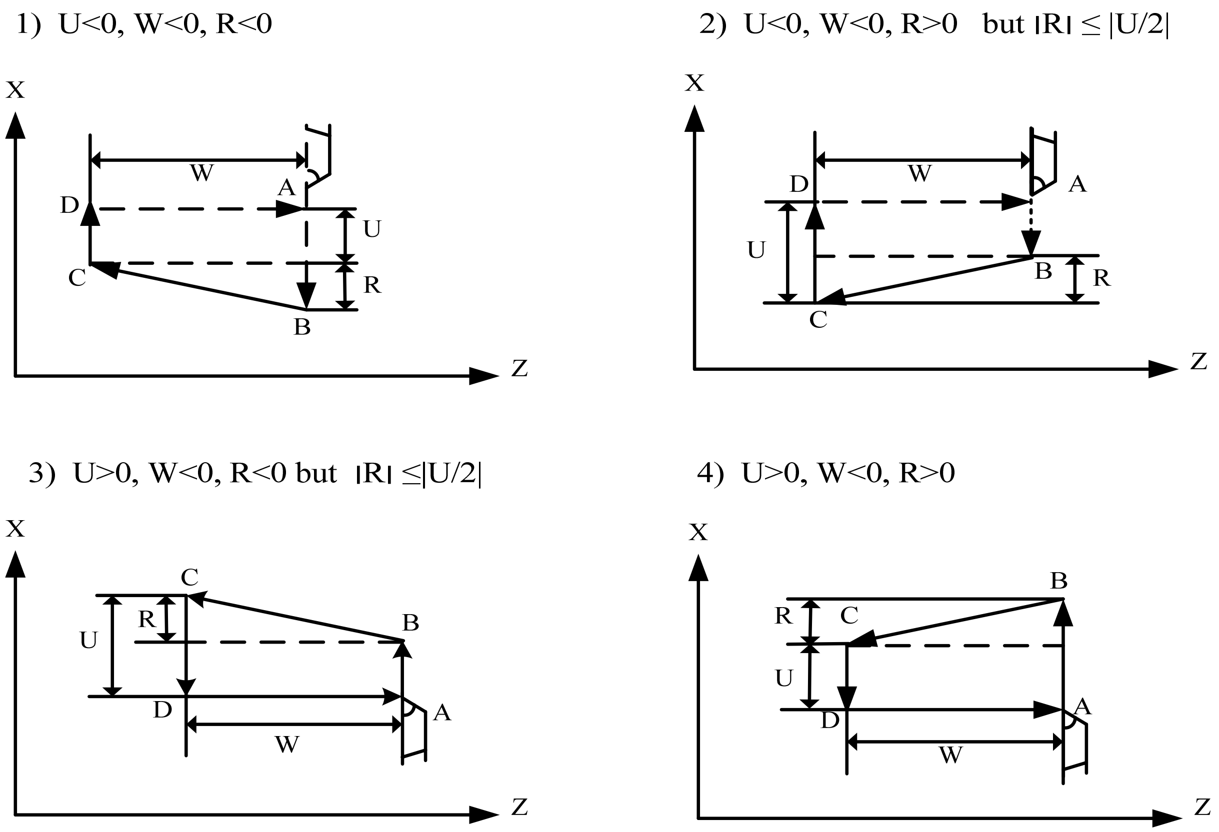


Fig. 4-4-1-3 G90 code path

For example: The part program is edited by G90 code shown in Fig. **4-4-1-4**.

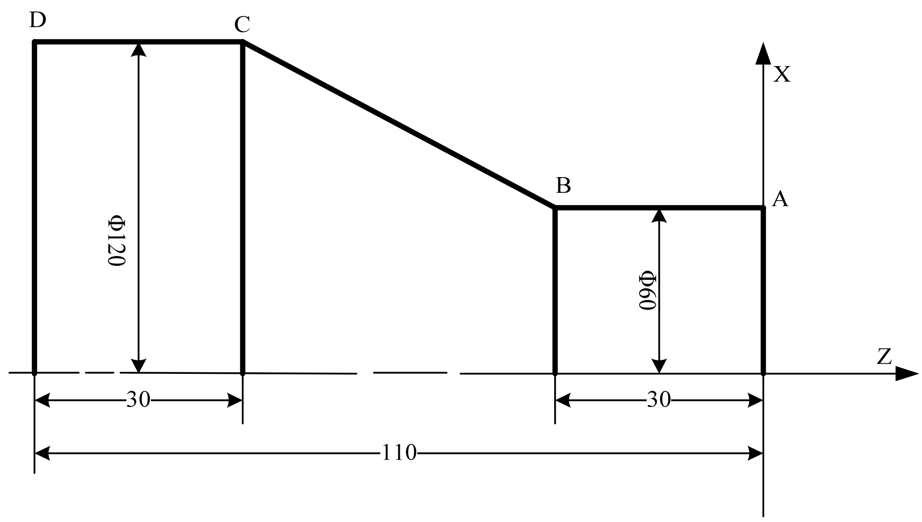


Fig. 4-4-1-4

Program:

O0001;

M3 S300;

G0 X130 Z5;

G90 X120 Z-110 F200；（C→D）
X60 Z-30；（A→B）
G0 X130 Z-30；
G90 X120 Z-80 R-30 F150；（B→C）
M5 S0；
M30；

4.4.2 Radial Cutting Cycle（G94）

Command format：G94 X（U）__ Z（W）__ R__ F__；

Function: When the code is executed, the end single cycle machining can be performed, after the cycle completes, the tool returns to the start position.。

R means rapid traverse and F means cutting feed in Fig. 4-4-2-1 and 4-4-2-2. In increment programming, signs of numerical value after address U depend on X direction of path 2, signs after address W by Z direction of path 1.

Explanation：

X , Z: Absolute coordinate value of cycle end point, unit: mm;
U, W: Coordinate of cycle end point corresponding to cycle start point, unit: mm;
R: Z coordinate vector of the end face cutting from start point to end point, unit: mm;
F: Compound feedrate of X and Z axes in cycle, a modal code

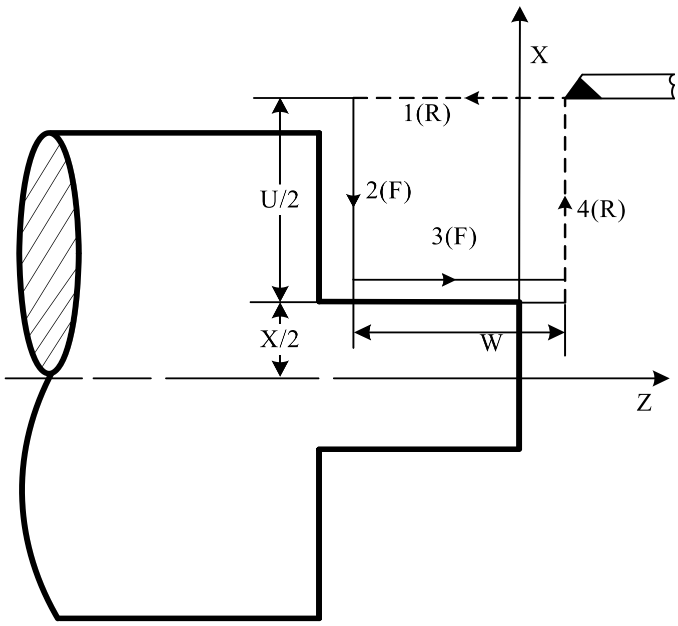


Fig. 4-4-2-1

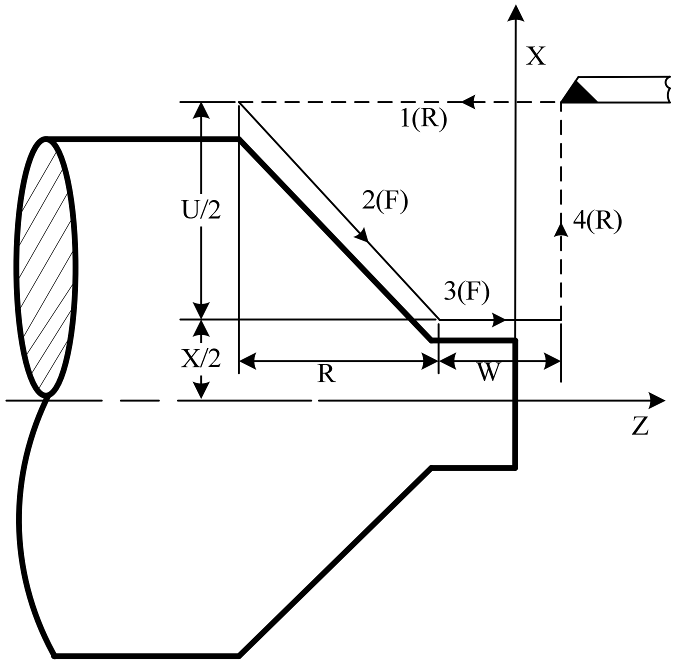
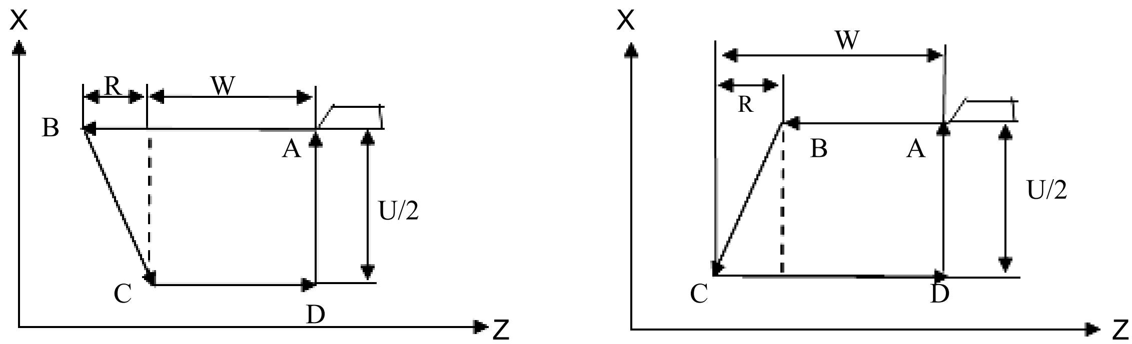


Fig. 4-4-2-2

According to the different tool start-up positions, there are four paths in G94 code, which is shown in Fig. 4-4-2-3:

1) U<0，W<0，R<0

2) U<0，W<0，R>0(│R│≤│W│)



3) U>0，W<0，R>0 (│R│≤│W│)

4) U>0，W<0，R<0

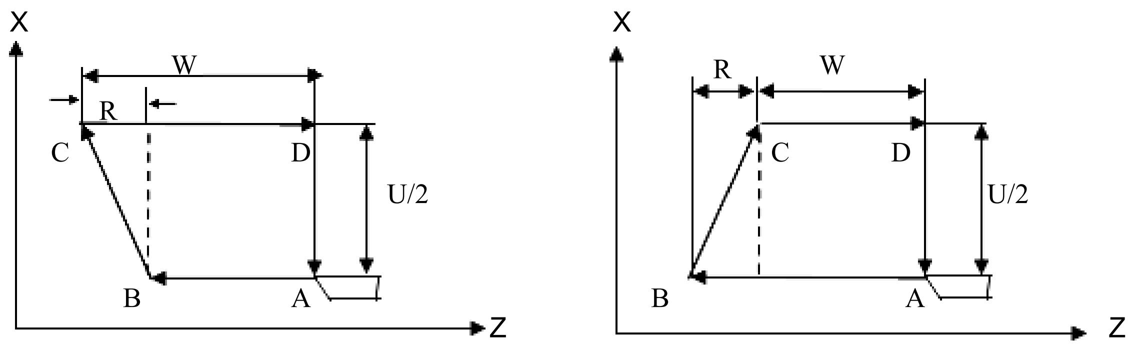


Fig. 4-4-2-3 G94 code path

Programming

Example: The part program is compiled by G94 code shown in **4-4-1-4**.

Program：

O0002；

M3 S1；

G0 X130 Z5；

G94 X120 Z-110 F100；（D→C）

G0 X120 Z0；

G94 X60 Z-30 R-50；（C→B→A）

M5 S0；

M30；

4.4.3 Thread Cutting Cycle（G92）

Format:

G92X(U)__ Z(W)__ J__K__ F__ L__ ；（Metric thread）

Specified thread lead (F)

G92X(U)__ Z(W)__ J__K__ I __ L__ ；（Inch thread）

Specified thread lead (teeth /inch)

Function: Executing the code can process the straight, taper single cycle with a constant lead, then the cycle completes and the tool returns to the start point. During thread cutting, it does not need the tool retraction grooving. J, K is separate X, Z run-out length. In Fig.4-4-3-1, Fig.4-4-3-2, the dotted line (R) means the rapid traverse, and the solid line (F) means the cutting feed. When J, K sets its value, the system executes X, Z thread run-out based on the setting value of J and K; when only J or K value is set, the system executes the thread run-out by 45°, When the user does not require J and K to set the run-out length, the run-out length=the value set by data parameter P473 X0.1 X pitch is executed by the system. When is omitted, K=J thread run-out is performed; J=0 or J=0, K=0, there is no thread run-out; when J≠0 and K=0, K=J thread run-out is performed; there is no thread run-out, J0, K≠0 can be set.

Explanation:

X, Z: Coordinate value of cycle end point, unit: mm;

U, W: Coordinate of cycle end point corresponding to cycle start point, unit: mm;

J: X run-out length, without sign. Its range: 0~9999, its unit: mm, J is specified by radius;

K: Z run-out length, without sign. Its range: 0~999999, its unit: mm;

R: Radius difference of thread start point and end point, unit: mm;

F: The thread lead in metric system, the range is 0.001~~500, unit: mm, mode code;

I: Number of teeth/inch of inch system thread, the range is 0.06~~25400, unit: teeth/inch, mode code;

L: Number of thread head, its range is 1~360, unit: head, modal code: it is default to 1 without specification;

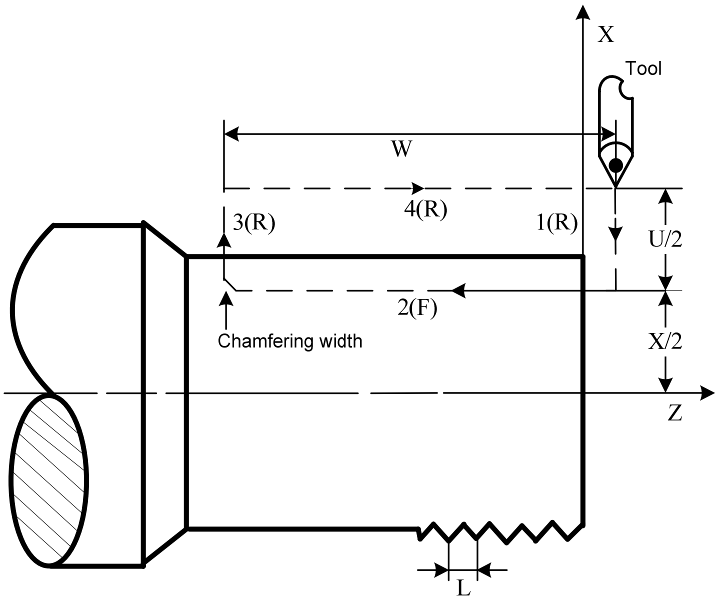


Fig. **4-4-3-1**

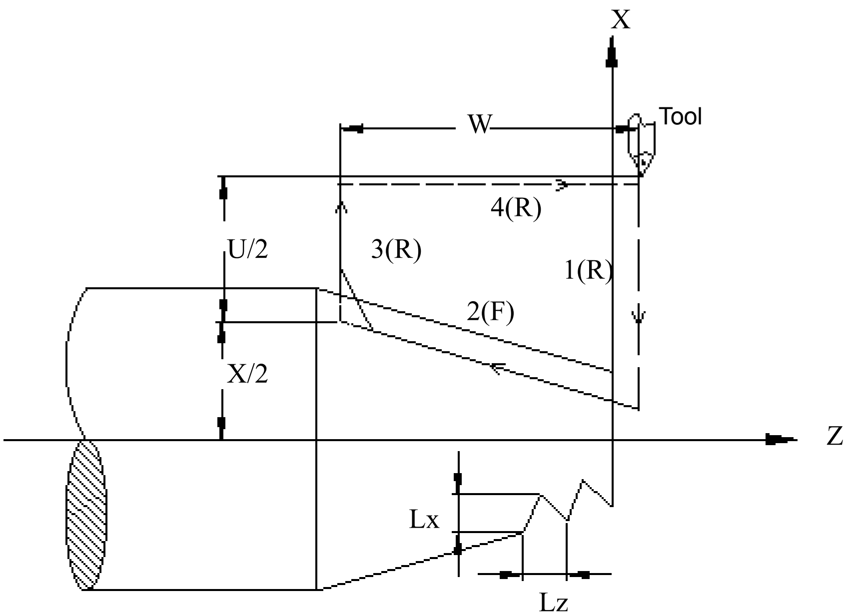


Fig. **4-4-3-2**

Note:

1. Notes of thread cutting are the same as those of G32 thread cutting, refer to **Section 4.2.13**;
2. If the feed hold signal (dwell) is input in thread cutting cycle, the cycle continues until the movement 3 completes, and then it stops;
3. The thread lead range and spindle speed limitation are same as those of G32 thread cutting;
4. When G92 processes the straight thread, the tool start-up point of G92 is same as the thread end point in X direction, it will alarm because the inner or outer thread can not be differed;
5. About the range of R value in G92, refer to Fig. 4-4-1-3;
6. When one of J, K is set to 0, or they are not specified, 45°thread run-out is performed;
7. In MDI mode, I address value can be input but its relevant value is not displayed. Do not run G92 in MDI mode.

Example: Firstly, the part program is edited by G90 code shown in Fig. 4-4-3-3, and G92 machines a thread.

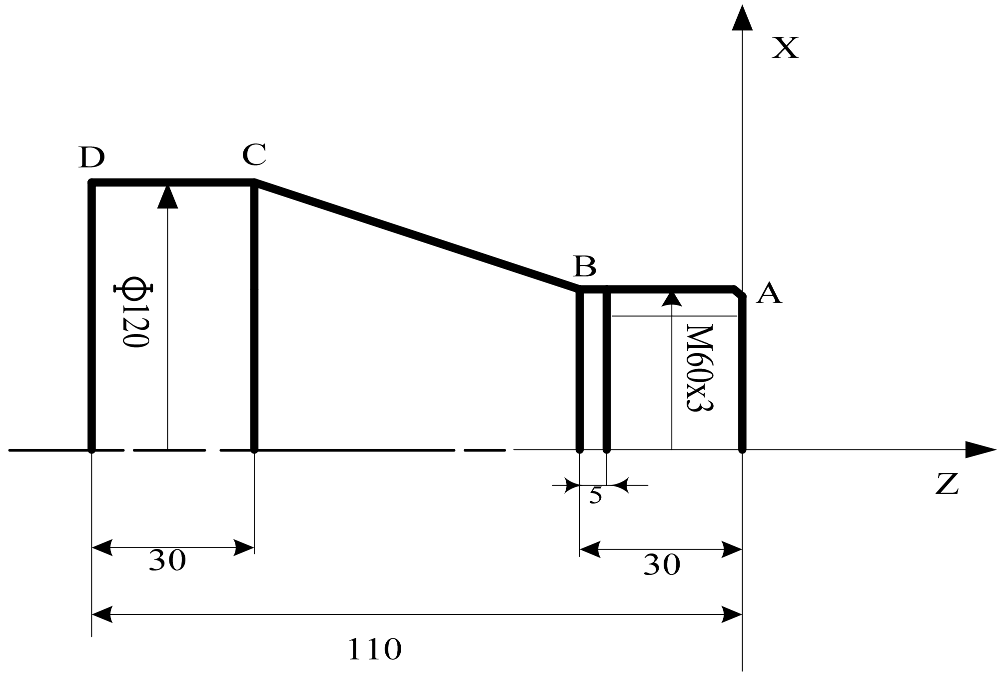


Fig. **4-4-3-3**

Program：

O0001；

M3 S300；

G0 X150 Z50；

T0101；　　　　　　　　（Outer turning tool）

G0 X130 Z5；

G90 X120 Z-110 F200；　（C→D）

X60 Z-30；（A→B）

G0 X130 Z-30；

G90 X120 Z-80 R-30 F150；（B→C）

G0 X150 Z150；

T0202；　　　　　　　　（Thread tool）

G0 X65 Z5；

G92 X58.5 Z-25 F3；　　（Machining thread, cutting divided into 4 times）

X57.5 Z-25；

X56.5 Z-25；

X56 Z-25；

M5 S0；

M30；

4.4.4　Notes of Single Fixed Cycle Codes

1）In the single fixed cycle, data X (U), Z (W) and R are modal values. When new X (U), Z (W) and R are not specified , the previous code data are valid;

2）In the single fixed cycle, X (U), Z(W) and R are cleared when the system specifies the non-modal G code besides G04 or other codes in Group 01 besides G90, G92 or G94;

3）There is only the block without motion codes after G90, G92 or G94, the system does not repeatedly execute the fixed cycle.

（Example）　N003 M3；

…

…

N010 G90 X20.0 Z10.0 F2000；

N011 M8；（do not repetitively execute G90）

…

…

4）　In the fixed cycle state, if M, S and T are commanded, then, the fixed cycle can process with M, S and T functions at the same time. Like the following examples, when the fixed cycle is cancelled by accident due to the codes of G00 and G01 after codeing M, S and T, please code the fixed cycle again.

（Example）　N003 T0101；

…

N010 G90 X20.0 Z10.0 F2000；

N011 G00 T0202；

N012 G90 X20.5 Z10.0；

4.5　Compound Fixed Cycle Code

To simplify programming, multiple cycle codes of the system includes axial roughing cycle G71, radial roughing cycle G72, closed cutting cycle G73, finishing cycle G70, axial grooving multiple cycle G74, axial grooving multiple cycle G75 and multiple thread cutting cycle G76. When the finishing path and the cutting depth of roughing are specified , the system automatically counts the cutting path and machining times.

4.5.1　Axial Roughing Cycle（G71 type Ⅰ）

Command format： G71u（Δd）R（e）；

G71 P（NS） Q（NF） U（Δu） W（Δw）F　S　T　；

N（NS）G0/G1 X（U）．．．．．．；

．．．．．．．．．；

．．．．F；

．．．．S；　　blocks for finishing path

．．．．T；

．

N（NF）．．．．．．；

Function: According to the finishing path, cutting depth, tool infeed and tool retraction amount given by blocks NS～NF , the system automatically counts the path of roughing, which is shown in 4-5-1-1. the tool cuts the workpiece in paralleling with Z. The code is applied to the formed roughing of non-formed rod.

Explanation：

Δd: Cut depth every time without sign. Cut-in direction is determined by AA'（specified by radius），its range: 0.001mm～99999.999 mm. It is a modal code and is valid till it is specified next time. P463 can set it. The parameter value can be changed according to the program code.

e：Tool retraction amount (specified by radius), unit: mm. Its value: 0mm～99999.999mm. it is a modal code and is valid till it is specified next time. P464 can set it. The parameter value can be changed according to the program code.

NS：Block number of the first block of finishing path.

NF：Block number of the last block of finishing path.

Δu：X finishing allowance's distance and direction. Its value range: -999999.999～999999.999mm.

Δw：Z finishing allowance's distance and direction. Its value range: -999999.999～999999.999mm.

F：cutting feedrate，its range: feed per minute 1mm/min～6000mm/min，feed per rev 0.001mm/r～500mm/r.

S：Spindle speed.

T：Tool, tool offset number.

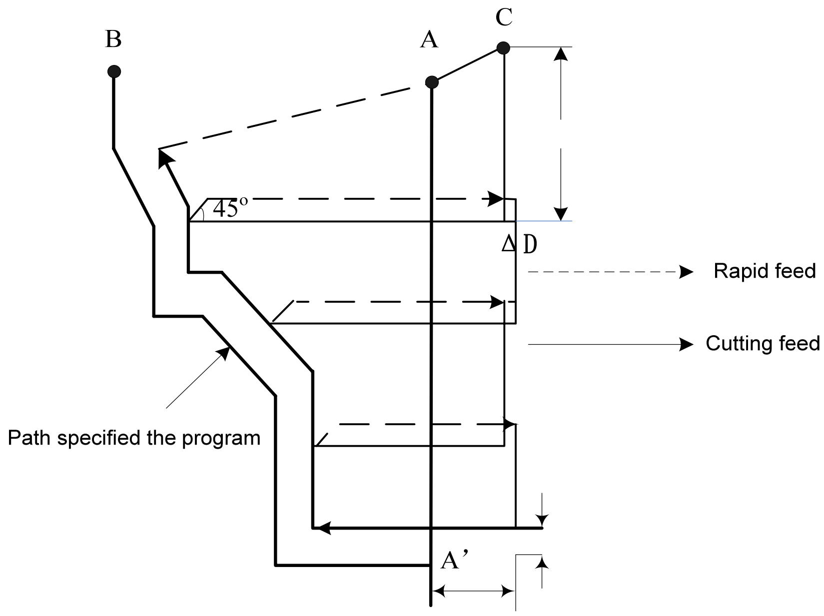


Fig. **4-5-1-1** G71 code path

1. △d,△u are specified by the same U and different with or without being specified P, Q codes.
2. A cycle movement is performed by G71 code specified by P and Q.
3. In G71 cycle, F, S and T functions are invalid among the sequence number NS~~NF blocks, which can be ignored. However, F, S and T which are commanded before or in

G71 block are valid. F, S and T are only valid for G70 code cycle among sequence number NS~~NF blocks.

4. With constant surface speed control selection function, G96 or G97 is invalid among sequence number NS~~NF blocks, the codes before or in G71 are valid.
5. According to the different cutting direction, the G71 code path has the following four situations (Fig. 4**-5-1-2**). Anyway, the tool cuts parallel with Z axis, signs of Δu and Δw are shown below:

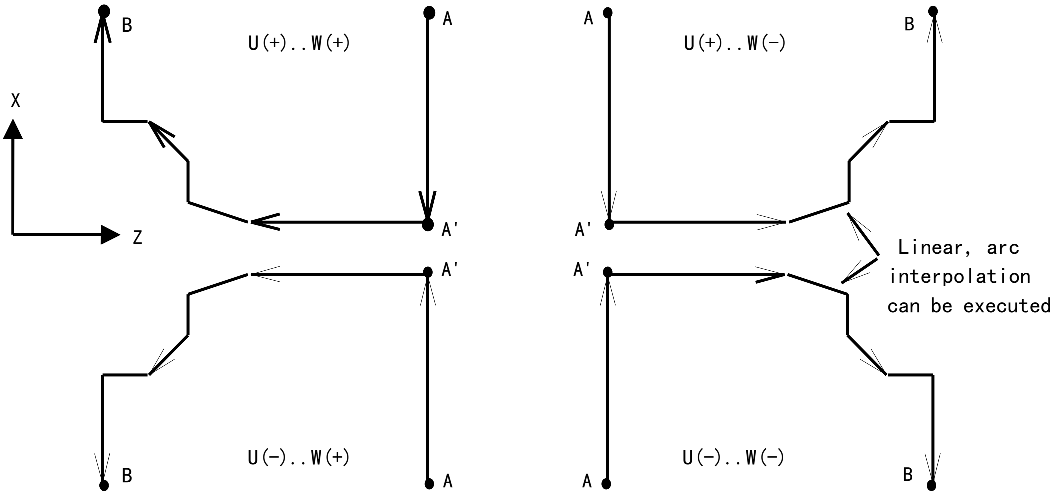


Fig. 4-5-1-2

6. The blocks of NS sequence number between A and A' can include G00 or G01 code, but Z coordinates of A and A' must be consistent.
7. X and Z axis must be monotone increasing or decreasing between A' and B
8. In the blocks of sequence number from NS to NF, the subprograms can not be called.
9. Up to 100 blocks between NS and NF can be compiled. ERR137 alarm occurs when the block quantity exceeds 100.

Example: Use the compound fixed cycle G71 to edit the part program shown in Fig.4-5-1-3.

Ⅰ Programming

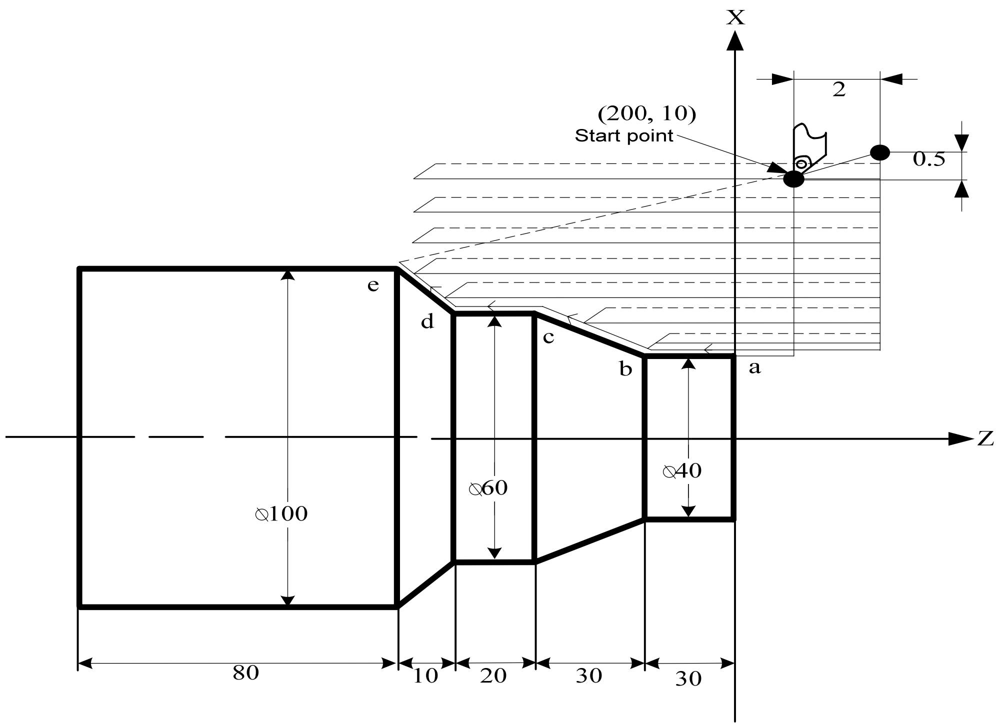


Fig. 4-5-1-3

Program：

O0001；

N010 G0 X220.0 Z50； （Position to a safety position）

N020 M3 S300； （Spindle rotation CCW，speed: 300r/min）

N030 M8； （Cooling ON）

N040 T0101； （Import a roughing tool）

N050 G00 X200.0 Z10.0； （Rapid position, approach the workpiece）

N060 G71 U0.5 R0.5； （Cut depth 1 mm [diameter]; tool retraction[diameter] each time）

N070 G71 P080 Q120 U1 W2.0 F100 S200； （Roughing a—d，X allowance 1mm，Z 2mm）

N080 G00 X40.0； （Position to X40）

N090 G01 Z-30.0 F100 S200；（a→b）

N100 X60.0 W-30.0；（b→c） Finishing path block a→b→c→d→e

N110 W-20.0；（c→d）

N120 X100.0 W-10.0；（d→e）

N130 G00 X220.0 Z50.0； (Rapid retract to the safety position）

N140 T0202； (Change No. 2 finishing tool，and execute its tool offset）

N150 G00 X200.0 Z10.0； （Position to the cycle start point commanded by G70）

N160 G70 P80 Q120； (Finishing a--- e）

N170 M05 S0； (Stop the spindle, speed）

N180 M09； （Stop cooling）

N190 G00 X220.0 Z50.0 T0100； (Rapid retract to the safety position, retrieve the reference tool, clear tool offset）

N200 M30； （End of program）

4.5.2 Grooving Cycle Machining（G71 type Ⅱ）

G71 type II：the type Ⅱ is different from type 1, the outline contour along X axis does not need monontonely increase or decrease. The contour can be machined, up to 20 groovings when its shape along Z monontonely changes.

The first tool needs to be vertical, and Z axis can perform machining when Z direction is a monotonous change's shape.

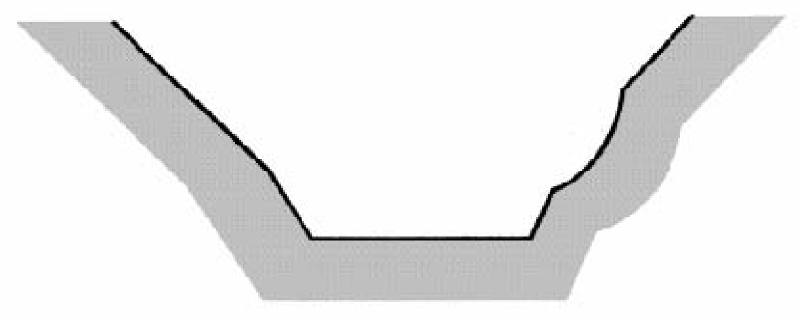


Fig. 4-5-2-1

Because Z is the non-monotonously change, an alarm occurs, which is shown in Fig. **4-5-2-2**.

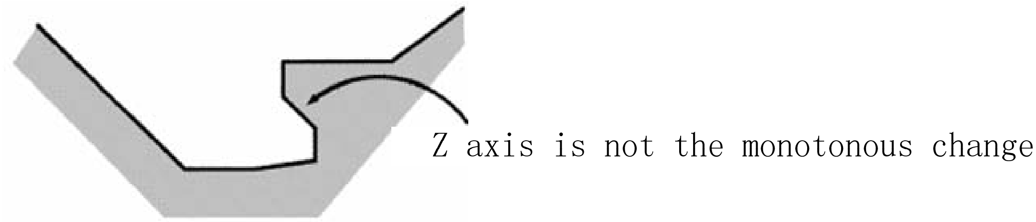


Fig. 4-5-2-2

Command format: G71 U（Δd）R（e）;

G71 P（NS） Q（NF） U（Δu） W（Δw）F S T ;

N（NS） G0/G1 X（U） Z（W）. ;
. . . . ;
. . . . F;
. . . . S;　　blocks for finishing path
. . . . T;
.
N（NF）. . . . . ;

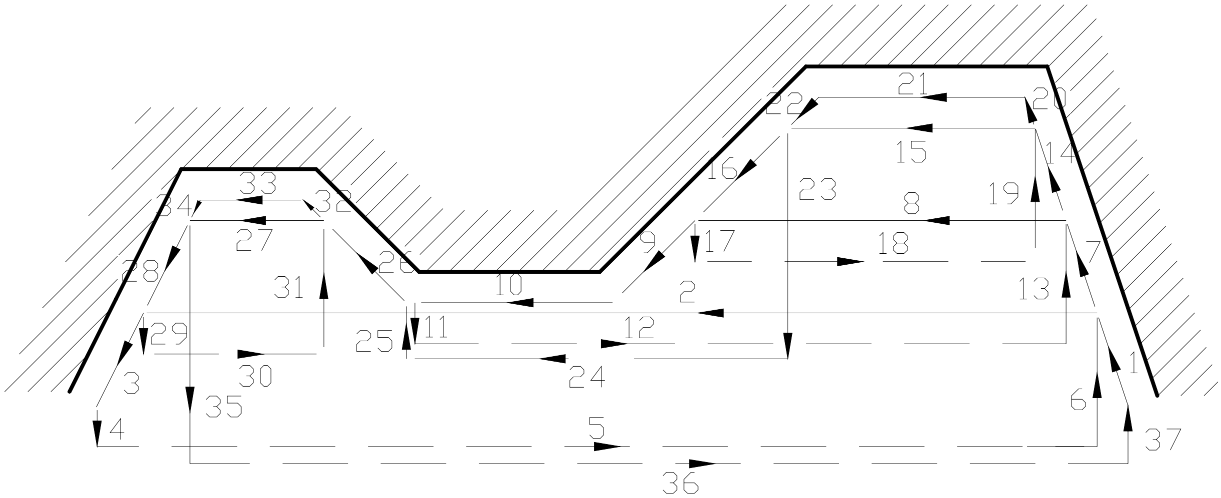


Fig. 4-5-2-3（G71 type Ⅱ machining path）

After the turning, the system should execute the tool retraction, the retraction amount is specified by R（e）or No. 464 as Fig. **4-5-2-4.**

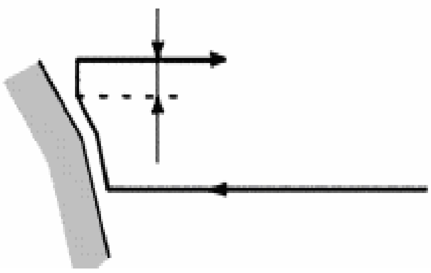


Fig. 4-5-2-4

Note：

1. ns block is only G00, G01. When the workpiece is type II, the system must specify the two axes X(U) and Z(W), and W0 must be specified when Z does not move.
2. The tool retraction point should be high or low as possible to avoid crashing the workpiece.
3. For type II, only X finishing allowance can be specified; when Z finishing allowance is specified, the whole machining path offsets, and it can be specified to 0.
4. Other notes are consistent with those of G71 type I.
5. P0461 compound turning cycle G71，G72 are non-monotone allowable value (the plane 1st axis), roughing direction's axis of type I, type II is non-monotone change, an alarm occurs. Sometime, a tiny non-monotone change's shape is formed at case of automatically creating programs, the parameter without a negative sign is set as a allowable value. So, even if a non-monotone change's shape is contained, G71, G72 can be executed.
6. P0462 compound turning cycle G71，G72 are non-monotone allowable value (the plane 2nd axis), roughing direction's axis of type I, type II is non-monotone change, an alarm occurs. Sometime, a tiny non-monotone change's shape is formed at case of automatically creating programs, the parameter without a negative sign is set as a allowable value. So, even if a non-monotone change's shape is contained, G71, G72 can be executed.
7. P0477 compound turning fixed cycle G71, G72 start position's idle stroke amount of cutting feed, cutting feed distance of rapidly traverse to tool infeed points.

4.5.3 Radial Roughing Cycle（G72 type I）

Command format：G72 W（Δd）R （e）F_ S_ T_；

G72 P（NS） Q（NF） U（Δu） W（Δw） ；

N（NS）G0/G1 Z(W)．．；
．．．．．．．；
．．．． F；
．．．． S；　　blocks for finishing path
．．．． T；
N（NF）．．．．．；

Function: According to the finishing path, the finishing allowance, the path of tool infeed and retract tool, the system automatically counts the path of roughing, the tool cuts the workpiece in paralleling with X. The code is applied to the formed roughing of non-formed rod.

Explanation:

Δd：cutting depth every time, no sign. The cutting direction is depended on AB, the range is 0.001 mm～99999.999mm. It is a modal code, and it will be valid until it is commanded next time. Moreover, according to program codes, P463 can also specify it and the parameter value can also be rewritten according to the program code.

e：tool retraction amount, its unit: mm，its range: 0mm～99999.999mm. It is a modal code, and it will be valid until it is commanded next time. Moreover, according to program codes, P464 can also specify it and the parameter value can also be rewritten according to the program code.

NS：the sequence number of the first block among the block group of finishing path.

NF：the sequence number of the last block among the block group of finishing path.

Δu：X finishing allowance distance and direction, and its range is -999999.999mm～999999.999mm.

Δw: Z finishing allowance distance and direction, and its range is -999999.999mm～999999.999mm。

F：cutting feedrate, its range is feed per minute 1mm/min～6000mm/min，feed per rev 0.001mm/r～500mm/r.

S：spindle speed.

T：tool, tool offset number.

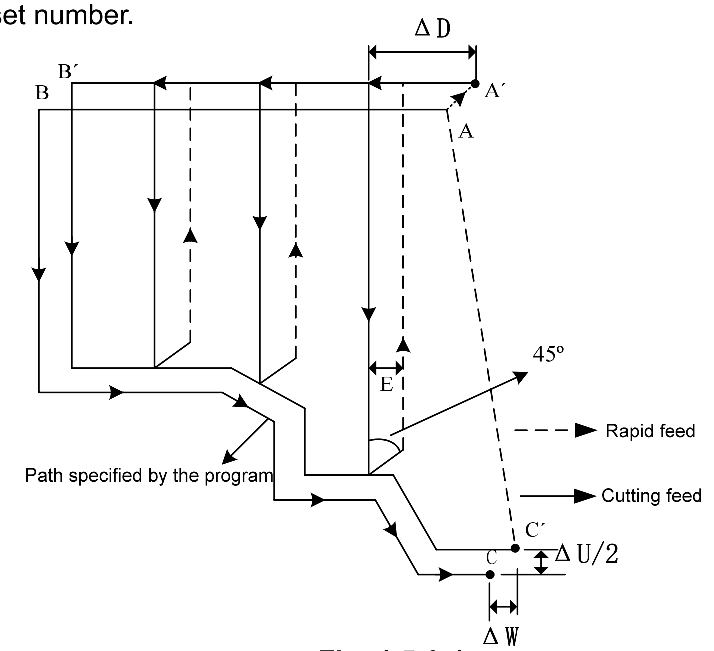


Fig. 4-5-3-1

Explanation:

1. △d, △u are specified by the same W and different with or without being specified P, Q codes.
2. A cycle movement is performed by G72 code specified by P and Q.
3. In G72 cycle, F, S and T functions are invalid among the sequence number NS~~NF blocks, which can be ignored. However, F, S and T which are commanded before or in G71 block are valid. F, S and T are only valid for G70 code cycle among sequence number NS~~NF blocks.
4. With constant surface speed control selection function, G96 or G97 is invalid among sequence number NS~~NF blocks, the codes before or in G72 are valid.
5. According to the different cutting direction, the G72 code path has the following four situations (Fig. 4-5-3-2). Anyway, the tool cuts parallel with X axis, signs of Δu and Δw are shown below:
6. The blocks of NS sequence number between A and A' can include G00 or G01 code, but X coordinates of A and B must be consistent.
7. X and Z axis must be monotone increasing or decreasing between B and C
8. In the blocks of sequence number from NS to NF, the subprograms can not be called.
9. Up to 128 blocks between NS and NF can be compiled. ERR137 alarm occurs when the block quantity exceeds 128.

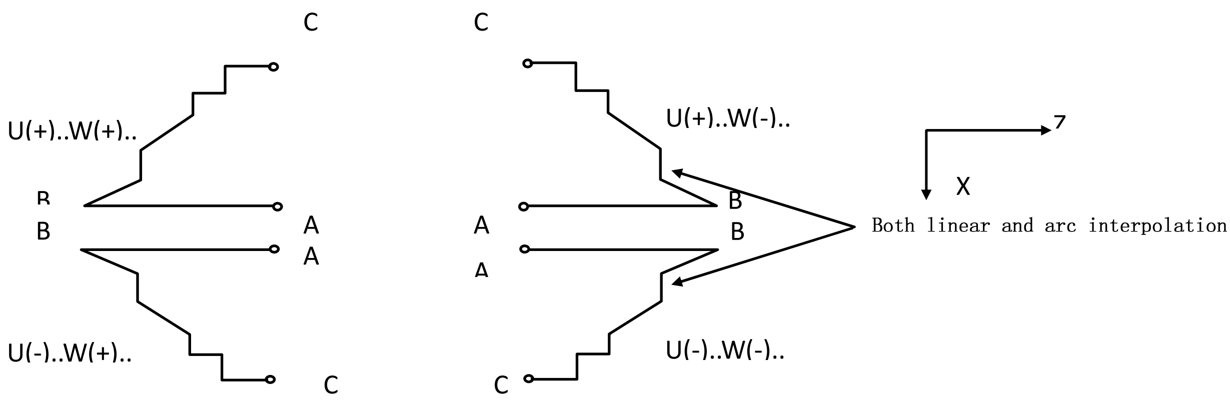


Fig. 4-5-3-2 Four shapes of G72 code path

Example: Use the compound fixed cycle G72 to edit the part program shown in Fig. **4-5-3-3**

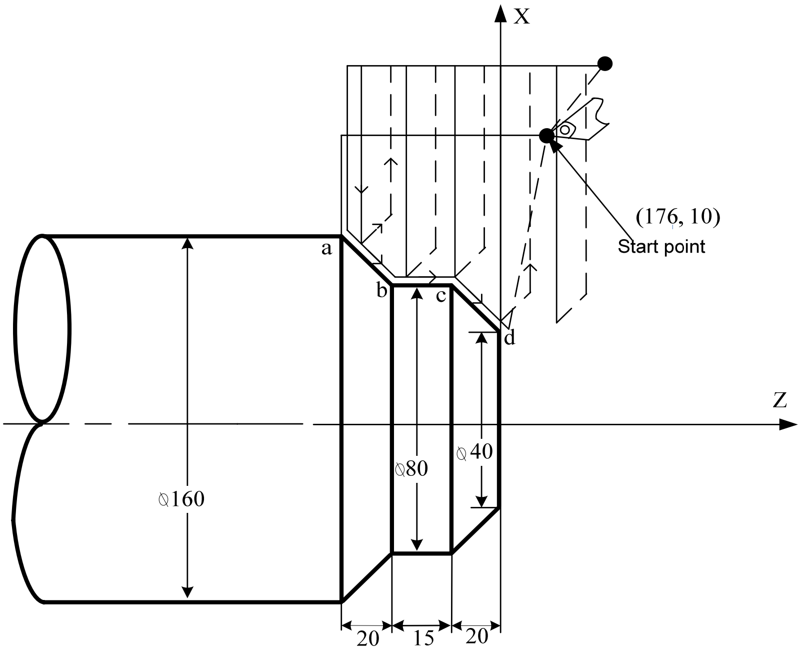


Fig. 4-5-3-3

Program:

PO0002；
N010 G0 X220.0 Z50.0； （Position to a safety position）
N015 T0202； （Change No. 2 tool and execute its tool offset）
N017 M03 S200； （Spindle rotation CCW，speed: 200）
N020 G00 X176.0 Z10.0；（Rapid position, approach the workpiece）
N030 G72 W2.0 R1.0； （Tool infeed 2mm，tool retraction 1mm）
N040 G72 P050 Q090 U1.0 W1.0 F100 S200； （Roughing a—d，X allowance 1mm，Z 2mm）
N050 G00 Z-55.0 S200 ；（Rapid position）
N060 G01 X160.0 F120； （Tool infeed to a）
N070 X80.0 W20.0； （Machining a—b） Blocks for finishing path
N080 W15.0； （Machining b—c）
N090 X40.0 W20.0 ； （Machining c—d）
N100 G0 X220.0 Z50.0； （Rapid retract to the safety position）
N105 T0303； （Change No. 3 g tool，and execute its tool offset）
N108 G00 X176.0 Z10.0； （Rapid return to G70 position）
N110 G70 P050 Q090； （Finishing a—d）
N115 G0 X220.0 Z50.0； (Move to the safety position to execute tool change)
N120 M5 S0 T0200； （Stop the spindle, change No. 2 tool, cancel the tool offset）
N130 G0 X220.0 Z50.0； （Rapid return to the start point）
N140 M30； （Program end）

4.5.4 Grooving Cycle Machining（G72 type II）

The type II is different from the type I as follows:

1）Relative definition: more one parameter than the type I .

2）X external contour need not be the monotonous increasing or the monotonous decreasing be up to 10 grooves as follows:

Fig. 4-5-4-1

But, X external contour must be the monotonous increasing or the monotonous decreasing, and the following contour cannot be machined:

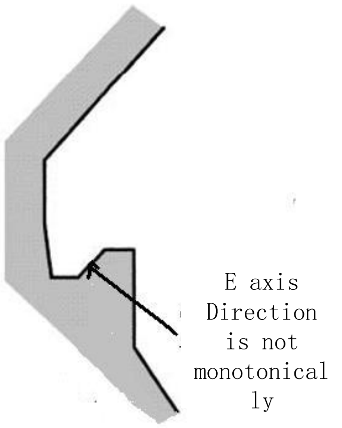


Fig. **4-5-4-2**

1）The first tool cutting need not the vertical: the machining can be executed when Z is the monotonous change shape as follows:

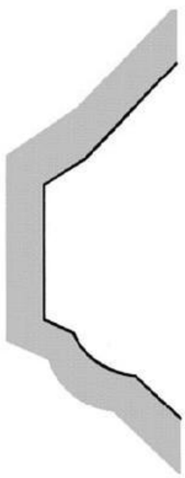


Fig. 4-5-4-3

2) After the turning, the system should execute the tool retraction, the retraction travel is specified by R (e) or No. 464 as follows:

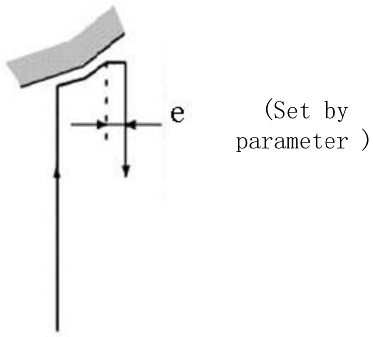


Fig. 4-5-4-4

3) The finishing allowance specifies only Z direction. When X is specified, the whole machining path offsets and it is better to specified it to 0.

Command format: G72 W (Δd) R (e) F_ S_ T_;

G72 P (NS) Q (NF) U (Δu) W (Δw) ;

N (NS) G0/G1 X(U) Z(W) . . ;
. . . . ;
. . . . F;
. . . . S;   Blocks for finishing
. . . . T;
N (NF) . . . . . . ;

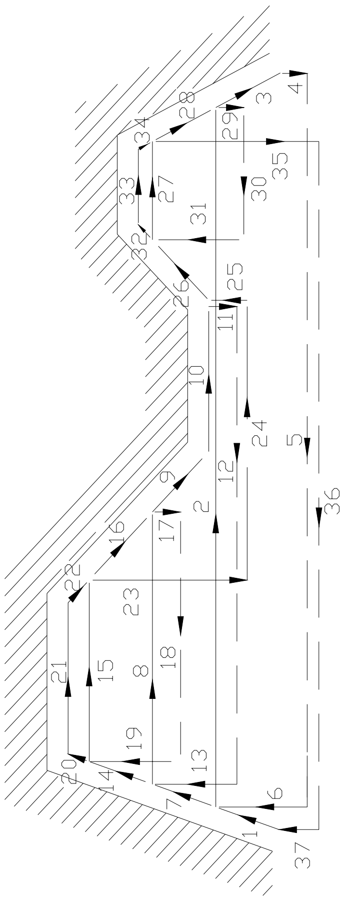


Fig. 4-5-4-5（G72 type machining path）

4.5.5 Closed Cutting Cycle（G73）

Command format：G73u （Δi）W（Δk） R （d）;

G73 P（NS）Q（NF）U（Δu）W（Δw）F S T ;

Function: Use the cycle code, it can cut repeatedly along the path specified by NS~~NF blocks, and the tool moves forward one time after cutting for each time. About the semi-finished products of forging, molding and roughing, it can improve the high-efficiency machining.

Explanation:

Δi：X retraction distance and direction (radius value), unit: mm; modal codes are valid until it is specified next time. Moreover, the parameter P467 can also set it and the parameter value can be rewritten according to the program codes.

Δk：Z retraction distance and direction (radius value), unit:mm; modal codes are valid until it is speicified next time. Moreover, the parameter P466 can also set it and the parameter value can be rewritten according to the program codes.

D：Times of closed cutting, unit: time; mode codes are valid until it is speicified next time. Moreover, the parameter P468 can also set it, and the parameter value can also be rewritten according to the program codes.

NS：The sequence number of the first block which forms the finishing shape in block group;

NF：The sequence number of the last block which forms the finishing shape in block group;

Δu：X finishing allowance, its range: -999999.999mm～999999.999mm ；

Δw：Z finishing allowance, its range:-999999.999mm～999999.999mm；

F：Cutting feedrate, its range: 1 mm/min～6000mm/min；

S：Spindle speed；

T：Tool and tool offset number；

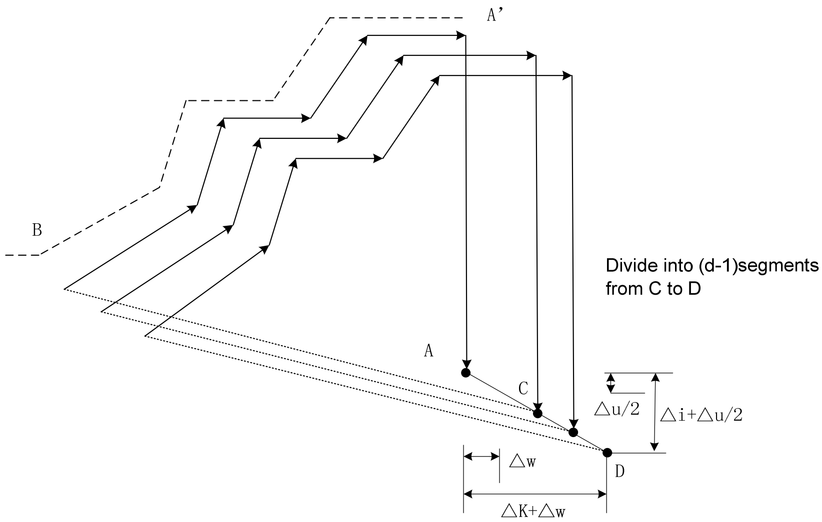


Fig. **4-5-5-1** G73 code running path

1. F, S and T functions in any blocks of NS~NF are invalid, but they are only valid when specified in G73.
2. Δi，Δk，Δu and Δw are specified by address U and W, and the difference is whether it includes specified P and Q.
3. Blocks between NS and NF in G73 can not call subprograms.
4. Use NS~NF blocks to realize cycle machining, please pay attention to codes of Δu, Δw, Δi and Δk during programming. After the cycle completes, the tool returns to point A.
5. Δi orΔk is 0 in program, input U0 or W0; or set parameters P73 and P74 as 0; otherwise, it will get affected by G73 setting value of last time.
6. Up to 100 blocks can compiled into the sequence number NS~NF. An alarm ERR0460 occurs when the block quantity exceeds 100.

Example: Use the closed cutting cycle G73 code to edit the machining program of the part shown in Fig.4-5-5-2.

Ⅰ Programming

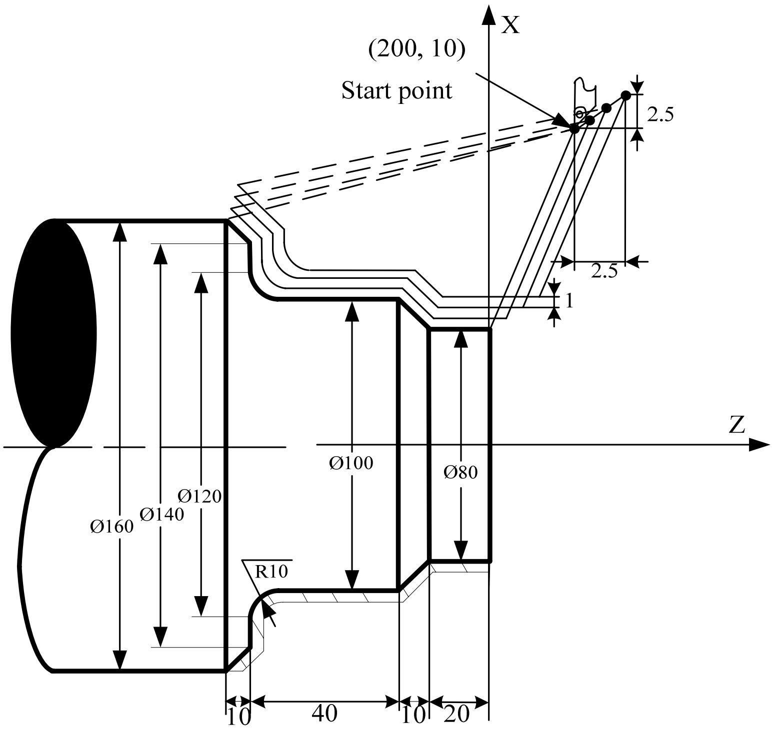


Fig. 4-5-5-2 G73 code example

Programs are as below: (diameter specification, metric input, least blank dimension Ø86）

N008 G0 X260.0 Z50.0 ； （Position to a safety position）
N009 T0101； （Change No.1 tool and executes its tool offset）
N010 G98 M03 S300； （The spindle rotation CW, speed 300）
N011 G00 X200.0 Z10.0； （Rapidly position to the start point）
N012 G73 U2.0 W2.0 R3 ； （X tool retraction 4mm，Z tool retraction 2mm，divided into 3 times to roughing, each tool’s diameter feed 2mm）
N013 G73 P014 Q020 U0.5 W0.5 F100 ；（X leaves 0.5mm，Z leaves 0.5mm finishing allowance）

N014 G00 X80.0 W-10.0 S500 ；
N015 G01 W-20.0 F120 ；
N016 X100.0 W-10.0 ；
N017 W-30.0 ；
N018 G02 X120 W-10.0 R10.0 F100 ；
N019 G01 X140.0 ；
N020 G01 X160.0 W-10.0 ；

(N014–N020: Blocks for finishing shape)

N021 G0 X260.0 Z50.0； (Move a safety position to conveniently execute tool change)
N022 T0303； （Change No. 3 tool，and execute its tool offset）
N023 G00 X200.0 Z10.0； （Rapidly return to G70 positioning）
N024 G70 P014 Q020； （Finishing）
N025 M5 S0 T0200； （Stop the spindle, change No. 2 tool, cancel the tool compensation）
N026 G0 X260.0 Z50.0； （Rapidly return to the start point）

N027 M30； （Program end）

4.5.6 Finishing Cycle（G70）

Command format：G70 P（NS）Q（NF）；

Function: When the code is executed, the tool begins finishing from the start position along with the finishing path of the work piece which is specified by NS~NF blocks. After roughing commanded by G71, G72 and G73, G70 code can execute finishing.

Explanation:

NS: The sequence number of the first block which forms the finishing shape in block group;

NF: The sequence number of the last block which forms the finishing shape in block group;

G70 code path is decided by programming path of NS~~NF blocks. The corresponding position relations of NS and NF in G70~~G73 are as below:

```
........
........
G71/G72/G73 P（NS） Q（NF） U（Δu） W（Δw） F  S  T ；

N（NS） ......
 ........
· F
· S
· T
.
.
.
N（NF）......
        .
G70 P（NS） Q（NF）；
        .
```

1. F, S and T functions are invalid which is specified by “NS” and “NF” in blocks G71, G72 and G73; however, during executing G70, F, S and T are valid, which are specified by the sequence number between “NS” and “NF” blocks.
2. When G70 cycle machining completes, the tool returns to the start point and reads the next block.
3. In G70, the blocks between NS and NF can not call subprogram.

Example: see G71, G72 code examples.

4.5.7 Axial Grooving Cycle（G74）

Command format: G74 R（e）；

G74 X（U）Z（W）P（Δi）Q（Δk）R（Δd）F；

Function: When the code is executed, the system can determine the running path of tool according to the cutting end point set by the block (the point is determined by the coordinate value of X and Z axes in the block), and by the values of e, Δi, Δk and Δd. In the cycle, the chips of outer shape cutting can be processed; moreover, if X (U) and P are omitted, only Z axis movement is deep hole cycle. About the path, it is shown in figure 4-5-7-1.

Explanation:

e: The retraction amount range after cutting Δk along Z direction is 0～9999.999 mm；the mode code is valid until the next specification. Moreover, P469 can also set it, according to the program codes, the parameter value can also be rewritten, the radius code;

X： X absolute coordinate value of the cutting end point B2, unit: mm;

U： X total movement amount from the cutting end point B2 to the start point A, unit: mm;

Z： Z absolute coordinate value of cutting end point B2, unit: mm;

W：Z total movement amount from the cutting end point B2 to the start point A, unit: mm;；

Δi： X cycle movement amount every time (without sign and radius value), unit: mm;

Δk：Z cutting movement amount every time (without sign), unit: mm;

Δd：X tool retraction of cutting to the end point（radius value, unit：mm；

F： Cutting feedrate. Its range: feed per minute 1mm/min～8000mm/min，feed per rev 0.001mm/r～500mm/r".

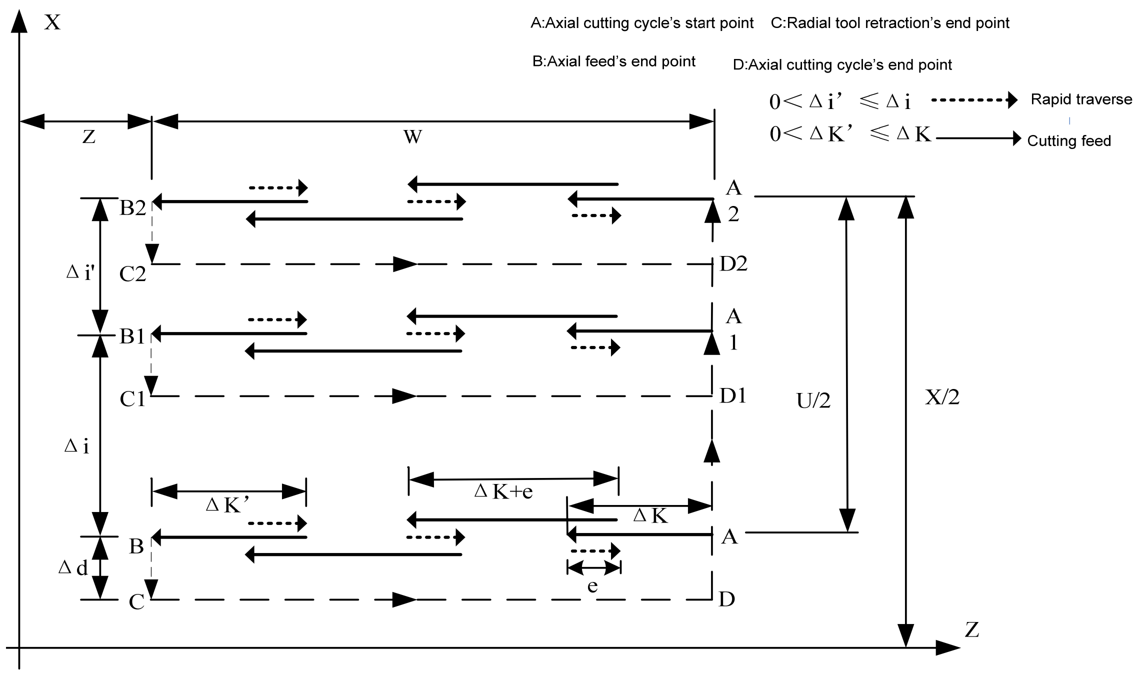


Fig. **4-5-7-1**

1. e and Δd are specified by address R, their difference is whether there is specified X (U), that is to say, if X (U) is commanded, it is Δd, otherwise, it is e;
2. The cycle movement adopts G75 code specified by X (U). When only "G74 R（e）" is executed, the cycle operation is not performed.

Example: Use G74 code to edit the part program shown in Fig.4-5-7-2.

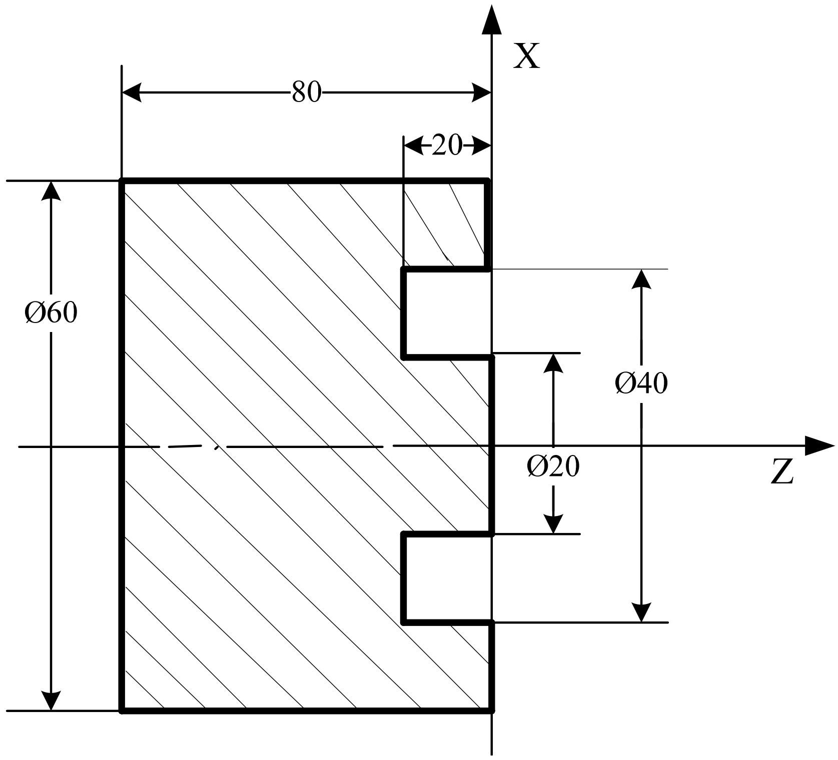


Fig. 4-5-7-2

Program:

O0001；	（Program name）
G0 X100 Z50；	（Rapid position）
T0101；	（tool width 2mm）
M3 S500 G97；	（Start the spindle, set speed to 500）
G0 X36 Z5；	（Position at the start point of machining，X adds the tool width）
G74 R1 ；	（code Z tool retraction amount）
G74 X20 Z-20 P2 Q3.5 F50；	（X cycle movement amount every time 4mm，Z cycle movement amount every time 3.5mm）
G0 Z50；	（Z tool retraction）
X100；	（X tool retraction）
M5 S0；	（stop the spindle）
M30；	（end of program）

4.5.8 Radial Grooving Cycle（G75）

Command format： G75 R（e）；

G75 X（U） Z（W） P（Δi）Q（Δk）R（Δd）F ；

Function: When the code is executed, the system determines the running path of the tool based on the cutting end point set by blocks (the point is determined by X and Z coordinate values) and by the values of e, Δi, Δk and Δd. Equivalent to G74, it interchanges X and Z, and the chips of end face cutting can be performed in the cycle, and the external diameter can be executed channel and cutting machining (Z, W and Q can be omitted). The path is shown in Fig. 4-5-8-1.

Explanation:

e: The retraction amount range after cutting Δi along X direction is 0～9999.999 mm；the mode code is valid until the next specification. Moreover, P76 can also set it, according to the program codes, the parameter value can also be rewritten, the radius code;

X: X absolute coordinate value of the cutting end point B2, unit: mm;

U: X total movement amount from the cutting end point B2 to the start point A, unit: mm

Z: Z absolute coordinate value of the cutting end point B2, unit: mm;

W: Z total movement amount from the cutting end point B2 to the start point A, unit: mm;

Δi：X cycle movement amount every time (without sign and radius value), unit: mm;

Δk：Z cutting movement amount every time (without sign), unit: mm;

Δd：Retraction amount of cutting to the end point, unit: mm;

F: Cutting feedrate.

G74 and G75 all can be used for cutting, grooving or hole machining and the tool can automatically retract.

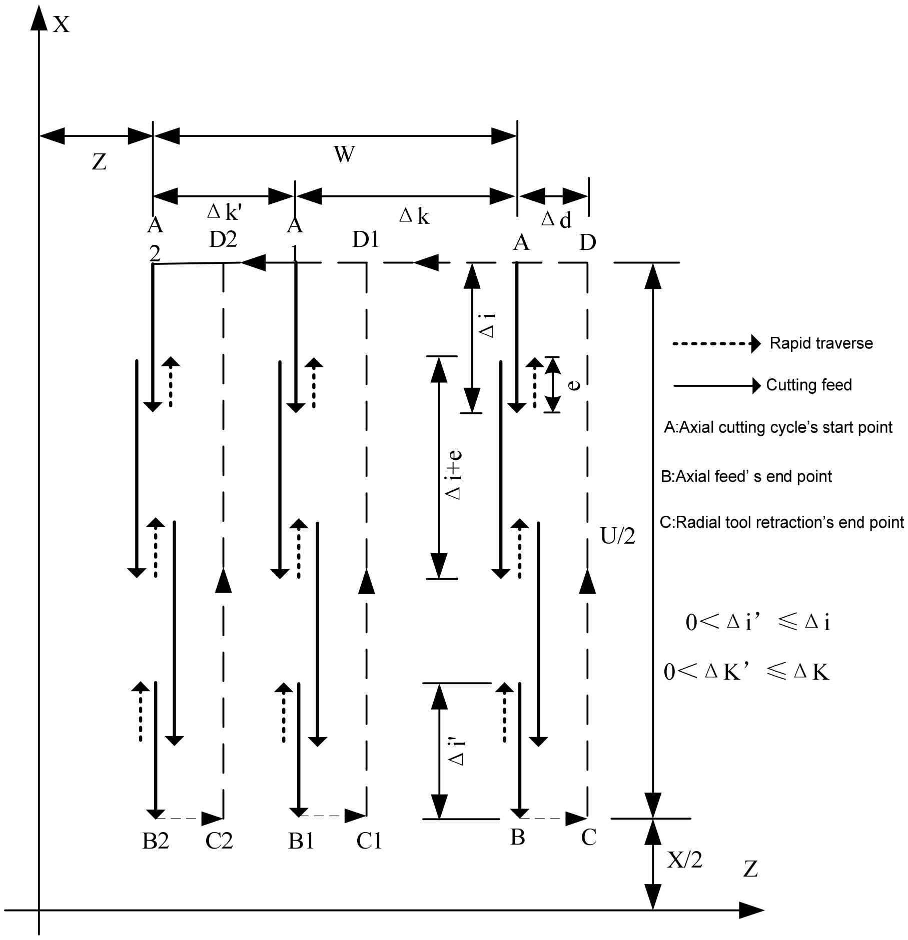


Fig. 4-5-8-1

1. e and Δd are specified by address R, their difference is whether there is specified X (U), that is to say, if X (U) is commanded, it is Δd, otherwise, it is e;
2. The cycle movement adopts G75 code specified by X (U).

Example: Use G75 code to edit the part program shown in Fig.4-5-8-2.

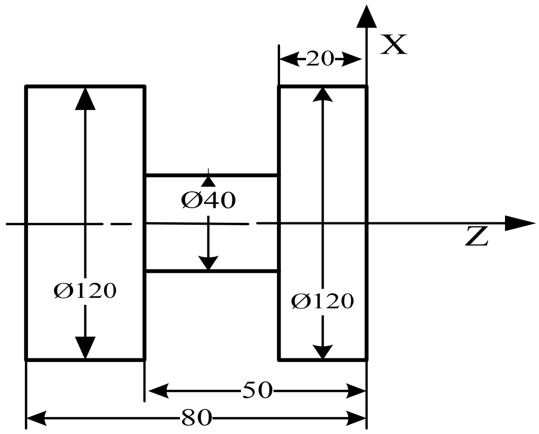


Fig. 4-5-8-2 G75 code cutting example

Program:

O0001； （Program name）
G0 X150 Z50； （Rapid position）
T0101； （tool width 4mm）
M3 S500 G97； （Start the spindle, set speed to 500）
G0 X125 Z-24； （Position at the start point of machining，Z adds the tool width）
G75 R1 ； （code X tool retraction amount）
G75 X40 Z-50 P2 Q3.5 F50； （X cycle movement amount every time 4mm，Z cycle movement amount every time 3.5mm）
G0 X150； （X tool retraction）
Z50； （Z tool retraction）
M5 S0； （stop the spindle）
M30； （end of program）

4.5.9 Multiple Thread Cutting Cycle（G76）

Command format: G76 P（m）（r）（a）Q（△dmin） R（d）；
G76 X（U） Z（W） R（i）P（k）Q（△d）F（I） ；

Function: The system can automatically calculate and perform the thread cutting cycle for many times based on the data which are specified by the code address, and the code path is shown as Fig. **4-5-9-1**.

Explanation:

X, Z: The coordinate value of thread end point (thread bottom), unit: mm;

U, W: The coordinate value which the thread end point corresponding to the start point of machining, unit: mm;

m: The repeated times 1~~99 for final finishing, the code value is mode. It is valid before the next specification. Moreover, **P472** can also set it, and the parameter value can also be rewritten based on the program code. Finally, the finishing repeated times can be 1~99;

r: Thread chamfering value. If take L as lead among the range of 0.1L~~9.9L, and 0.1L is taken as one level, then the two digits can be specified by 00~~99. If the code is mode, it will be valid till the next specification. Moreover, **P474** can also set it, and the parameter value can be rewritten based on the program code. In G76 program, the thread chamfering value can be set, then the cycle of thread cutting in G92 also function.

a：The angle of tool nose, also the screw thread, can be selected from the 6 angles of 80°,60°,55°，30°, 29°and 0°.The original angel value is specified by two digits. The code is mode, and it will be valid until the next specification. Moreover, parameter P78 can also be set, and the parameter value can also be rewritten according to the program codes. The angels of tool nose can be selected from the 6 angles: 80°, 60°, 55°, 30°, 29° and 0°;

△ dmin：Minimum cutting amount, unit: mm, When one time cutting amount (△D×$\sqrt{N}$-△D×$\sqrt{N-1}$) is less than △ dmin, △ dmin is taken as the cutting amount of one time. The code is mode, and it will be valid until the next specification. Moreover, it can also be set by P470, and the parameter value can be rewritten by the program code. The setting range of the minimum cutting is 0～9999999, unit: 0.001mm;

d：Finishing allowance, unit: mm. The code is mode, and it is valid until the next specification. It can also be set by P471, and the parameter value can be rewritten by the program code. The setting range of finishing allowance is 0～～9999.999, unit: 0.001mm;

i: Radius difference of threading parts, unit: mm, i=0 is the straight thread of cutting;

k: Height of screw thread (distance in X direction is commanded by the radius value), unit: mm;

△d：Cutting depth at the first time, radius value, unit: mm.

F: Thread lead, unit: mm;

I: Tooth quantity each inch.

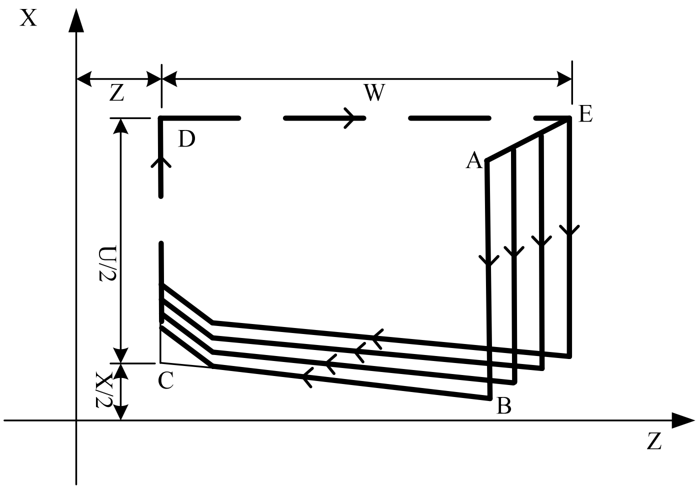


Fig. 4-5-9-1

The details of cutting method is referred to Fig. **4-5-9-2**：

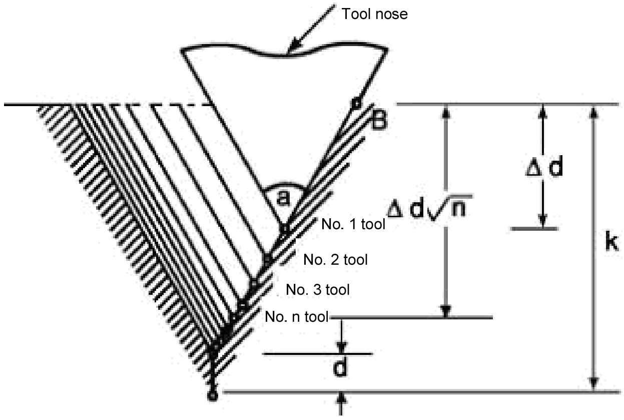


Fig. **4-5-9-2**

1. Use the data specified by P, Q and R, it can differ whether there is address X (U) and Z (W) or not.
2. The cycle movement is commanded by G76, which is specified by address X (U) and Z (W).
3. In cycle machining, the tool is one-sided tool edge for machining, the load of tool nose can be reduced.
4. The cutting amount at the first time is △d, and the N times is △D× $\sqrt{N}$, and the cutting amount is same in each time
5. Considering the code of each address, there are four figures of machining, which can also process the internal thread. In the thread cutting shown in figure **4-5-8-1**, only the feedrate is specified by F code between B and C, the others are rapid feedrate.

   In the cycle, the code of the increment is determined by the following methods:

   U: It is determined by the path from direction A to C;

   W: It is determined by the path from direction C to D;

   R (I): It is determined by the path from direction A to C;

   P (K): Positive;

   Q（△D）: Positive.
6. About the precautions of cutting thread, they are same as those of G32 cutting thread.
7. The specified thread chamfering value is also valid for G92 thread cutting cycle.
8. m, r and a are specified by address P in one time.

Example: the thread cutting compound cycle G76 compiles a program shown in Fig. 4-5-9-3, and the machining thread M68×6.

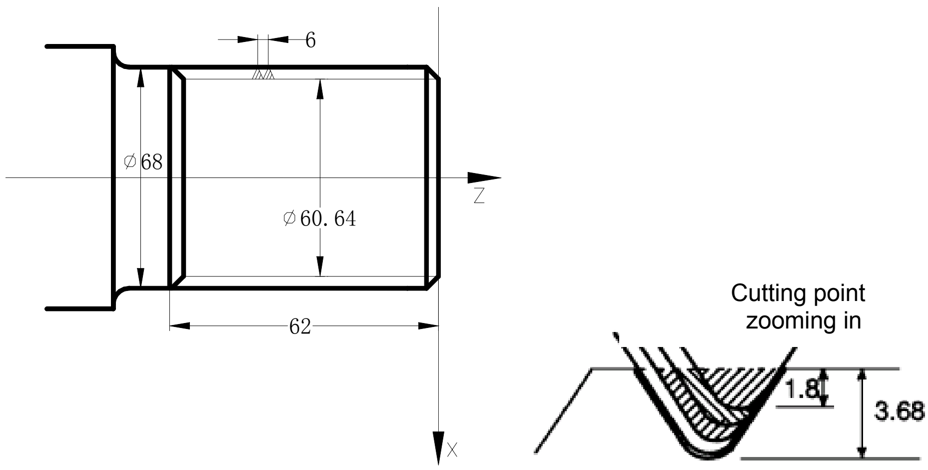


Fig. 4-5-9-3

Program:

G00 X100 Z50；	（position to the safety position）
M03 S300；	（Start the spindle, specify the speed）
G00 X80 Z10；	（Rapid position to the start point of machining）
G76 P011060 Q0.1 R0.2；	（execute the thread cutting）
G76 X60.64 Z-62 P3.68 Q1.8 F6.0；	
G00 X100 Z50；	（return to the program's start point）
M5 S0；	（the spindle stop）
M30；	（Program end）

4.5.10 Notes for Compound Fixed Cycle Code

1. The necessary parameters P, Q, X, Z, U, W and R must be commanded correctly in the specified compound fixed cycle blocks
2. In the blocks of G71, G72 and G73, when P codes the sequence number, the sequence number corresponding to the block must code G00 or G01 of G codes in group 01, otherwise, P/S alarm occurs.
3. In MDI mode, G70，G71，G72，G73，G74，G75 and G76 codes can not be executed; even if it is commanded, it can not be executed.
4. In blocks of G70, G71, G72 and G73, the range of the sequence number specified by P and Q codes, the following codes can not be included.
   ★Group 01 codes except for G00，G01，G02 and G03;
   ★ M98/M99；
   ★G04 is valid in the finishing and the last tool of roughing.
5. During executing the compound fixed cycle (G70~~G76), the movement can be stopped and the manual operation can be performed. However, when the fixed cycle in compound type should be performed again, it must return to the position before the manual operation. If it restarts without return, the manual movement amount can not be added in the absolute value, the following movements will misplace, and the absolute value is same as the manual movement value.
6. During executing G70，G71，G72 and G73, the sequence number specified by P and Q can not coincide in the program.
7. About the precautions of thread cutting specified by G76, the G32 thread cutting is same as G92 thread cutting cycle, and the specified chamfering value is also valid for G92 thread cutting cycle.

4.6 Tool Compensation Function

The actual tool nose is not one point, but a section of circular. Because the circular of tool nose, there exist the errors between the actual machining result and the work piece program; the tool compensation function C can compensate the tool radius to eliminate the above errors.

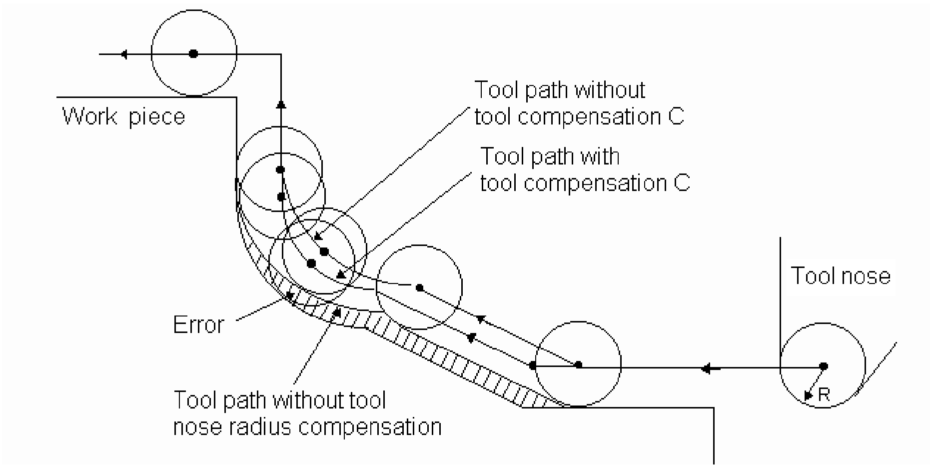


Fig. 4-6-1

4.6.1 Basic Concept of Tool Compensation Function C

4.6.1.1 Imaginary Tool Nose Concept

In the following Fig. **4-6-1-1-1,** the tool nose A is a imaginary point, so it does not exist actually. Therefore, it is called the imaginary tool nose (or the ideal tool nose). The actual tool nose radius center is very difficult to set at the start position while it is very easy for the imaginary one, which is shown in the following figure. It is same as the tool nose center; it does not need to consider the tool nose radius when using the imaginary tool nose programming.

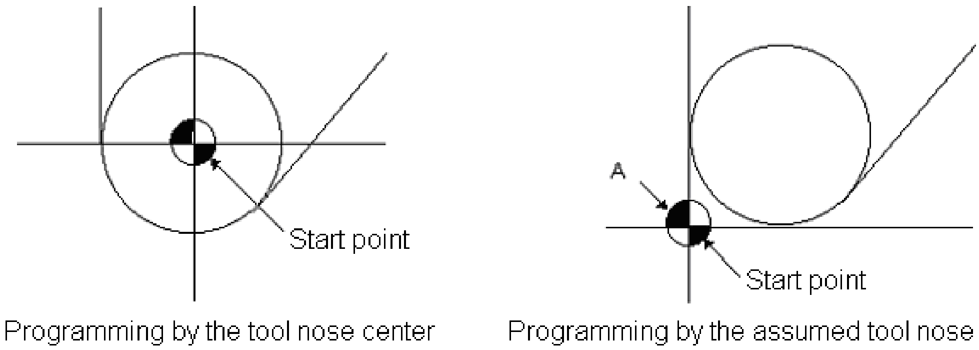


Fig. 4-6-1-1-1 Tool nose radius center and imaginary tool nose

Note: About the machine with zero point, a standard point, such as the tool post center, can be taken as the start point. The distance from the standard point to the center of tool nose radius or the distance of imaginary tool nose is set to the tool offset value.

Set the distance from the standard point to the center of tool nose radius as the offset value, like setting the radius center of tool nose as the start point; set the distance from the standard point to the imaginary tool nose as the offset value, like setting the imaginary tool nose as the start point. To set the offset value of tool, normally, measuring the distance from the standard point to the imaginary tool nose is easier than measuring the distance from the standard point to the tool nose radius center. Therefore, generally, the distance from the standard point to the imaginary tool nose is to set the tool offset value.

When the center of tool post is taken as the start point, the tool offset value is shown in Fig. 4-6-1-1-2：

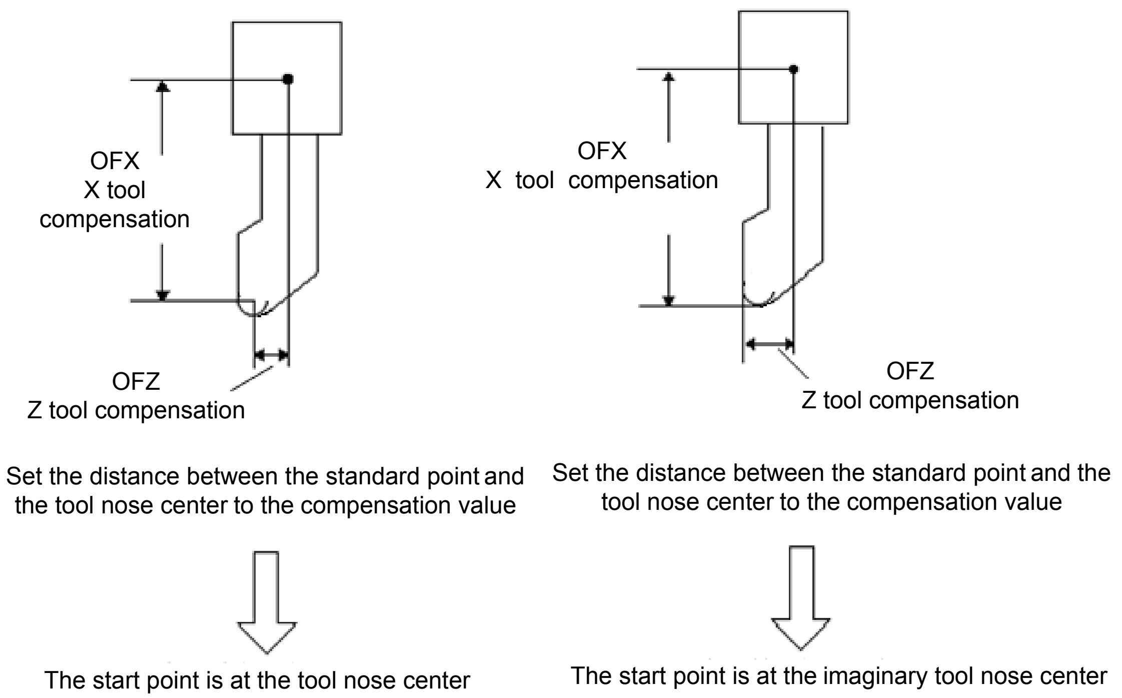


Fig. 4-6-1-1-2 Setting the tool offset value under condition of the center of tool post as the standard point

Fig. 4-6-1-1-3 and Fig. 4-6-1-1- 4 respectively take the tool paths programmed by the tool nose center and the imaginary tool nose. The left figure is without the tool nose radius compensation, and the right figure is with the tool nose radius compensation.

If the tool nose radius compensation is not used, the tool nose center path is same as the programmed path.

If the tool nose radius compensation is used, it achieves the cutting precisely.

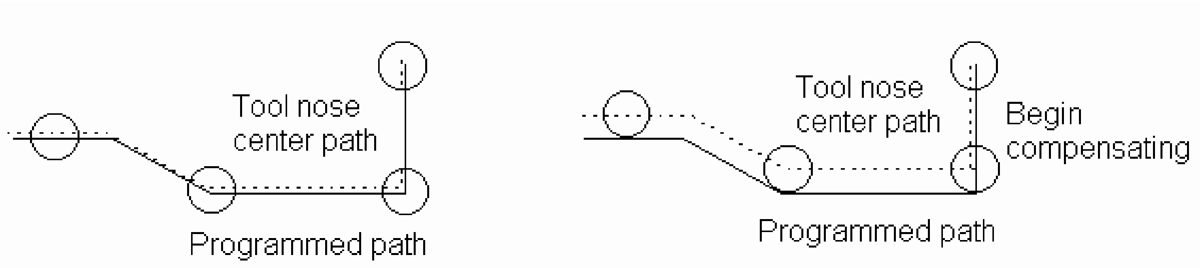


Fig. 4-6-1-1-3 Tool path during tool nose programming

Without the tool nose radius compensation, the imaginary tool nose path is same as the programmed path.

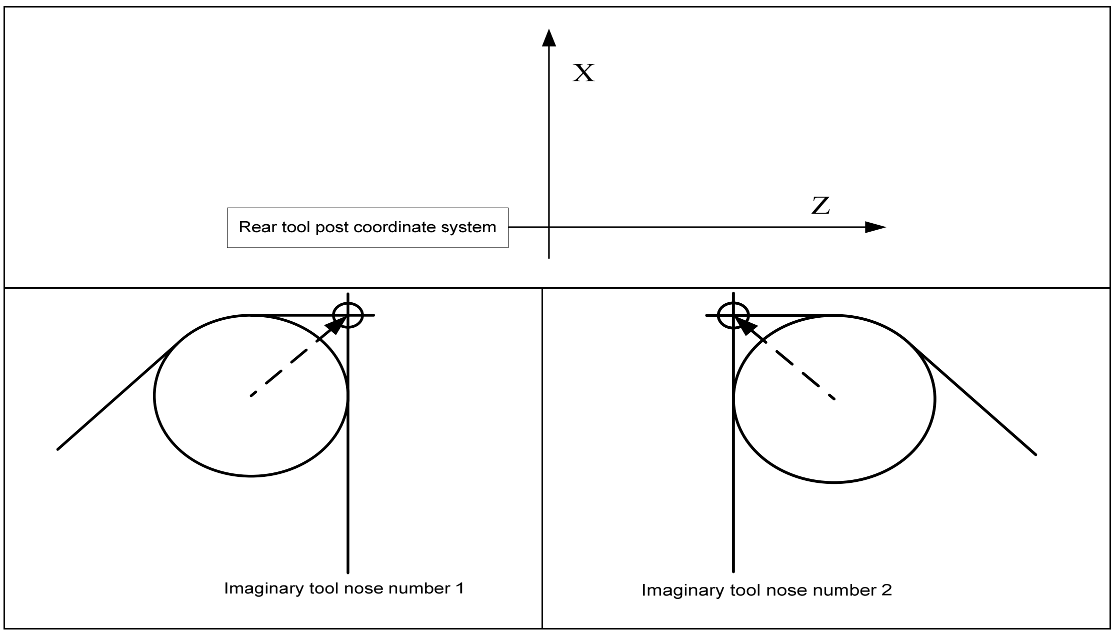
Use the tool nose radius compensation, and achieve the cutting precisely.

Fig. 4-6-1-1-4 Tool path during the imaginary tool nose programming

4.6.1.2 Imaginary Tool Nose's Direction

In the actual machining, due to the machining requirement of the work piece, the different positional relations exist between the tool and the work piece. From the center of the tool nose, the direction of the assumed tool nose is determined by that of the tool during cutting.

The assumed tool nose number defines the positional relation between the assumed tool nose point and the tool nose circular center. Number of assumed tool nose totally has 10 (0~~9), and represents 9 directions of the positional relations. Before the tool nose radius compensation, the number of assumed tool nose must set with the compensation amount in the tool nose radius compensation memorizer. The direction of assumed tool nose can be selected from the eight specifications in the following figures. These figures illustrate the relation between the tool and the start point, and the end point of arrow is the assumed tool nose.



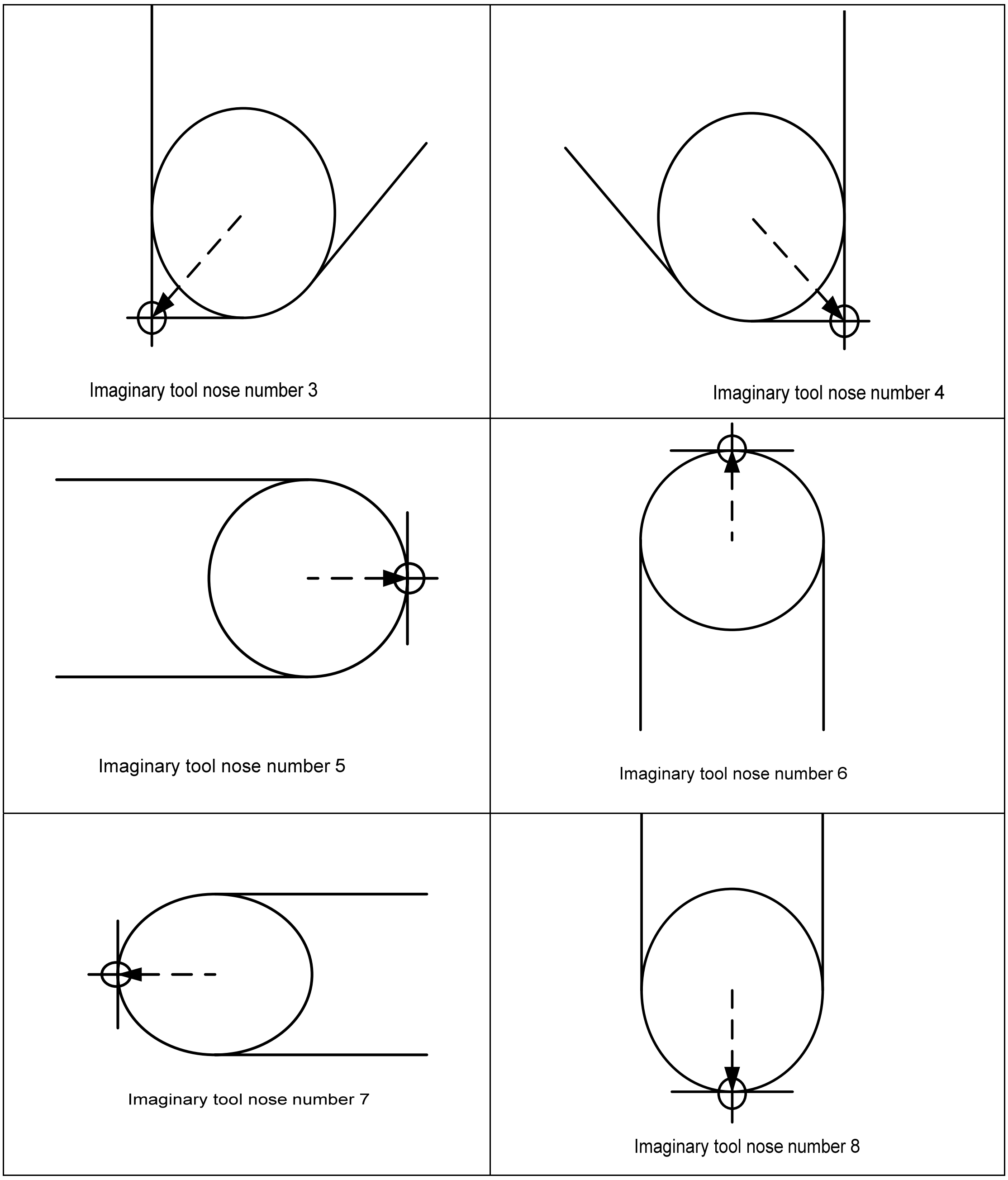
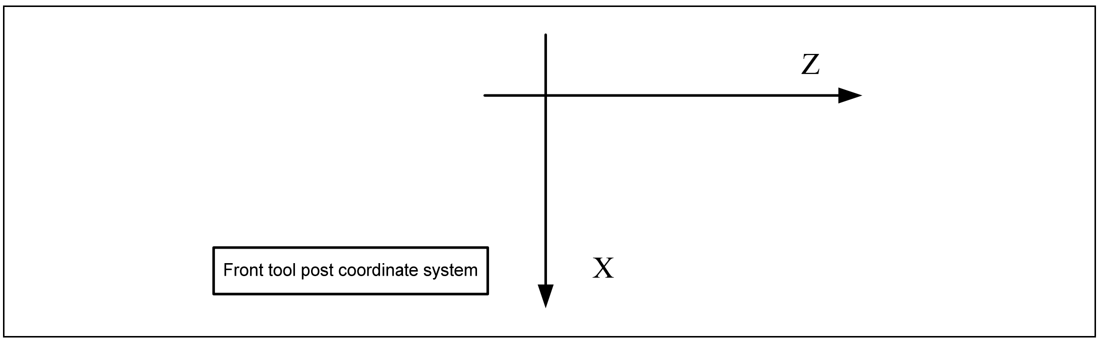


Fig. 4-6-1-2-1 imaginary tool nose number of rear tool post coordinate system



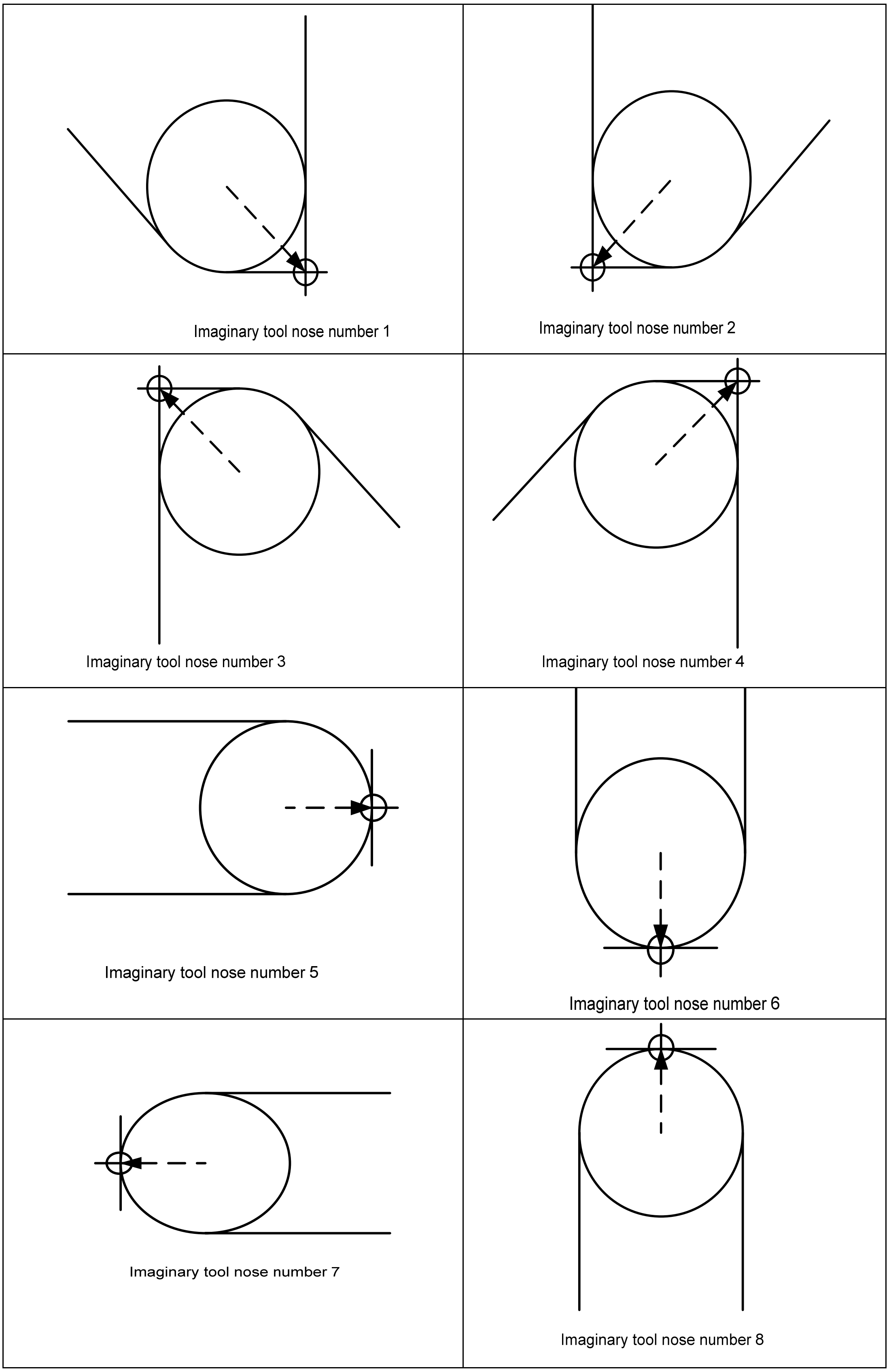


Fig. 4-6-1-2-2 imaginary tool nose number of front tool post coordinate system

When the tool nose center complies with the start point, the tool nose number to 0 or 9 is set. Correspond to the compensation number of each tool and T sets the assumed tool nose number of each tool.

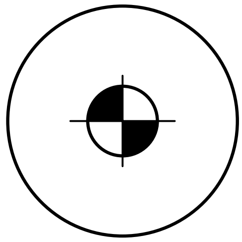


Fig. 4-6-1-2-3 Tool nose center consistent with the start point

4.6.1.3 Setting Compensation Values

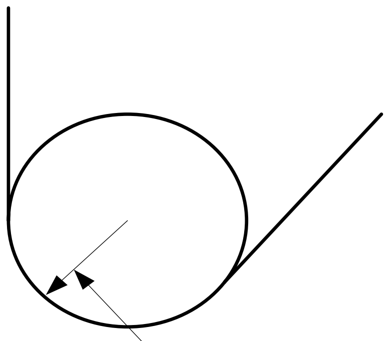


Fig. 4-6-1-3-1 Compensation value of tool nose radius

Before compensating the tool nose radius, the following compensation values should be set: X, Z, R and T. Among them, X and Z respectively is the tool offset value from the tool post center to the tool nose in X and Z axis; R is the radius compensation value of assumed tool nose; T is the assumed tool nose number. Value of each group corresponds to one tool compensation number, and set in the interface of tool compensation. About the details, refer to Section *Operation*, *Rewriting and Setting the Tool Compensation Values*.

The details are shown in the following table **4-6-1-3-1**：

Table 4-6-1-3-1 Display page of the system tool nose radius compensation values

No.	X	Z	R	T
001	0.020	0.030	0.020	2
002	0.060	0.060	0.016	3
..	..	..	..	..
..	..	..	..	..
..	..	..	..	..
015	0.030	0.026	0.18	9
064	0.050	0.038	0.20	1

4.6.1.4 Relative Position between the Tool and the Workpiece

When the tool nose radius compensates, the relative position of tool and work piece should be specified. In the coordinate system of rear tool post, when the tool center path is at the right side of the programmed path (part path) forward direction, which is called as the right tool compensation, which can be realized by G42 code; when the tool center path at the left side of the programmed path (part path) forward direction, it is called as the left tool compensation, which can be realized by G41 code; and the front tool post is opposite. About the relative position of the tool and the work piece in the codes G40, G41 and G42, refer to the table **4-6-1-4-1**:

Table 4-6-1-4-1

Code	Function	Note
G40	Cancel the tool nose radius compensation	About the details, refer to Fig. 4-6-1-4-1 and Fig. 4-6-1-4-2
G41	The tool nose radius compensation (left) in the rear tool post coordinate system, the tool nose radius compensation (right) in the front tool post coordinate system.	
G42	The tool nose radius compensation (right) in the rear tool post coordinate system, the tool nose radius compensation (left) in the front tool post coordinate system	

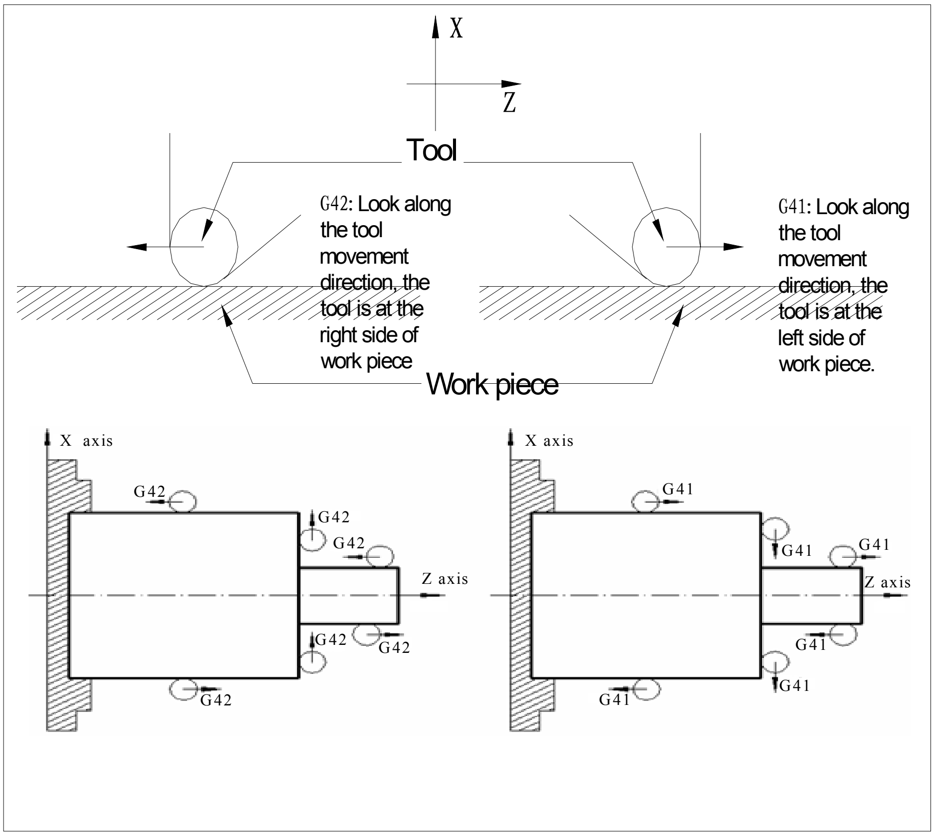


Fig. 4-6-1-4-1 Tool nose radius compensation in rear tool post coordinate system

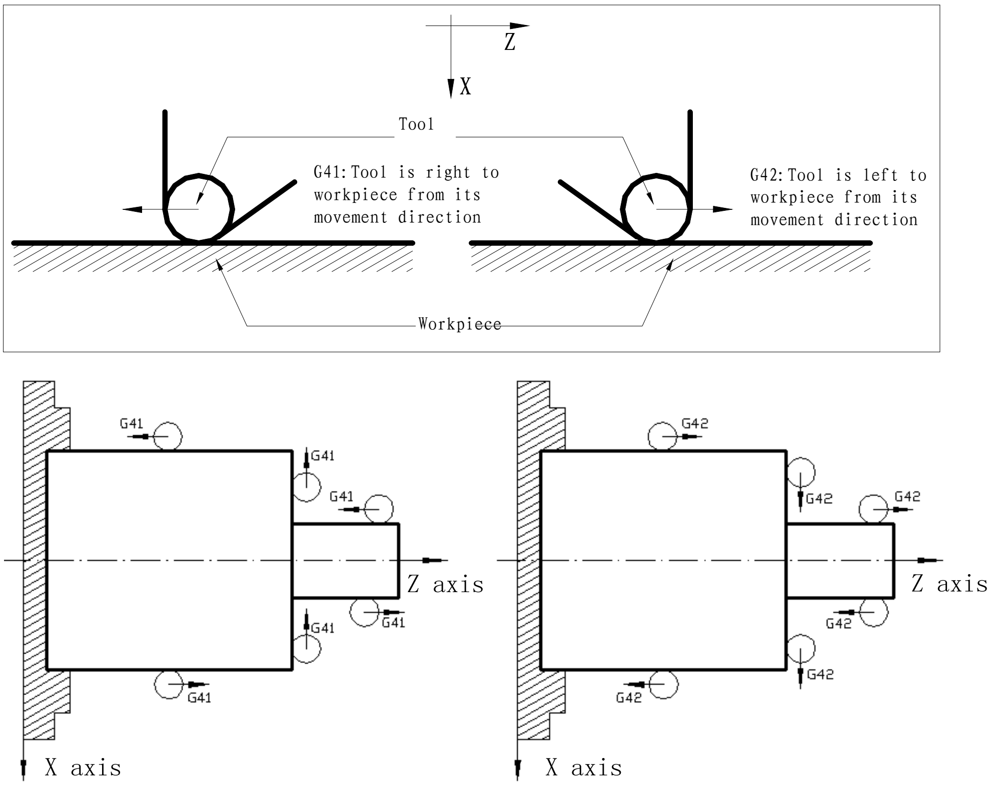


Fig. 4-6-1-4-2 Tool nose radius compensation in front tool post coordinate system

4.6.1.5 Inside and Outside

Compensate the tool nose radius; the corners of the former and the latter programmed path are different, and the tool compensation paths are different. Therefore, the point of intersection of two movement blocks at the side of work piece is greater or is just 180°, it is called as “inside”. And the angle is between 0～180°, it is “outside”.

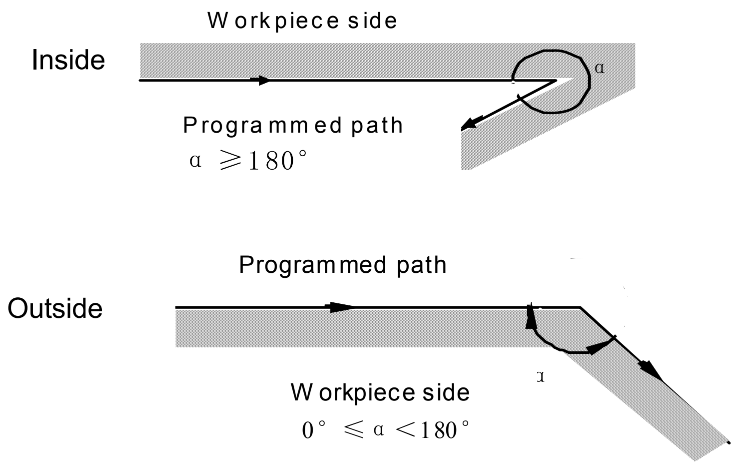


Fig. 4-6-1-5-1 Inside and outside

4.6.1.6 G41, G42 and G40 Command Format

Command format：

$$\left\{\begin{matrix} G40 \\ G41 \\ G42 \end{matrix}\right\} \left\{\begin{matrix} G00 \\ G01 \end{matrix}\right\} X\text{—}\ Z\text{—}$$

Note 1 : G40, G41 and G42 all are mode G codes.

Note 2: After setting the tool compensation, G02 or G03 code can follow G41/G42.

4.6.2 Tool Compensation

4.6.2.1 Analyzing the Tool Nose Radius Compensation

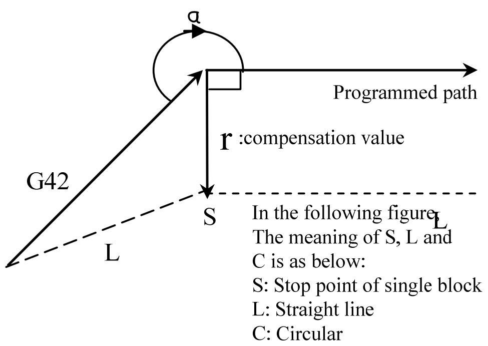
Generally, realizing the tool radius compensation requires three steps: setting, executing and cancelling the tool compensation.

1. Setting the tool compensation

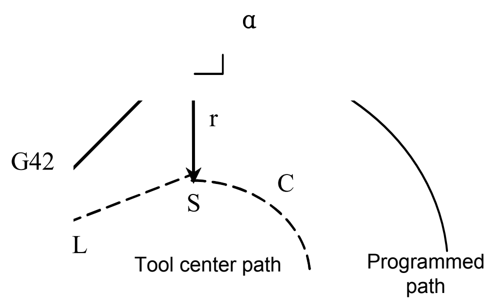
Change the offset cancelling mode into the offset mode; it is called setting the tool compensation. About the details of the tool compensation, it is shown in **Fig.4-6-2-1-1**：

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

（i）Straight line— straight line

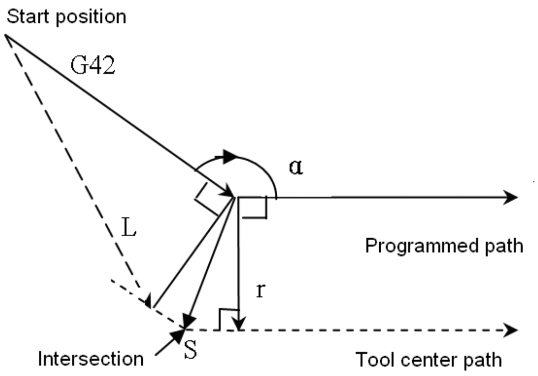


（ii）Straight line— arc



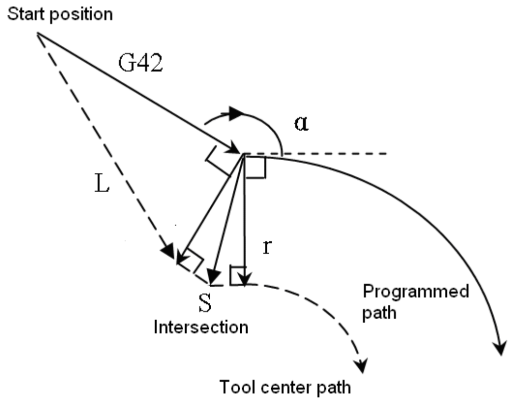
b) Tool movement around the outside of a corner at an obtuse angle (180°＞α≥90°）

(i) Straight line--straight line



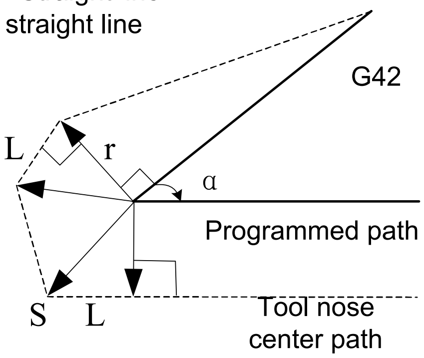
Remark: An intersection is a position at which the compensated paths of two blocks intersect with each other.

(ii) Straight line—arc

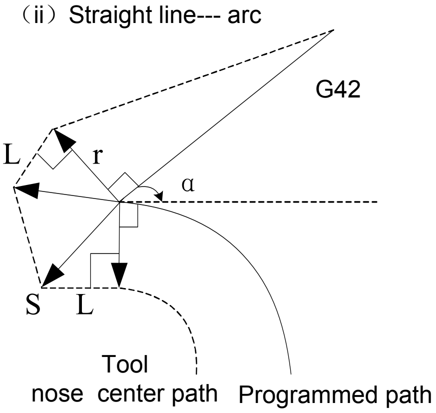


(c) Tool movement around the outside of an acute angle (α<90°)

(i) Straight line—straight line



(ii) Straight line--- arc



(d) Tool movement around the outside linear→linear at an acute angle less than 1 degree (α<1°)

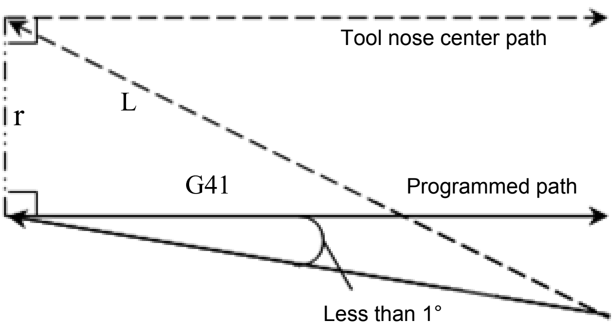


Fig. 4-6-2-1-1 Setting the tool compensation

Note 1: During setting the tool compensation, without specifying the compensation number or the tool compensation number is 0, #036 alarm occurs.

Note 2: During setting the tool compensation, it requires G0 or G1 to be executed; if the circular is commanded, #036 alarm occurs.

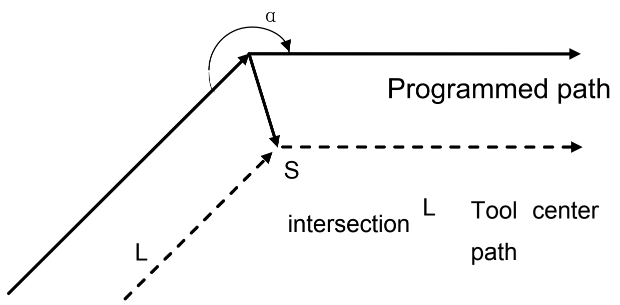
2. Executing the tool compensation

After setting the tool compensation, the offset path before cancelling the tool compensation is called executing the tool compensation.

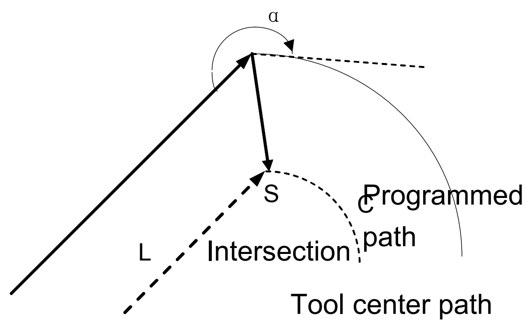
About the details of the tool compensation, it is shown in **Fig.4-6-2-1-2** and **4-6-2-1-3:**

(a) Tool movement around the inside of a corner（α≥180°）

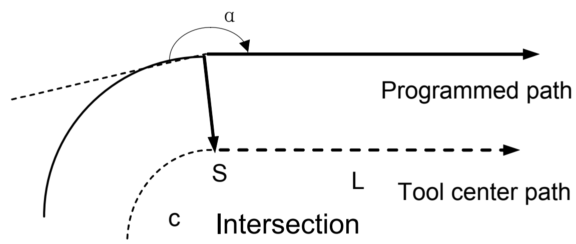
Straight line---- straight line



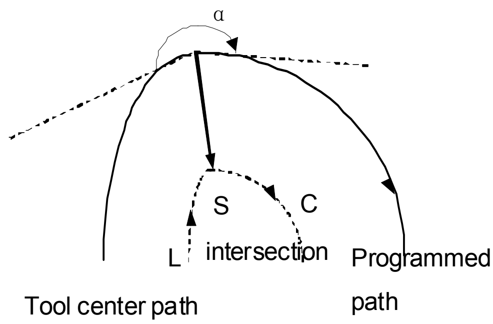
Straight line----arc



Arc---Straight line

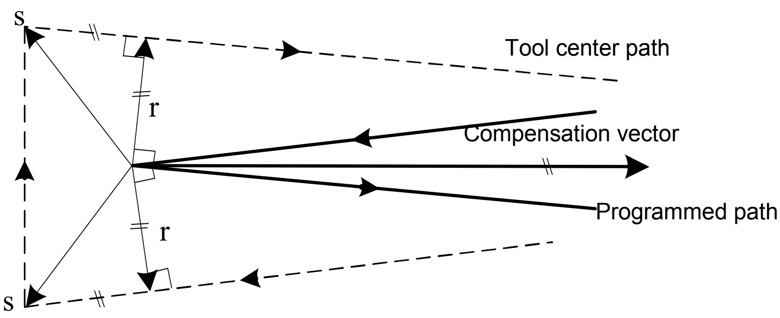


Arc---arc



（i）Straight line---straight line

（V）Inside machining is less than 1° and compensation vector is magnified.

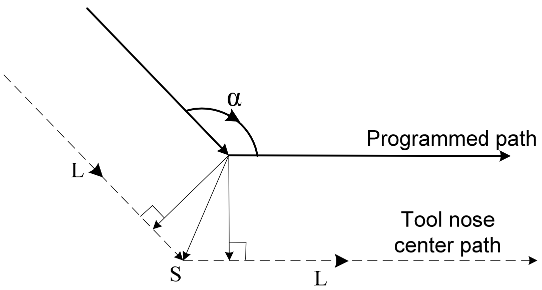


Consider the following situations through the same method:

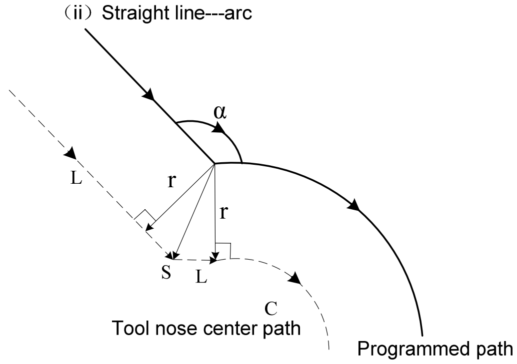
（ii）Arc--- straight line
（iii）Straight line---Arc
（iv）Arc---Arc

(b) Tool movement around the outside corner at an obtuse angle （180°＞α≥90°）

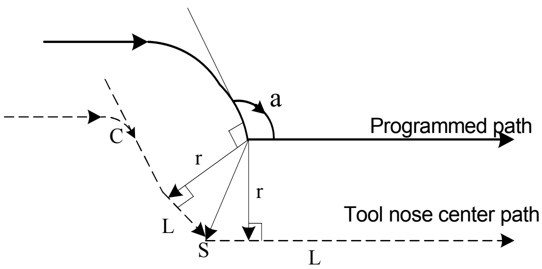
（i）Straight line---straight line



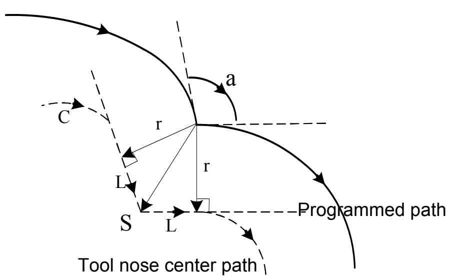
（ii）Straight line---arc



（iii）Arc---straight line



（iv）Arc---arc



(c) Tool movement around the outside corner at an acute angle（α＜90°）

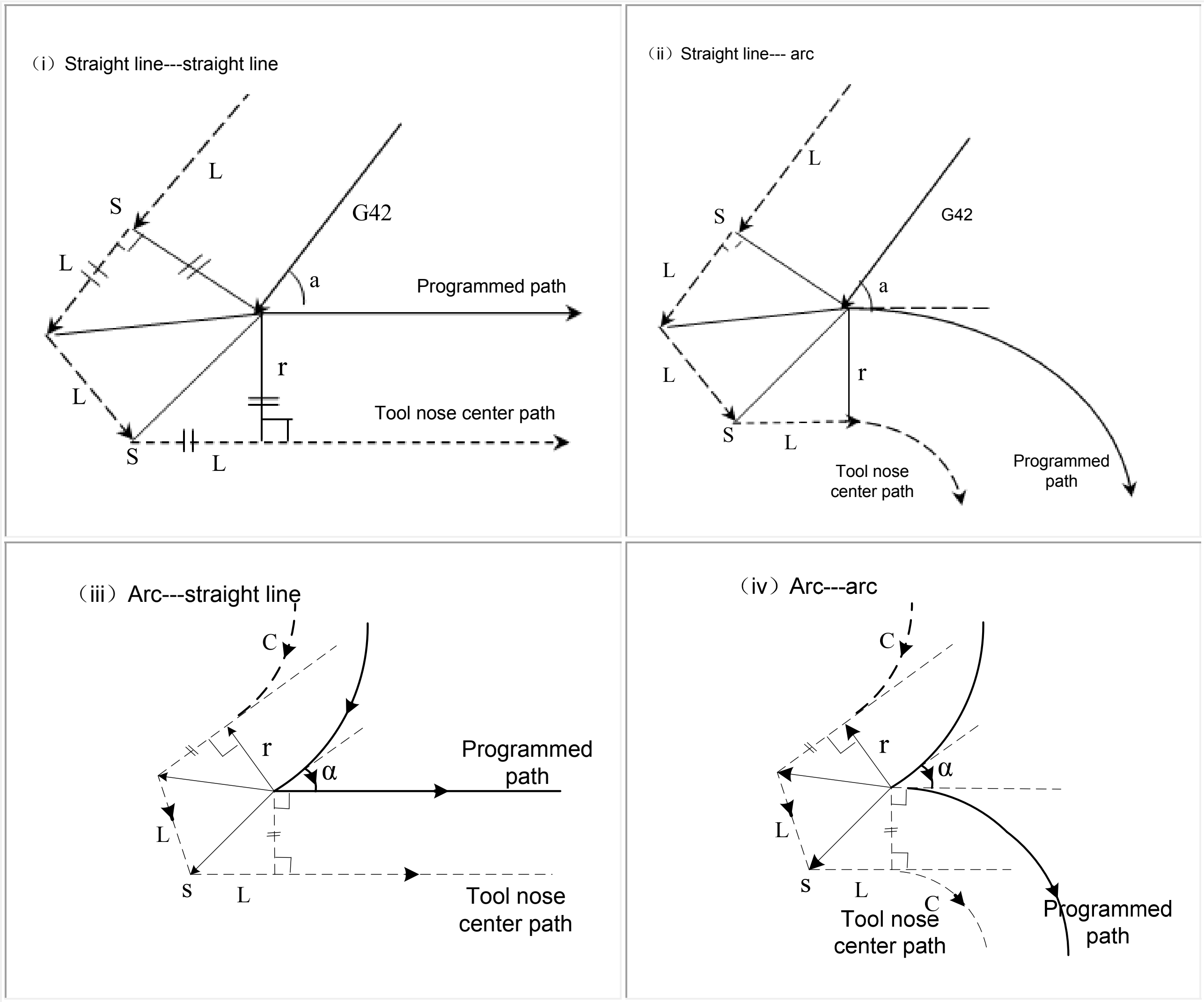
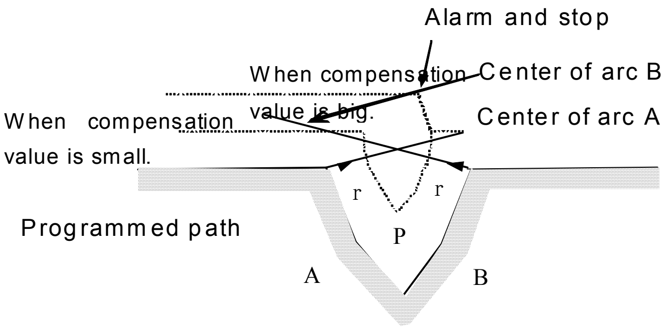


Fig. 4-6-2-1-2 executing tool compensation

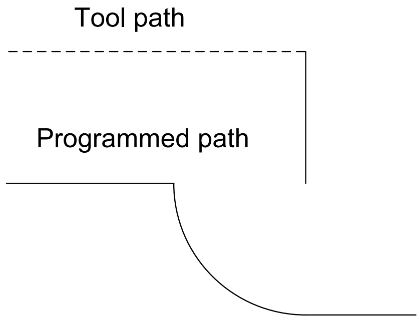
(d) Special situations

1) Without intersection



In left figure, when the radius value is small, there exists intersection in compensation path of arc; while the radius changes bigger, perhaps the intersection does not exist and the system alarms (P/S33).

2) The center of arc is identical with the start position or the end position



When the tool radius is too big, the start or finish point of arc coincides with the center of a cycle. The system alarms P/S38, and the programs are shown as below:

G01 W20；

G02 U30 W15 R15；

G01 W20；

Fig. 4-6-2-1-3 executing tool compensation ②

Ⅰ Programming

3. Cancelling the tool compensation

In the compensation mode, the block can satisfy any conditions as below, the system enters the compensation cancel mode, and the movement of block is called cancelling tool compensation.

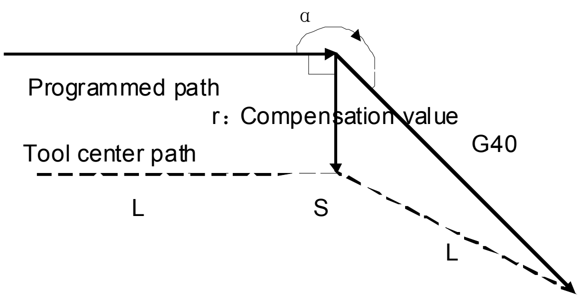
(a) Use code G40, C tool compensation is cancelled, and during the execution of tool compensation cancel, the circular codes (G02 and G03) are not allowed to use; otherwise, it alarms (NO.34) and the tool stops.

(b) The tool radius compensation number is specified to 0.

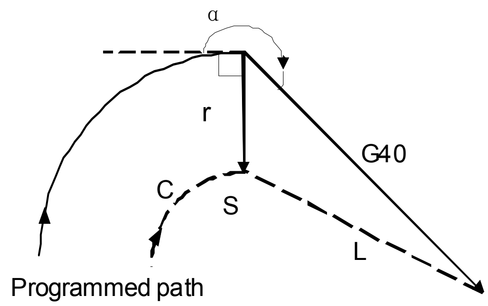
The details of tool compensation cancel are shown in the following figures:

(a) Tool movement around the inside of a corner（α≥180°）

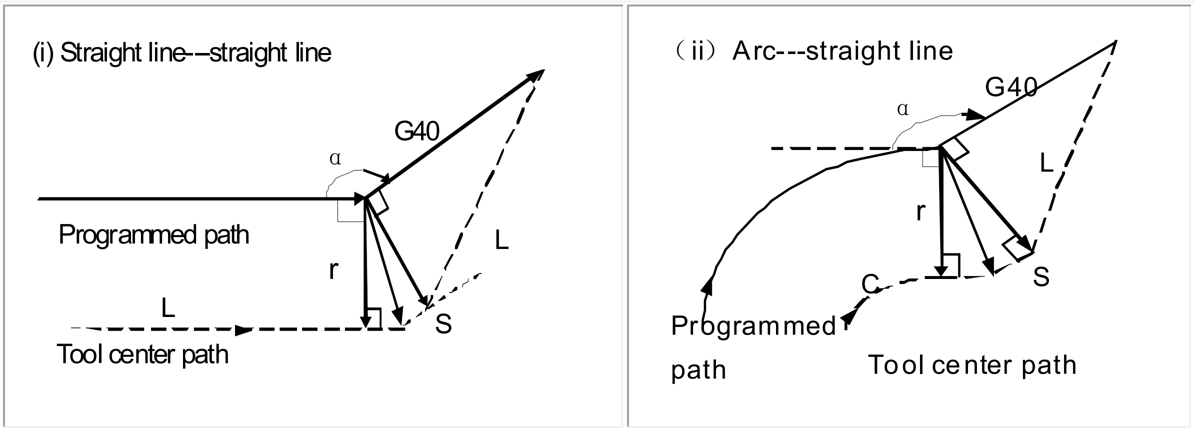
(i) Straight line---straight line



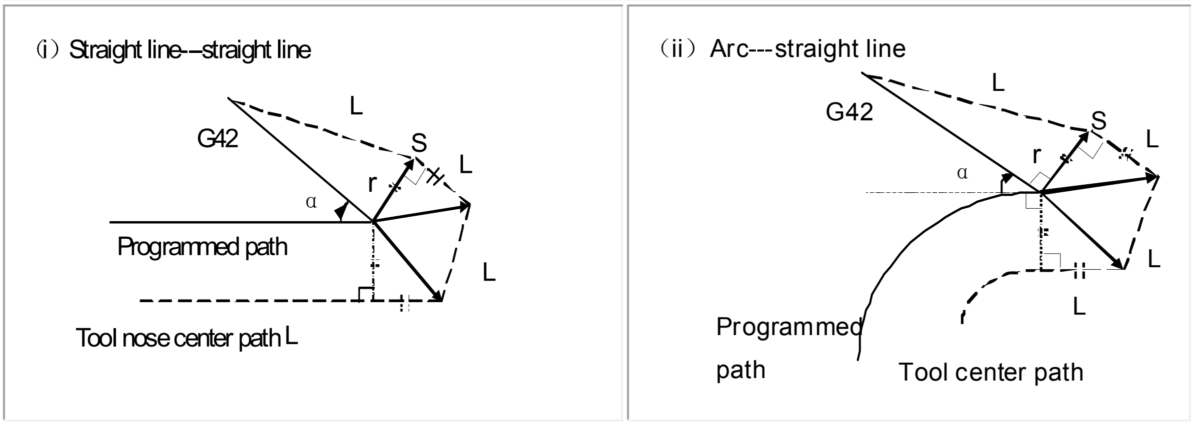
（ii）Arc---straight line



(b) Tool movement around the outside corner at an obtuse angle（180°＞α≥90°）



（c）Tool movement around the outside corner at an acute angle （α<90°)



（d）Tool movement around the outside linear→linear at an acute angle less than 1 degree（α<1°).

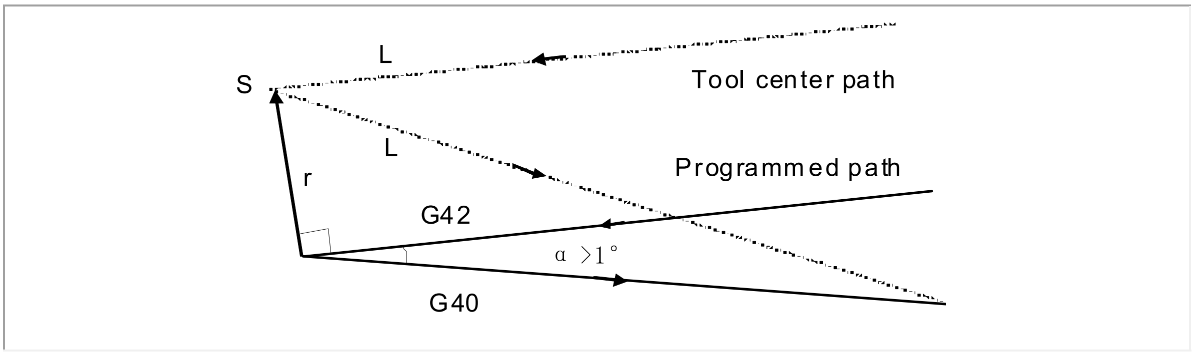


Fig. 4-6-2-1-4　 Cancel tool compensation

4.6.2.2　Change Compensation Direction in the Tool Compensation

Tool radius compensation G codes (G41 and G42) determine the direction of compensation; signs of compensation amount are as below:

Table 4-6-2-2-1

Sign / G code	+	-
G41	Left side compensation	Right side compensation
G42	Right side compensation	Left side compensation

In the special situation, the direction can be changed in the compensation mode, but it's not allowed to change at the block beginning. Once the compensation direction changes, the concept of inside and outside doesn't exit. Assume the following compensation amount as the positive.

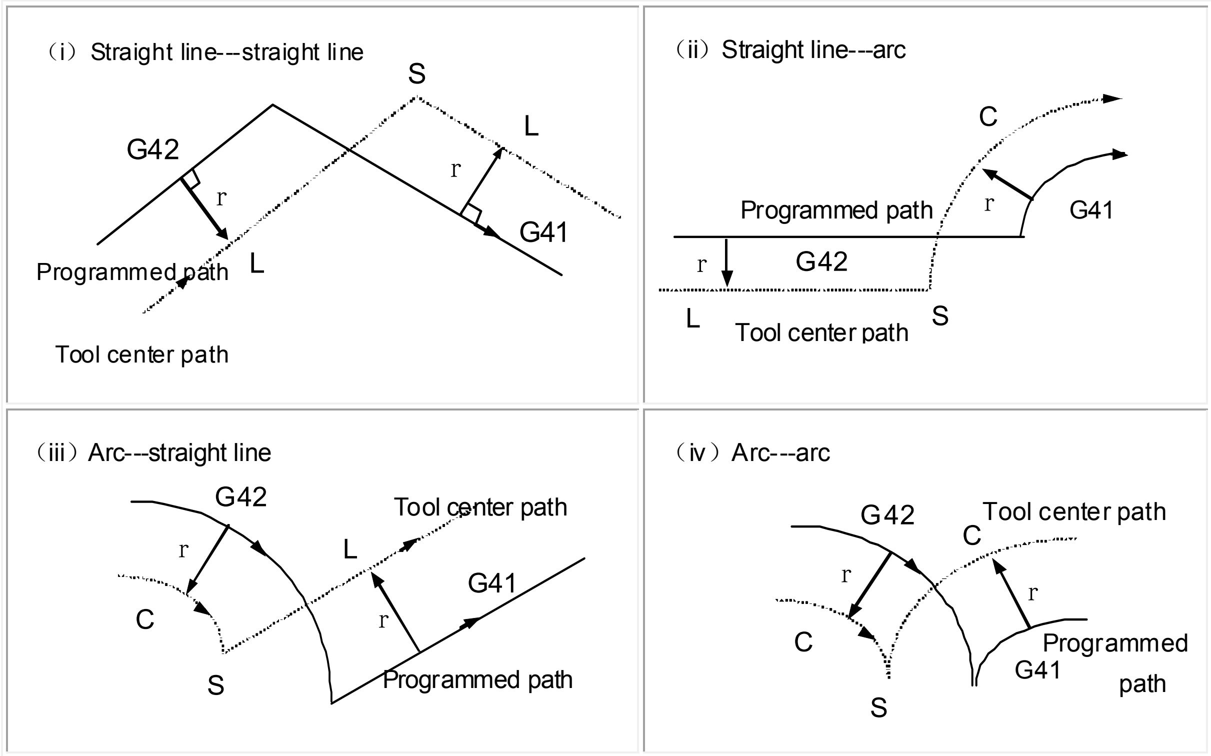


Fig. 4-6-2-2-1 Change in compensation direction during tool compensation

If compensation is executed normally without an intersection, G41 and G42 change the offset direction from block A to B. If the intersection of offset path is not required, the start point of block B is vertical to the vector of block B.

i) Linear----linear

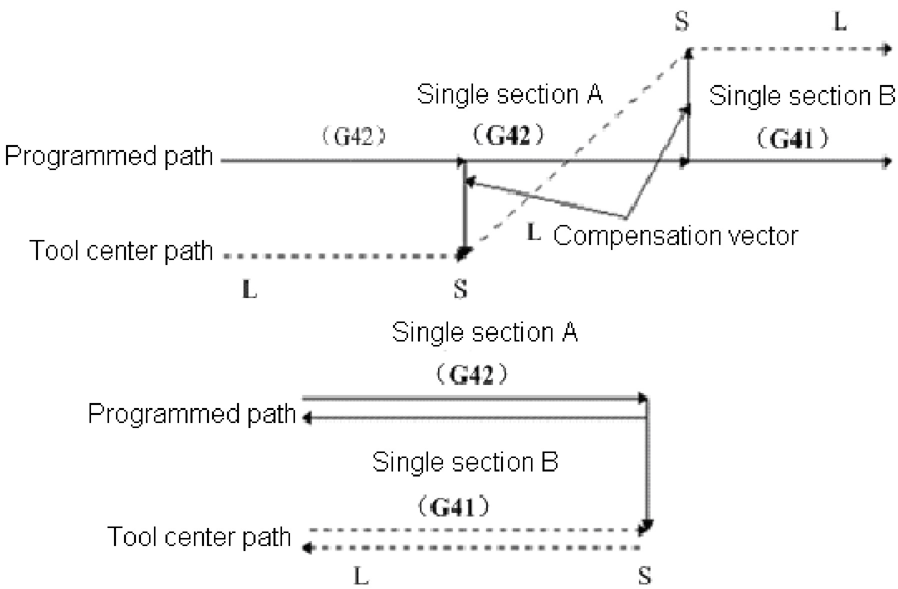


Fig. 4-6-2-2-2 Linear----linear

ii) Linear----circular

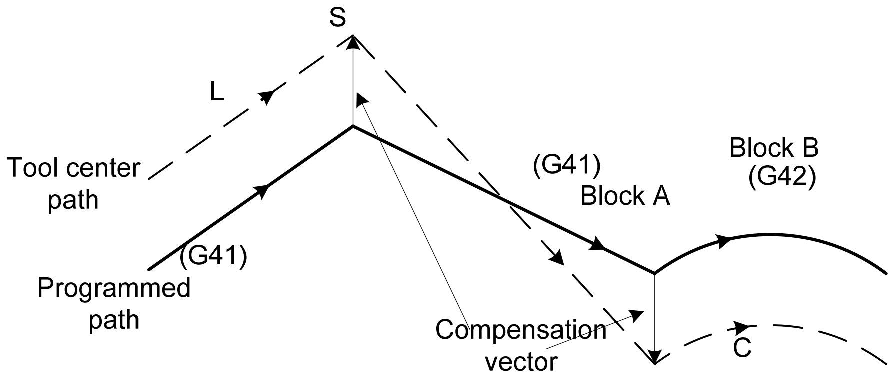


Fig. 4-6-2-2-3 Linear----circular, without intersection (change the compensation direction)

4.6.2.3 Cancel Tool Compensation Temporarily

In the compensation mode, if the following codes are specified, the compensation vector will be cancelled temporarily. Then, the compensation vector will automatically recover. Then, it is different from the compensation cancel mode, the tool can directly move from the intersection to the code point of cancelling compensation vector. After the compensation mode recovers, the tool moves to the intersection directly.

1. Setting a coordinate system（G50）

N1 T0101；
N2 G42 G00 X0 Z0；
N3 G01 U-30 W30；
N4 U30 W30；
N5 G50 X0 Z60；
N6 G01 U-30 W30；
N7 G01 U30 W30；
N8 G00 X0 Z0；
N8 M30；

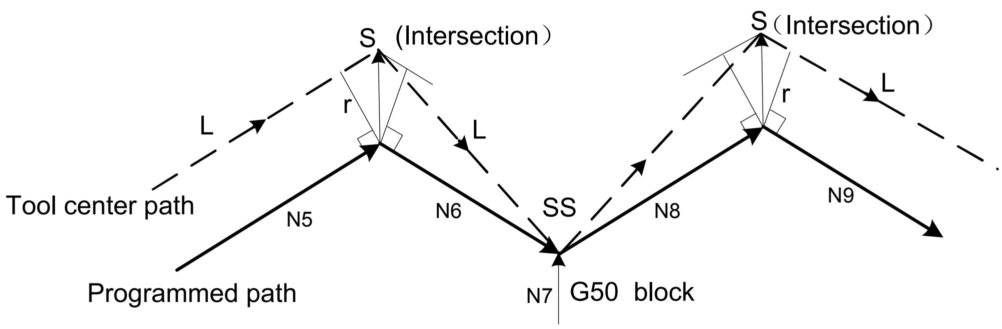


Fig. 4-6-2-3-1 Cancel the tool compensation temporarily①

Note: SS represents the point which the tool stops two times in a single block

2. G90, G92 and G94 fixed cycle, G71~G76 fixed cycle, thread codes G32/G33/G34.

Programming

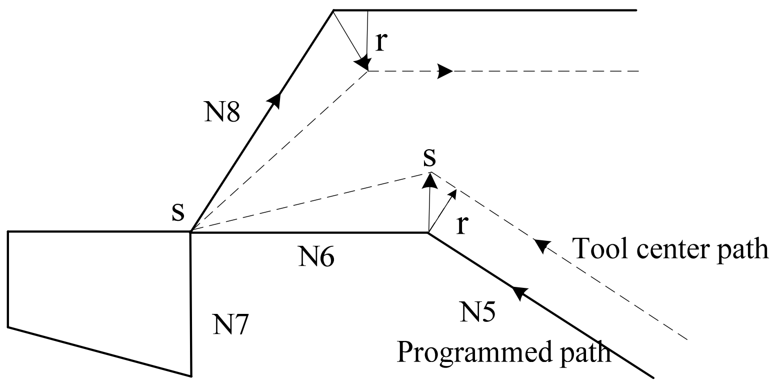


Fig. 4-6-2-3-2 Cancel tool compensation temporarily②

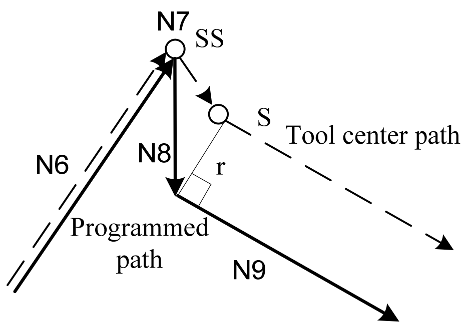
N1 T0101
N2 G0 X100 Z100
N3 G0 X0 Z0
N4 G42 G90 X-20 W-50 F500 （Cancel the tool compensation temporarily）
N5 G0 X50 Z50 （Recover the tool compensation）
N6 G0 X100 Z100
N7 M30

Note: G90/G94 tool compensation cancel temporarily will realize only when G41/G42 shares with the same block of G90/G94; otherwise, the system will automatically process G90/G94 as the normal tool offset compensation. About the details, refer to G90/G94 tool nose radius compensation.

4.6.2.4 Non-Movement Codes in Tool Compensation

1. Non-movement codes at the beginning of compensation

If the tool doesn't move at the beginning of compensation, the vector of compensation is not be created.

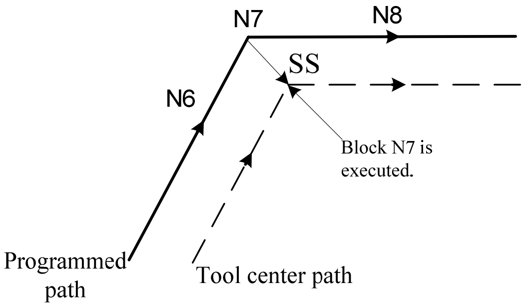


N1 T0101；
N2 G0 X0 Z0；
N3 G01 U-30 W20 F500；
N4 G42 U0；
N5 U30；
N6 U20 W20；
N7 G40 G0 X100 Z100；
N8 M30；

Fig. 4-6-2-4-1 Non-movement codes at the beginning of the tool compensation

2. Non-movement codes in the compensation mode

In the compensation mode, when only one block free of the tool movement is commanded, the vector and the tool center path are same as those of the block which isn't commanded. (refer to Section **4.6.2.1,** executing tool compensation. The block without the tool movement is executed at the stop point of single block.

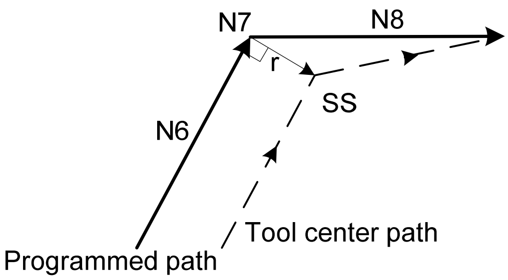


N3 T0101；
N4 G0 X100 Z100；
N5 G41 G01 X0 Z0；
N6 U-30 W20；
N7 G04 X5；
N8 W30；
N9 G40 G0 X100 Z100；
N10 M30；

Fig. 4-6-2-4-2 Non-movement codes during tool compensation

3. Non-movement codes during cancelling compensation

When the block, which is commanded with the block of cancelling compensation, is free of the tool movement, it will form the length as the compensation amount. The direction is vertical to the vector of the movement direction in the former block, and the vector will be cancelled in the next movement code.



N3 T0101；
N4 G0 X100 Z100；
N5 G41 G01 X0 Z0 F500；
N6 U-30 W20；
N7 G04；
N8 G0 X100 Z100；
N9 M30；

Fig. 4-6-2-4-3 Non-movement codes during cancelling tool compensation

4.6.2.5 Tool Compensation Interference Check

The tool overcut is called as "interference". The interference check of tool compensation can check the situation of tool overcut in advance, that is to say, even if the overcut does not occur, the system also checks the interference.

(a) Basic conditions of interference

(1) The path direction of tool and that of program are different. (The angle between paths is between 90°and 270°.)

(2) During machining the circular, besides the above condition, the angle between the start position and the end position in the center of tool path has big difference with that of start position and end position in the programmed path (More than 180°).

Example ①

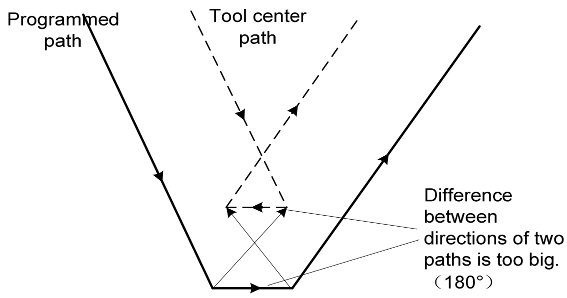


Fig. 4-6-2-5-1 Tool compensation interference ①

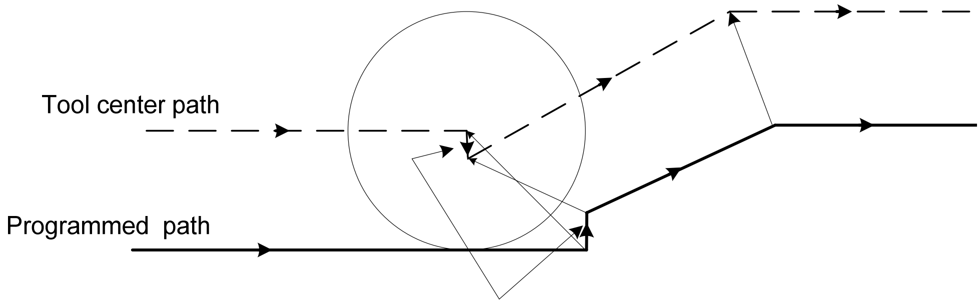


Fig. 4-6-2-5-2 Tool compensation interference ②

(b) Interference example

(1) One smaller depth is less than compensation amount.

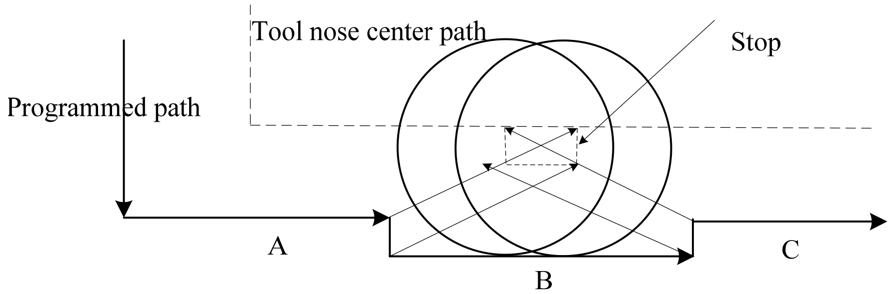


Fig. 4-6-2-5-3 Tool compensation interference example ①

Program:

N1 T0101； (R<=10)

N2 G0 X0 Z30；

N3 G42 G01 X50 Z0 F500；

N4 U50；

N5 W20；

N6 U10；

N7 W20；

N8 U-10；

N9 W20；

N10 G40 G0 X0 Z30；

N11 M30；

In the above programs, the tool nose radius compensation value of #01 tool is **R<=10**; when **R>10**, the system will alarm the interference because the program direction in block B is opposite with that of the tool radius compensation.

(2) Concave depth is less than the compensation amount.

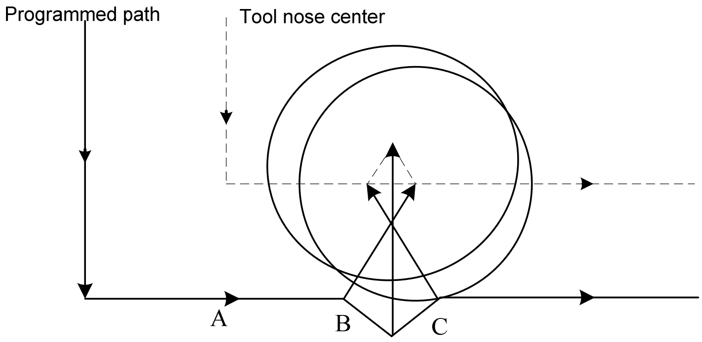


Fig. 4-6-2-5-4 Tool compensation interference example②

Program :

N1 T0101； (R<=25)
N2 G0 X0 Z30；
N3 G42 G01 X50 Z0 F500；
N4 U50；
N5 W20；
N6 U10 W10；
N7 U-10 W10；
N8 W20；
N9 G40 G0 X0 Z30；
N10 M30；

In the above programs, the tool nose radius compensation value of #01 tool is **R<=25**; when **R>25**, the system will alarm the interference because the program direction of block c is opposite with that of the tool radius compensation path.

4.6.2.6 Tool Nose Radius Compensation in G90/G94

1. About each cycle path, the tool nose center path is generally parallel with the programmed path.

① **G90**

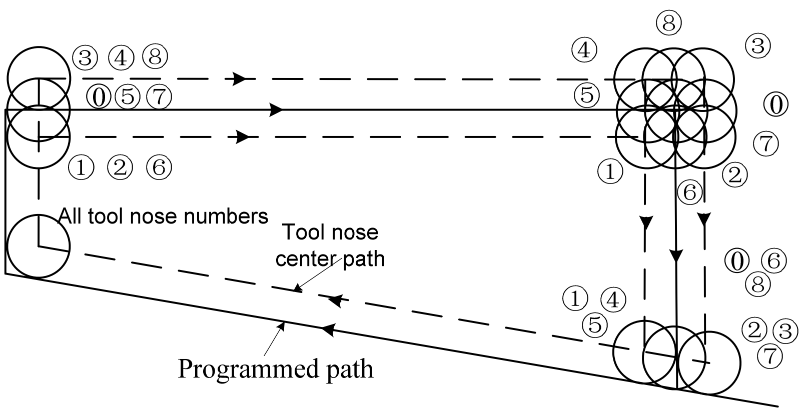


Fig. 4-6-2-6-1 G90 tool nose radius compensation

② **G94**

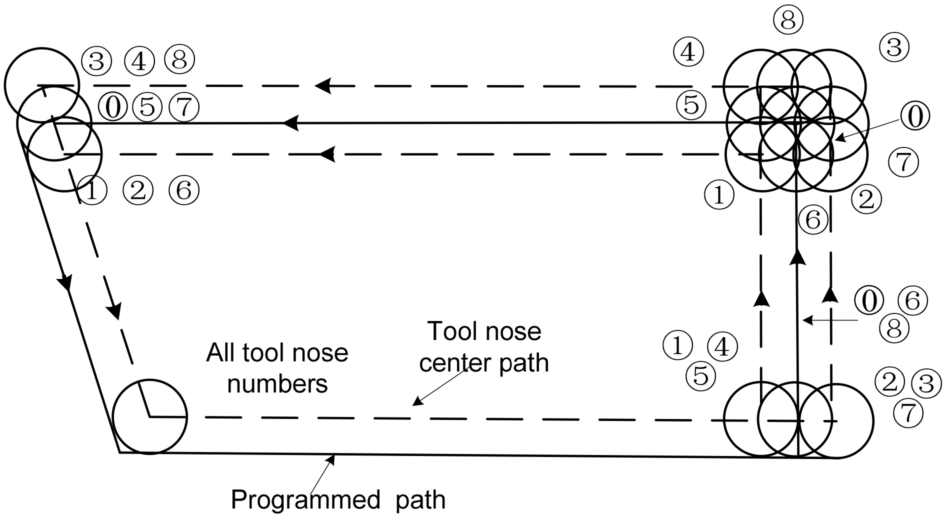


Fig. 4-6-2-6-2 G94 **tool nose radius compensation**

(b) G41 or G42 mode is used, and the offset direction is shown below.

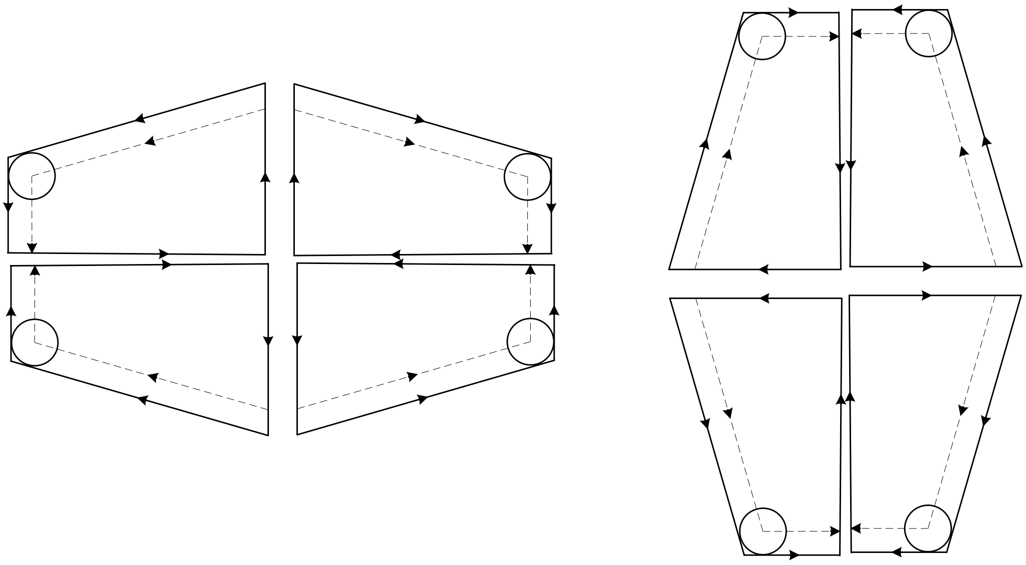


Fig. 4-6-2-6-3 G90/G94 **Tool nose radius compensation**

4.6.2.7 Tool Nose Radius Compensation in G70

In the finishing cycle (G70), the system can realize the tool nose radius compensation, and the tool center path moves along with the finishing path and automatically offsets one compensation value. To realize G70 tool nose radius compensation, G70 can share with G41/G42 for execution or specify G41/G42 in finishing cycle block.

4.6.3 Notes for Tool Compensation C

1. When the system continuously specifies 30 or more non-movement codes in compensation, an

alarm occurs. For example:

N1 M05 ; .... M code output
N2 S21 ; .... S code output
N3 G04 X10 ; .... Dwell
……
N29 G01 U0 ; .... Movement distance 0
N30 G98 ; .... Only G code.

2. In MDI mode, it executes the block rather than the tool nose radius compensation.
3. After power on or executing M30, the system enters the cancel mode at once. The program must end in the cancel mode. Otherwise, the tool can not position at the end position, the tool will stop in the position keeping a vector distance from the end position.
4. For setting and cancelling the tool nose radius compensation, only G00 or G01 code can be used rather than the circular codes (G02 or G03); otherwise, it will alarm (NO.34).
5. Before calling the subprogram, namely, before executing M98, the system must be in the compensation cancel mode. After entering the subprogram, the offset can start, but it must be in the compensation cancel mode before returning the main program (before M99); otherwise, an alarm occurs.
6. If the compensation value (R) is negative, G41 and G42 interchange in the program. If the tool center moves along the outside of work piece, the tool compensation will move along with the inside, vise versa. Because the compensation value code changes, the tool nose offset direction also changes, but the assumed tool nose direction remains unchanged. Do not change at random.
7. Generally, in compensation cancel mode or tool change, the compensation value should be changed. If the value is rewritten in the compensation mode, only new compensation value is valid after tool change.
8. When the tool compensation programs are executed, mistake or alarm occurs due to some reasons; while G codes remain same, for example, G41 is still G41, G42 is G42; if it requires cancelling the tool compensation, it can input G40 to run in MDI mode, then the tool compensation can be cancelled.

4.6.4 Machining Examples of Tool Compensation C

Tool compensation C example①：

Machine the parts shown in the following Fig. **4-6-4-1**, the part dimension is shown in the figure, the radius of tool nose R=1, and it is the first tool.

Ⅰ Programming

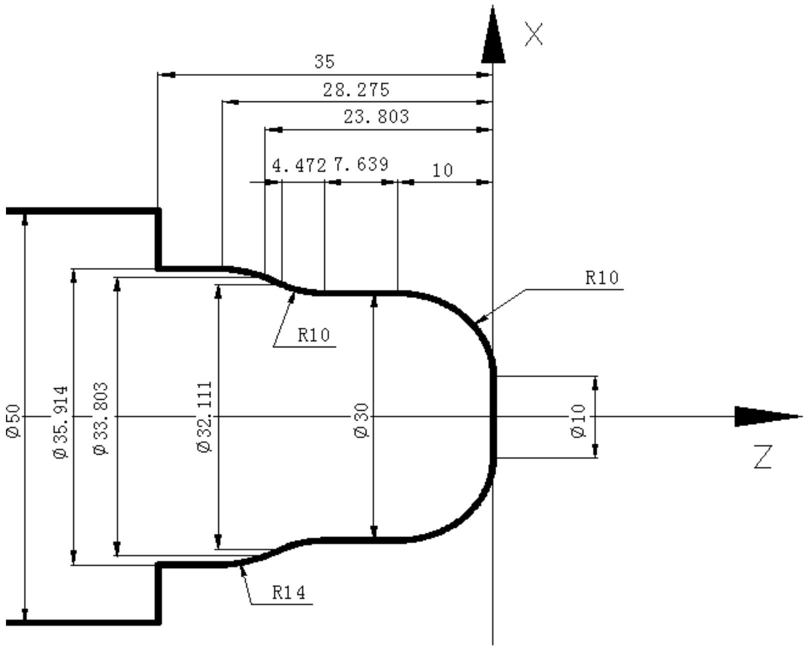


Fig. 4-6-4-1 Tool compensation C example ①

Program:

O0001；
N010 G0 X100.0 Z100.0；　（Position to a safety position）
N020 M3 S300；　（Spindle rotates CCW, speed:300r/min）
N030 M8；　（Cooling ON）
N040 T0101；　（Change No.1 tool and execute its tool compensation）
N050 G00 X10.0 Z10.0；　（Rapid position, approach to the work piece）
N060 G42 G1 Z0 F80；　（Start to execute the tool nose radius compensation）
N070 G3 X30 Z-10 R10；
N080 G1 Z-17.639；
N090 G2 X32.111 Z-22.111 R10；
N100 G1 X33.803 Z-23.803；
N110 G3 X35.914 Z-28.275 R14；
N120 G1 Z-35；
N130 X50；
N140 G40 G0 X80 Z80；　（Cancel the tool nose radius compensation）
N150 M09；　（Cooling OFF）
N160 G00 X100.0 Z100.0 T0200；　（Rapid return to the safety position, change the referebce tool and clear the tool offset）
N170 M30；　（Program end）

Tool compensation C example②：

Machine the parts which are shown in Fig. **4-6-4- 2**, the part dimension is shown in the figure, the radius of tool nose R=1, and it is the first tool.

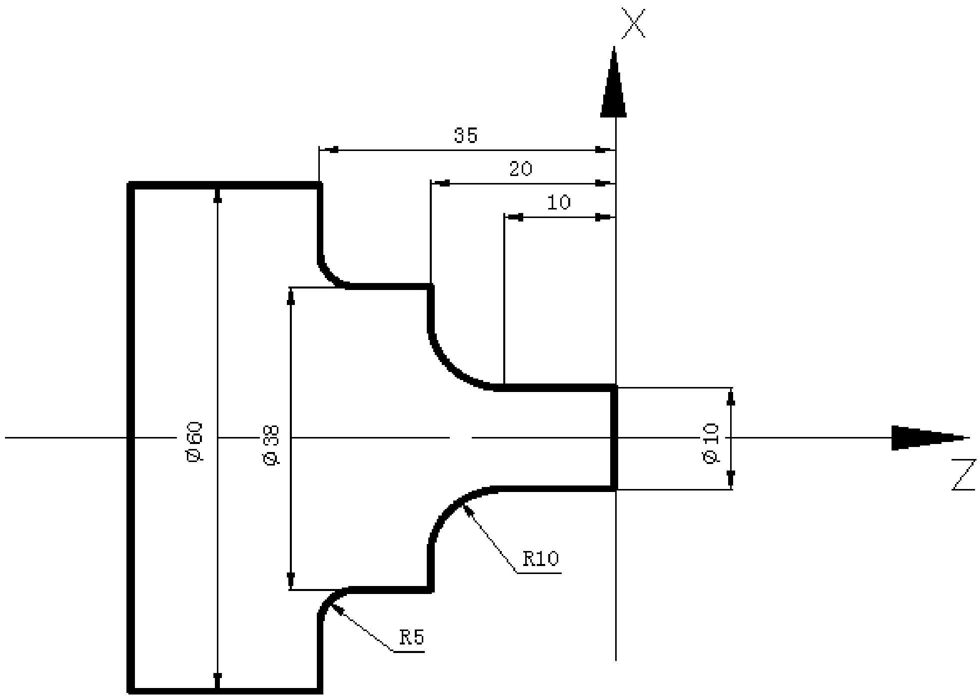


Fig. 4-6-4-2 Tool compensation C example②

O0002
NI G0 X100 Z100;
N2 M3 S800;
N3 M8;
N4 T0202;
N5 G0 X70 Z10;
N6 G71 U3 R1;
N7 G71 P8 Q14 U0 W0 F120;
N8 G0 X10;
N9 G1 Z-10 F80;
N10 G02 X30 W-10 R10;
N11 G1 X38;
N12 Z-30;
N13 G02 X48 W-5 R5;
N14 G1 X60;
N15 G0 X100 Z80;
N16 M3 S300;
N17 T0101;
N18 G0 X70 Z10;
N19 G42 G70 P8 Q14; (G42 and G70 share the same block to execute the tool nose radius compensation)
N20 G40 G0 X80 Z50; (Cancel the tool nose radius compensation)
N21 G0 X100 Z100 T0200;
N22 M30;

Ⅰ Programming

Tool compensation C example③：

Machine the parts shown in Fig. **4-6-4-3** , the part dimension is shown in Fig. **4-6-4-3**, the radius of tool nose R=1, and it is the first tool.

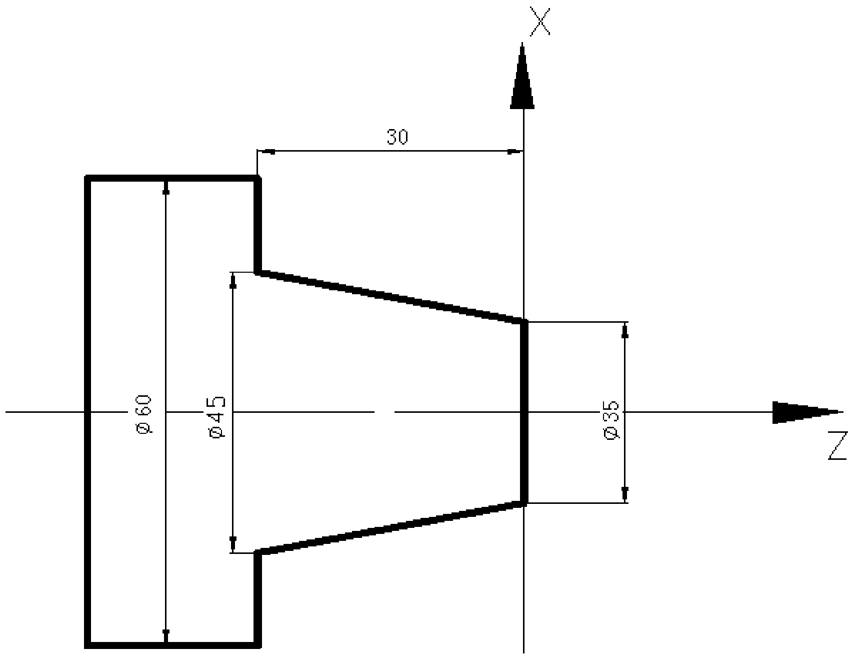


Fig. 4-6-4-3 Tool compensation C example ③

O0003
N1 G0 X100 Z100；
N2 M3 S800；
N3 M8；
N4 T0101；
N5 G42 G0 X70 Z10； （Start to execute tool nose radius compensation）
N6 G90 X45 Z-30 R-5 F80；
N7 G40 G0 X80 Z80； (Cancel tool nose radius compensation)
N8 G0 X100 Z100 T0100；
N9 M30；

4.7 Macro Function G Code

4.7.1 User Macro Program

Some function which is achieved by one group of codes can store in the memorizer in advance, and one code represents these functions. In the programs, as long as the representative codes are written, these functions can be realized. This group of codes is called as the user macro body, and the representative codes are called as “the user macro code”. The user macro body sometimes is abbreviated as the macro. And the user macro code is also called as the macro calling code.

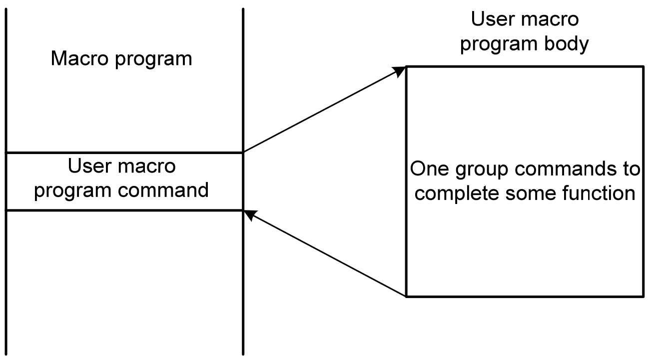


Fig. 4-7-1-1

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

4.7.2 Macro Variable

In the user macro, the normal CNC codes can be used, and the variable, calculation and transfer codes can also be used.

The user macro the program starts with a program, and ends with M99.

```
O0066;                     Program number
G65  H01  ……;              Operational command
G90  G00  X#101  ……; CNC command using variables
……
……
……
G65  H82  ……;              Transfer command

……
……
M99;                       User macro program body ends
```

Fig. 4-7-2-1 （composition of user macro program body）

1. The usage methods of variables:

The variable can code the parameter value of the user macro body. The variable value can be set by the main program assignment or by LCD/MDI , or assign the count value during executing the user macro body.

Many variables can be used and they are differed by variable numbers.

(1) Expression of variables

The variable number behind # can indicate the variable, the format is shown below:

#i (i = 1， 2， 3， 4 ……)

(Example) #5， #109， #1005

(2) Quotation of variables

Variables can replace the numerical values after the address.
(Example) F#103 #103 = 15: it is the same with F15.
G#130 #130 = 3: it is the same with G3.

Note 1: Address O and N can not quote the variable. O#200 and N#200 can not be used for programming.
Note 2: If it exceeds the maximum code value specified by the address, it can not be used. When #30=120, M#30 exceeds the max code value
Note 3: Display and setting the variable value: the variable value can display on LCD screen, the variable value can also be set by keys.

2. Variable types

According to the different variable numbers, the variables can be differed from the common and the system variables, and their purposes and characteristics are different.

（1）Null variable#0： (the variable is always null and no value is assigned to the variable)

（2）Local variables #1～#50：They can only be used for data storage in a macro, such as the results of operations. whether the local variables are cleared or not after reset is set by bit parameter NO:52#7. When a macro program is called, arguments are assigned to local variables.

（3）Common variable #100～#199，#500～#999：whether to clear the common variables #100～#199 is set by NO：52#6 after reset or emergency stop.

The common variable is common in the main program and in each user macro called by the main program, that is to say, variable #i used by some user macro and #i used by other macro programs are same. Therefore, the common variable #i of calculation result in some macro can be also used in other macro programs.

The purpose of the common variable, which is not stipulated in the system, can be freely used by the user.

Table 4-7-2-1

Variable number	Variable type	Function
＃100～＃199	Common variables	They are cleared at power-off, and all are initialized to “null” at power-on
＃500～＃999		Data is saved in files and it will not be lost even if the power is turned off.

（4）System variables: They are used for reading and writing a variety of CNC data, which are shown below:

1）Interface input signal #1000 --- #1015 (read signal input to system from PLC by bit, i.e. G signal)
#1032 (read signal input to system from PLC by byte, i.e., G signal)

2）Interface output signal #1100 --- #1115 (write signal output to PLC from the system by bit, i.e. F signal)
#1132 (write signal output to PLC from the system by byte, i.e. F signal)

3）Tool length compensation value #1500 --- #1755 (read-write)

4）Length wear compensation value #1800 --- #2055 (read-write)

5）Tool radius compensation value #2100 --- #2355 (read-write)

6）Radius wear compensation value　#2400 --- #2655 (read-write)
7）Alarm　#3000
8）User data table　#3500 --- #3755 (read only, must not write)
9）Modal message　#4000 --- #4030 (read only, must not write)
10）Position message　#5001 --- #5030 (read only, must not write)
11）Workpiece zero offset amount　#5201 --- #5235 (read-write)
12）Additional workpiece coordinate　#7001 --- #7250 (read-write)

3．Explanation for system variables

1）Modal message

Table 4-7-2-2

No.	Function	Group
#4000	G04,G28,G31,G50,G65,G70,G71,G72,G73,G74,G75,G76	Group 00
#4001	G00,G01,G02,G03,G32,G34,G90,G92,G94	Group 01
#4002	G96,G97	Group 02
#4003	To extend	Group 03
#4004	To extend	Group 04
#4005	G98，G99	Group 05
#4006	G20,G21	Group 06
#4007	G40,G41,G42	Group 07
#4008	To extend	Group 08
#4009	To extend	Group 09
#4010	To extend	Group 10
#4011	To extend	Group 11
#4012	To extend	Group 12
#4013	To extend	Group 13
#4014	G54,G55,G56,G57，G58,G59	Group 14
#4015	To extend	Group 15
#4016	G17,G18,G19	Group 16
#4017	To extend	Group 17
#4018	To extend	Group 18
#4019	To extend	Group 19
#4020	To extend	Group 20
#4021	To extend	Group 21
#4022	D	
#4023	H	
#4024	F	
#4025	M	
#4026	S	
#4027	T	
#4028	N	
#4029	O	
#4030	P(currently selected additional workpiece coordinate system)	

Note 1: P code indicates the current selected additional workpiece coordinate system.
Note 2: When G#4002 code is being executed, the value obtained in #4002 is 17, 18 or 19.
Note 3: The modal message can be read but not written.

2）Current position message

Table 4-7-2-3

Variable number	Position message	Relative coordinate system	Reading operation during moving	Tool offset value
#5001	X block end position (ABSIO)	Workpiece coordinate system	allowed	Tool nose position not involved（Position instructed by program）
#5002	Y block end position (ABSIO)			
#5003	Z block end position (ABSIO)			
#5004	The 4th axis block end position (ABSIO)			
#5006	X block end position (ABSMT)	Machine coordinate system	unallowed	Tool reference Position involved（Machine coordinate）
#5007	Y block end position (ABSMT)			
#5008	Z block end position (ABSMT)			
#5009	The 4th axis block end position (ABSMT)			
#5011	X block end position (ABSOT)	Workpiece coordinate system		
#5012	Y block end position (ABSOT)			
#5013	Z block end position (ABSOT)			
#5014	The 4th axis block end position (ABSOT)			
#5016	X block end position (ABSKP)		allowed	
#5017	Y block end position (ABSKP)			
#5018	Z block end position (ABSKP)			
#5019	The 4th axis block end position (ABSKP)			
#5021	X tool length compensation value		unallowed	
#5022	Y tool length compensation value			
#5023	Z tool length compensation value			
#5024	The 4th axis block end position			
#5026	X servo position compensation			
#5027	Y servo position compensation			
#5028	Z servo position compensation			
#5029	The 4th axis servo position compensation			

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

3）Workpiece zero offset value and additional zero offset value：

Table 4-7-2-4

Variable number	Function
#5201 … #5204	External workpiece zero offset value of 1st axis … External workpiece zero offset value of 4th axis
#5206 … #5209	G54 workpiece zero offset value of 1st axis … G54 workpiece zero offset value of 4th axis
#5211 … #5214	G55 workpiece zero offset value of 1st axis … G55 workpiece zero offset value of 4th axis
#5216 … #5219	G56 workpiece zero offset value of 1st axis … G56 workpiece zero offset value of 4th axis
#5221 … #5224	G57 workpiece zero offset value of 1st axis … G57 workpiece zero offset value of 4th axis
#5226	G58 workpiece zero offset value of 1st axis

… #5229	… G58 workpiece zero offset value of 4th axis
#5231 … #5234	G59 workpiece zero offset value of 1st axis … G59 workpiece zero offset value of 4th axis
#7001 … #7004	G54 P1 workpiece zero offset value of 1st axis … G54 P1 workpiece zero offset value of 4th axis
#7006 … #7009	G54 P2 workpiece zero offset value of 1st axis … G54 P2 workpiece zero offset value of 4th axis
#7246 … #7249	G54 P50 workpiece zero offset value of 1st axis … G54 P50 workpiece zero offset value of 4th axis

4．Local variables

The correspondence relationship between address and local variable:：

Table 4-7-2-5

Argument address	Local variable No.	Argument address	Local variable No.
A	#1	Q	#17
B	#2	R	#18
C	#3	S	#19
I	#4	T	#20
J	#5	U	#21
K	#6	V	#22
D	#7	W	#23
E	#8	X	#24
F	#9	Y	#25
M	#13	Z	#26

Note 1: The assignment is done by an English letter followed by a numerical value. Except letters G, L, O, N, H and P, all the other 20 letters can assign values for arguments. Each letter from A-B-C-D… to X－Y－Z can assign a value once and the assignment needs not to be performed in alphabetical order. The addresses that assign no values can be omitted.

Note 2: G65 must be specified before any argument is used.

5．Notes for custom macro body

1) Input by keys

Press key # behind the parameter words G, X, Y, Z, R, I, J, K, F, H, M, S, T, P, Q for inputting "#".

2) Either operation or transfer code can be specified in MDI mode.

3) H, P, Q, R of the operation and transfer codes preceding or behind G65 are all used as parameters for G65.

H02 G65 P#100 Q#101 R#102 ; Correct

N100 G65 H01 P#100 Q10 ; Correct

4) Variable range: the input figure is no more than 7-digit and the result is no more than 8-digit no matter negative or positive.

5) The result of the variable operation can be a decimal fraction with a precision of 0.0001. All operations, except H11 (OR operation), H12 (AND operation), H13 (NOT operation), H23 (ROUNDING operation) with decimal portions neglected in operation, are done without the decimal portions abnegated.

Example:

#100 ＝ 35， #101 ＝ 10， #102 ＝ 5

#110 ＝ #100÷#101 (＝3.5)

#111 ＝ #110×#102 (＝17.5)

#120 ＝ #100×#102 (＝175)

#121 ＝ #120÷#101 (＝17.5)

6) The execution time of operation and transfer code differs depending on different conditions. The average time is usually 10ms.

7) When the variable value is not defined, the variable becomes “null”. When the variable #0 is always null, it is only read instead of being written.

a. Quotation

When an undefined variable is referred, the address itself is ignored.

For example:

When the variable #1 value is 0，and the variable #2 value is null, G00X#1 Y#2 execution result is G00X0；

b. Operation

Except for using <Null> assigns, <Null> in other conditions is the same that of 0.

Table 4-7-2-6

#1=<vacant>	#1=0
#2=#1 ↓ #2=<空>	#2=#1 ↓ #2=0
#2=#1*5 ↓ #2=0	#2=#1*5 ↓ #2=0
#2=#1+#1 ↓ #2=0	#2=#1+#1 ↓ #2=0

c. Conditional expression

<Null > in EQ and NE are different from 0.

Table 4-7-2-7

#1=< vacant >	#1=0
#1 EQ #0 ↓ Established	#1 EQ #0 ↓ Not established
#1 NE #0 ↓ Not established	#1 NE #0 ↓ Established
#1 GE #0 ↓ Established	#1 GE #0 ↓ Established
#1 GT #0 ↓ Not established	#1 GT #0 ↓ Not established

Ⅰ Programming

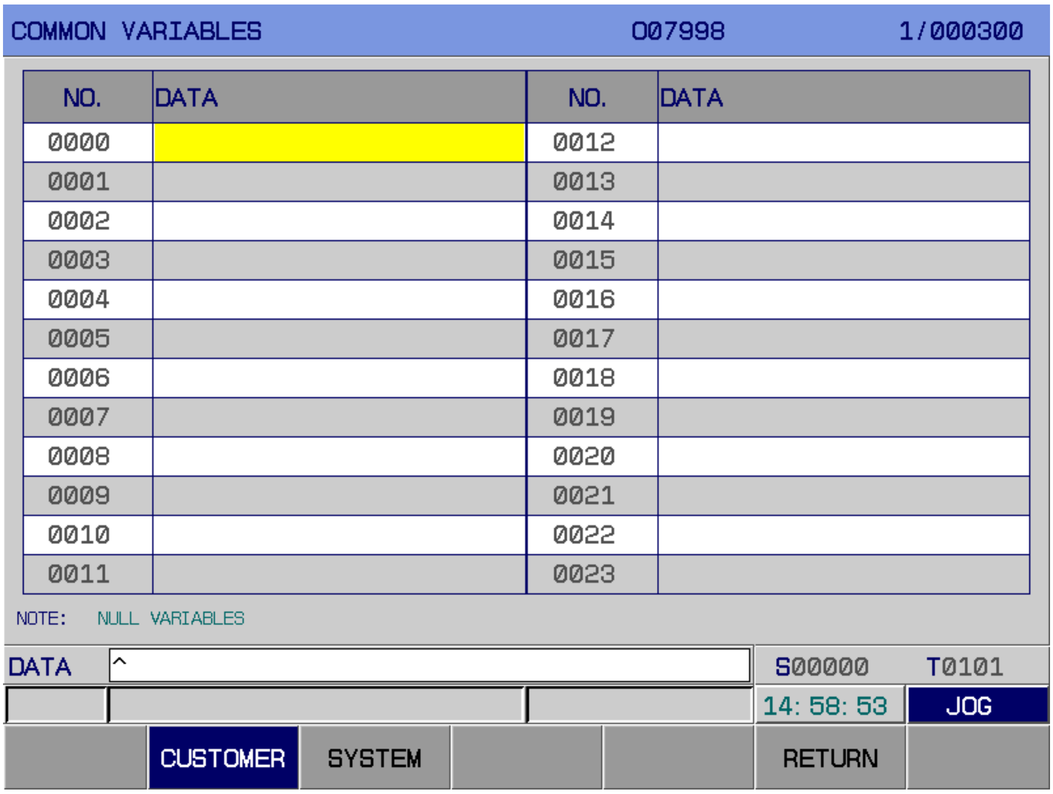


Fig.　4-7-2-2

When the variable value is null, the variable is null.

4.7.3　Non-Modal Call G65

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

Format:

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

Calling times

Program number of called user macro body

Behind G65 code, P is used to specify custom macro number, L is used to specify custom macro calling times, and the arguments are used to transfer data to custom macro.

If repetition is needed, specify the number of repeats behind L code from 1-9999; if L is omitted, the default time is 1.

If it is specified by arguments, the values will be assigned to the corresponding local variables.

Note 1: If the subprogram number specified by address P is not retrieved, an alarm (PS 078) is issued.
Note 2: No. 90000~99999 subprograms are the system reserved programs, if such subprograms are called, they can be executed, but the cursor will keep staying at block N65 and the program page displays the main program all the time. (The subprogram can be displayed by setting bit parameter No: 27#4)
Note 3: Up to five macro calls are nested.

4.7.4　User Macro Program Function A

1．General format:

G65 Hm P#i Q#j R#k ；

m： 01～99 indicate functions of operation code or transfer code

#i： Variable name for saving the operation result.

#j: Variable name 1 for operation, or a constant which is expressed directly without #.

#k: Variable name 2 for operation，or a constant.

Meaning： #i = #j ○ #k

└──────Operation sign, specified by Hm

(Example) P#100 Q#101 R#102……#100 = #101 ○ #102 ；

P#100 Q#101 R15 ……#100 = #101 ○ 15 ；

P#100 Q-100 R#102……#100 = -100 ○ #102

H code specified by G65 has no effect on the offset selection.

G code	H code	Function	Definition
G65	H01	Value assignment	#i ＝ #j
G65	H02	Addition	#i ＝ #j + #k
G65	H03	Subtraction	#i ＝ #j - #k
G65	H04	Multiplication	#i ＝ #j × #k
G65	H05	Division	#i ＝ #j ÷ #k
G65	H11	Logic addition (OR)	#i ＝ #j OR #k
G65	H12	Logic multiplication (AND)	#i ＝ #j AND #k
G65	H13	Exclusive OR	#i ＝ #j XOR #k
G65	H21	Square root	#i = $\sqrt{\#j}$
G65	H22	Absolute value	#i ＝| # j |
G65	H23	Complement	#i＝#j-trunc(#j ÷ #k)×#k
G65	H26	Compound multiplication and division operation	#i ＝ (#i × #j) ÷ #k
G65	H27	Compound square root	#i = $\sqrt{\#j^2+\#k^2}$
G65	H31	Sine	#i ＝ #j×SIN(#k)
G65	H32	Cosine	#i ＝ #j×COS(#k)
G65	H33	Tangent	#i ＝ #j×TAN(#k)
G65	H34	Arc tangent	#i ＝ ATAN(#j/#k)
G65	H80	Unconditional transfer	GOTO N
G65	H81	Conditional transfer 1	IF #j ＝ #k， GOTO N
G65	H82	Conditional transfer 2	IF #j ＝ #k， GOTO N
G65	H83	Conditional transfer 3	IF #j ＞ #k， GOTO N
G65	H84	Conditional transfer 4	IF #j ＜ #k， GOTO N
G65	H85	Conditional transfer 5	IF #j ＞ #k， GOTO N
G65	H86	Conditional transfer 6	IF #j ＜ #k， GOTO N
G65	H89	Alarm	

Table 4-7-4-1

2．Operation code：

1）Variable assignment：# I = # J

G65 H01 P#I Q#J；

（Example）G65 H01 P#101 Q1005； （#101 = 1005）
G65 H01 P#101 Q#110； （#101 = #110）
G65 H01 P#101 Q-#102； （#101 = -#102）

2）Additive operation：# I = # J+# K

G65 H02 P#I Q#J R#K；

（Example）G65 H02 P#101 Q#102 R15； （#101 = #102+15）

3）Subtraction operation：# I = # J- #K

G65 H03 P#I Q#J R# K；

（Example）G65 H03 P#101 Q#102 R#103； （#101 = #102-#103）

4）Multiplication operation：# I = # J×# K

G65 H04 P#I Q#J R#K；

（Example）G65 H04 P#101 Q#102 R#103； （#101 = #102×#103）

5）Division operation：# I = # J÷# K

G65 H05 P#I Q#J R#K；

（Example）G65 H05 P#101 Q#102 R#103； （#101 = #102÷#103）

6）Logic add（or）：# I = # J.OR. # K

G65 H11 P#I Q#J R#K；

（Example）G65 H11 P#101 Q#102 R#103； （#101 = #102.OR. #103）

7）Logic multiply（and）：# I = # J.AND. # K

G65 H12 P#I Q#J R#K；

（Example）G65 H12 P# 101 Q#102 R#103； （#101 = #102.AND.#103）

8）XOR(Exclusive OR)：# I = # J.XOR. # K

G65 H13 P#I Q#J R#K；

（Example）G65 H13 P#101 Q#102 R#103； （#101 = #102.XOR. #103）

9）Square root: # I = $\sqrt{\#J}$

G65 H21 P#I Q#J ；

（Example）G65 H21 P#101 Q#102 ； （#101 = $\sqrt{\#202}$ ）

10）Absolute value：# I = | # J |

G65 H22 P#I Q#J ；

（Example）G65 H22 P#101 Q#102 ； （#101 = | #102 |）

11）Remainder：# I = # J－TRUNC(#J/#K)×# K，TRUNC：rounding the decimal part

G65 H23 P#I Q#J R#K；

（Example）G65 H23 P#101 Q#102 R#103； （#101 = #102- TRUNC (#102/#103)×#103）

12）Composite multiply-divide operation：# I =（# I×# J）÷# K

G65 H26 P#I Q#J R# k；

（Example）G65 H26 P#101 Q#102 R#103； （#101 =（#101×# 102）÷#103）

13）Compound square root: # I = $\sqrt{\#J^2+\#K^2}$

Ⅰ Programming

G65 H27 P#I Q#J R# k；

（Example）G65 H27 P#101 Q#102 R#103； （#101 $\sqrt{\#102^2+\#103^2}$ ）

14）Sine：# I = # J•SIN（# K）（Unit：°）

G65 H31 P#I Q#J R#K；

（Example）G65 H31 P#101 Q#102 R#103； （#101 = #102•SIN（#103））

15）Cosine：# I = # J•COS（# K）（Unit：°）

G65 H32 P#I Q#J R# K；

（Example）G65 H32 P#101 Q#102 R#103； （#101 =#102•COS（#103））

16）Tangent：# I = # J•TAN（# K）（Unit：°）

G65 H33 P#I Q#J R# K；

（Example）G65 H33 P#101 Q#102 R#103； （#101 = #102•TAN（#103））

17）Arc tangent：# I = ATAN（# J /# K）（Unit：°）

G65 H34 P#I Q#J R# K；

（Example）G65 H34 P#101 Q#102 R#103； （#101 =ATAN（#102/#103））

Note 1: The unit of angular variable is degree.
Note 2: If the required Q and R are not specified in operations above, their values are 0 by default.
Note 3: trunc: rounding operation, the decimal portion is abandoned.

3．Jump code

1）Unconditional jump

G65 H80 Pn； n：Sequence number

（Example）G65 H80 P120;（jump to N120）

2）Conditional jump1 #J.EQ.# K (＝)

G65 H81 Pn Q#J R# K； n：Sequence number

(Example) G65 H81 P1000 Q#101 R#102；

101 = #102: jump to N1000，#101 ≠ #102: programs are executed orderly.

3）Conditional jump 2 #J.NE.# K (≠)

G65 H82 Pn Q#J R# K； n：Sequence number

(Example) G65 H82 P1000 Q#101 R#102；

101 ≠ #102: jump to N1000，#101 = #102: programs are executed orderly.

4）Conditional jump 3 #J.GT.# K (＞)

G65 H83 Pn Q#J R# K； n：Sequence number

(Example) G65 H83 P1000 Q#101 R#102；

#101 ＞ #102: jump to N1000，#101 ≤ #102: programs are executed orderly.

5）Conditional jump 4 #J.LT.# K (＜)

G65 H84 Pn Q#J R# K； n：Sequence number

(Example) G65 H84 P1000 Q#101 R#102；

101＜#102: jump to N1000，#101 ≥ #102: programs are executed orderly.

6）Conditional jump 5 #J.GE.# K (≥)

<u>G65 H85 Pn Q#J R# K；　n：Sequence number</u>

(Example) G65 H85 P1000 Q#101 R#102；

101 ≥ #102: jump to N1000，#101＜#102: programs are executed orderly.

7）Conditional jump 6 #J.LE. # K (≤)

<u>G65 H86 Pn Q#J R# K；　n：Sequence number</u>

(Example) G65 H86 P1000 Q#101 R#102；

101≤ #102: jump to N1000，#101＞#102: programs are executed orderly.

Note: The sequence number can be specified by variables. Such as G65 H81 P#100 Q#101 R#102;if the conditions are satisfied, it goes to the block of which the number is specified by #100.

4．Logic AND, logic OR and logic NOT codes

Example：

G65 H01 P#101 Q3；

G65 H01 P#102 Q5；

G65 H11 P#100 Q#101 Q#102；

The binary expression for 5 is 101, for 3 is 011, and the operation result is #100=7;

G65 H12 P#100 Q#101 Q#102；

The binary expression for 5 is 101, for 3 is 011, and the operation result is #100=1.

5．Macro variable alarm

Example：

G65 H99 P1；　　Macro variable 3001 alarm

G65 H99 P124；　Macro variable 3124 alarm

4.7.5　User Macro Program Function B

1. Arithmetic and logic operation

The operations listed in the following table can be executed on variables. The expressions on the right of the operation characters can contain constants and/or variables constituted by functions or operation characters. The variables #j and #k in the expression can be replaced by constants. The values of the variables on the left can also be assigned by an expression.

Table 4-7-5-1　Arithmetic and logic operation

Function	Format	Remarks
Definition	#i = #j	
Addition Subtraction Multiplication Division	#i = #j + #k; #i = #j - #k; #i = #j * #k; #i = #j / #k;	
Sine Arcsine Cosine Arc cosine Tangent	#i = SIN[#j]; #i = ASIN[#j]; #i = COS[#j]; #i = ACOS[#j]; #i = TAN[#j];	The angle is specified by degree. 90°30’ indicates an angle of 90.5°.

Arc tangent	#i = ATAN[#j] / [#k];	
Square root Absolute value Rounding-off Rounding up to an integer Rounding down to an integer Natural logarithm Exponential function	#i = SQRT[#j]; #i = ABS[#j]; #i = ROUND[#j]; #i = FUP[#j]; #i = FIX [#j j]; #i = LN[#j]; #i = EXP[#j];	
OR Exclusive OR AND	#i = #j OR #k; #i = #j XOR #k; #i = #j AND #k;	Logic operation is executed by the binary system.
BCD to BIN Bin to BCD	#i = BIN[#j]; #i = BCD[#j];	Used for switching with PMC signal

Explanation:

(1) **Angle unit**

The angle unit of functions SIN，COS，ASIN，ACOS，TAN and ATAN is degree, e.g., 90°30′ indicates an angle of 90.5°.

(2) **ARCSIN #i = ASIN [#j]**

Ranging from -90° to 90°.

When #j is beyond the range from -1 to 1, an alarm occurs.

（3）**ARCCOS #i = ACOS [#j]**

Ranging from 180° to 0°.

When #j is beyond the range from -1 to 1, an alarm occurs.

Variable #j can be replaced by constants.

（4）**ARCTAN #i = ATAN [#j] / [#k]**

Specify the lengths of two sides, separated by a slash（/）.

Ranging from 0° to 360°.

[Example] When #1 = ATAN [-1] / [-1]; is executed, #1=225°.

Variable #j can be replaced by constants.

（5）**Natural logarithm #i = LN [#j]**

When antilog (# j) is 0 or smaller, an alarm occurs.

Variable #j can be replaced by constants.

（6）**Exponential function #i = EXP [#j]**

When the operation result exceeds 99997.453535 (j is about 11.5129), an overflow occurs and an alarm is issued.

(7) **ROUND（rounding-off）function**

The round function rounds off at the first decimal place.

Example:

When #1=ROUND[#2]; is executed where #2 holds 1.2345, the value of variable #1 is 1.0.

（8）**Rounding up and down to an integer**

When the value operation is processed by CNC, if the absolute value of the integer produced by an operation on a number is greater than the absolute value of the original number, such an operation is referred to as rounding up to an integer. If the absolute value of the integer produced by an operation on a number is smaller than the absolute value of the original number, such an operation is referred to as rounding down to an integer. Please be careful when handling negative numbers.

Example:

Suppose that #1=1.2，#2=－1.2.

When #3=FUP[#1] is executed, 2.0 is assigned to #3.

When #3=FIX[#1] is executed, 1.0 is assigned to #3.

When #3=FUP[#2] is executed, -2.0 is assigned to #3.

When #3=FIX[#2] is executed, -1.0 is assigned to #3.

（9）**Abbreviations of the arithmetic and logic codes.**

When a function is specified in a program, the first two characters of the function name can be used to specify the function. (See Table 4-10-5-1）

Example：

ROUND→RO

FIX→FI

（10）**Operation sequence**

① Function

② Multiplication and division operation（* / AND）

③ Addition and subtraction operation（+ - OR XOR）

Example) $\#1 = \underbrace{\#2 + \underbrace{\#3 * \underbrace{SIN[\#4]}_{\boxed{1}}}_{\boxed{2}}}_{\boxed{3}}$;

$\boxed{1}$, $\boxed{2}$ and $\boxed{3}$ indicate the operation sequence.

（11）**Restrictions**

Brackets [,] are used to enclose an expression.

When a divisor of 0 is specified in a division or TAN[90], an alarm is given.

2. Jump and loop

1）Jump and loop

In the program, GOTO statement and IF statement are used to change the control flow. There are three types of jump and loop operations:

Jump and cycle
- GOTO statement（unconditional jump）
- IF statement（conditional jump：IF…THEN…）
- WHILE statement（When… loop is performed）

2）Unconditional jump

➢ GOTO statement

Jump to the block with sequence number n. The sequence number can be specified by an expression.

GOTOn; n: Sequence number（1 to 99999）

Example：

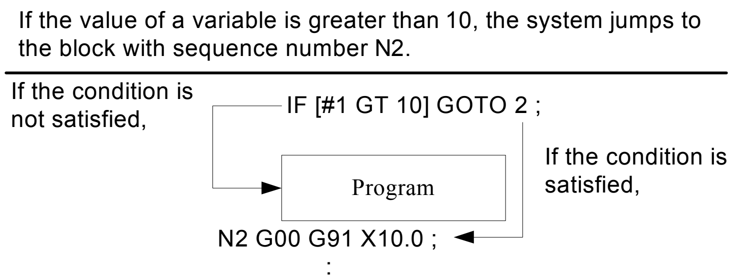
GOTO 1；

GOTO #10；

3）Conditional jump（IF statement) [<conditional expression >]

IF[<conditional expression **>]GOTO n**

If the specified conditional expression is satisfied, the system transfers to the block with sequence number n; if the specified conditional expression is not satisfied, the next block is executed.



IF[<conditional expression>]THEN

If the conditional expression is satisfied, a predetermined macro statement is executed. Only a single macro statement is executed.

If the values of #1 and #2 are the same，0 is assigned to #3.
IF[#1 EQ #2] THEN #3=0;

Explanation:

➢ Conditional expression

A conditional expression must include an operator, which is inserted between two variables or between a variable and a constant, and must be enclosed with brackets （[，]）. An expression can replace a variable.

➢ Operator

Operators each consists of two letters are used to compare two values to determine whether they are equal or one is greater or smaller than the other one.

Table 4-7-5-2 Operators

Operator	Meaning
EQ	Equal to（=）
NE	Not equal to ≠）
GT	Greater than（>）
GE	Greater than or equal to（≥）
LT	Smaller than（<）

LE	Smaller than or equal to（≤)

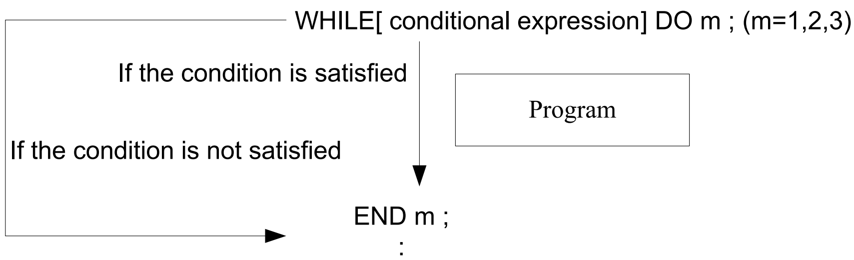
➢ Typical program

The program below calculates the sum of numerical value 1 to 10.

```
O9500;
#1=0;                        Initial value of the variable to hold the sum
#2=1;                        Initial value of the variable as an addend
N1 IF[#1 GE 10]GOTO 2;       Jump to N2 when the addend is greater than or equal to 10
#1=#1+#2;                    Calculation to find the sum
#1=#2+1;                     The next addend
GOTO 1;                      Traverse to N1
N2 M30;                      Program end
```

4）Loop (WHILE statement)

Specify a conditional expression behind WHILE, when the specified condition is satisfied, the program from DO to END is executed, otherwise, program execution proceeds to the block after END.



当 When the specified condition is satisfied, the program from DO to END is executed. Otherwise, program execution proceeds to the block after END. This kind of command format is applicable to IF statement. A number after DO and a number after END are the identification numbers for specifying the range of execution. The identification numbers are 1, 2 and 3. If numbers other than 1, 2 and 3 are used, an alarm occurs.

Explanation:

➢ Nestling

The identification numbers (1 to 3) in the loop from D0 to END can be used repeatedly as required. However, when a program includes crossing repetition loop (overlapped DO ranges), an alarm occurs.

1.The identification numbers (1 to 3) can be used as many times as required.

```
WHILE[ … ] DO 1;
    Program
END 1 ;
  :
WHILE[ … ] DO 1;
    Processing
END 1 ;
  :
```

2. The ranges of DO cannot overlap

```
WHILE[ … ] DO 1;
    Program
WHILE[ … ] DO 2;
  :
END 1 ;
    Program
END 2 ;
```

3. DO loops can be nested to 3 levels

```
WHILE[ … ] DO 1;
  :
  WHILE[ … ] DO 2;
    :
    WHILE[ … ] DO 3;
        Program
    END 3 ;
  END 2 ;
END 1 ;
```

4. The control can be jumped to the outside of a loop.

```
WHILE[ … ] DO 1;
IF[…] GOTO n ;
END 1 ;
Nn
```

5. Jump cannot enter the loop area.

```
IF […] GOTO n ;
  :
WHILE […] DO 1 ;
Nn… ;
END 1 ;
```

Explanation:

- Infinite loop

  When DO is specified without specifying WHILE statement, an infinite loop from DO to END is produced.

- Processing time

  When a transfer to a sequence number in GOTO statement occurs, the sequence number is searched for. Processing in the reverse direction is longer than the one in the forward direction. The processing time can be reduced by using WHILE statement for repetition.

- Undefined variables

In the conditional expression using EQ or NE, <vacant> and zero have different affects. In the other conditional expressions, <vacant> is taken as 0.

- Typical program

The program below calculates the sum of numbers 1 to 10.

```
O0001 ;
#1=0;
#2=1;
WHILE [#2 LE 10] DO 1;
#1=#1+#2;
#2=#2+1;
END 1;
M30;
```

Notes：

- When a macro program is called by G65, and F is used for transferring variables. The system performs call according to the variable value.
- A GOTO statement starts searching at the beginning of the program and searches from the start again when the corresponding line number is not retrieved. Try not to use the same N code in one program.
- When the variable number is expressed by a decimal fraction, the system will remove the decimal part with carry ignored.
- The values of local variables are retained before the main program ends. They are common to each subprogram.

Chapter 5 Miscellaneous Function M Code

The M codes of this machine available for users are listed as follows:

Table 5-1

	M **code**	**Function**
M codes used for control program	M30	The program ends and returns to the program beginning, the machining number increases by 1.
	M02	The program ends and returns to the program beginning, the machining number increases by 1.
	M98	Subprogram calling
	M99	Subprogram ends and returns/execution is repeated
	M00	Program dwell
	M01	Program optional dwell
M codes controlled by PLC	M03	Spindle CCW
	M04	Spindle CW
	M05	Spindle stop
	M08	Tool change
	M09	Cooling ON
	M10	Cooling OFF
	M11	Tailstock retreat
	M12	Chuck clamp
	M13	Chuck release
	M14	Spindle switched from speed control mode into position control mode
	M15	Spindle switched from position control mode into speed control mode
	M18	Spindle orientation cancel
	M19	Spindle orientation
	M20	C axis release
	M21	C axis clamp
	M26	M26 output（custom）
	M27	Close M26 output
	M28	Cancel rigid tapping
	M29	Code rigid tapping
	M35	M35 output（custom）
	M36	Close M35 output
	M41	Spindle 1st gear
	M42	Spindle 2nd gear
	M43	Spindle 3rd gear
	M44	M44 output（custom）
	M45	Close M44 output

When a movement code and miscellaneous function are specified in the same block, they are simultaneously executed.

When a numerical value is specified behind address M, code signal and strobe signal are sent to the machine. The machine uses these signals to turn on/off these functions. Usually only one M code can be specified in a block. In some cases, up to three M codes can be specified in a block by setting bit parameter No.33#7. Some M codes cannot be specified simultaneously because of the restrictions of the mechanical operation. See the machine manual provided by the tool builder for the mechanical operation restrictions on simultaneous specification for M codes in one block.

5.1 M codes Controlled by PLC

When an M code controlled by PLC is in the same block with a movement code, they are simultaneously executed.

5.1.1 Spindle Rotation CW/CCW（M03, M04）

Code ：M03（M04） Sx x x；

Explanation：

M03：Spindle rotation CW，

M04：Spindle rotation CCW

Sx x x specifies the spindle speed, or the current gear in gear control mode.
Unit: revolution per minute (r/min)
When it is controlled by a frequency converter, Sx x x specifies the actual speed. e.g. S1000 specifies the spindle to rotate at a speed of 1000r/min.

5.1.2 Spindle Stop（M05）

Code: M05. When M05 is executed in auto mode, the spindle is stopped, but the speed specified by S code is retained. The deceleration at spindle stop is set by the machine builder. It is usually done by energy consumption brake.

5.1.3 Cooling ON/OFF（M08, M09）

Code：M08，control the cooling ON. M09，control the cooling OFF. When the miscellaneous function is locked in auto mode, the code is not executed.

5.1.4 Chuck Control（M12, M13）

Code：M12，chuck release. M13，chuck clamp

5.1.5 Spindle Speed/Position Mode Switch (M14, M15)

Code：M14，the spindle is switched from speed control mode into position control mode.
M15，the spindle is switched from position control mode into speed control mode.

5.1.6 Spindle Orientation, Spindle Orientation Cancel（M18，M19）

Code: M18，cancel the spindle orientation. M19，the spindle performs orientation, which is used to the tool change orientation.

5.1.7 C-Axis Releasing, Clamping（M20, M21）

Code: M20，C releases. M21，C clamps.

5.1.8 Spindle Gear Control（M41, M42, M43）

Code: M41，the spindle's 1st gear. M42，the spindle's 2nd gear. M43，the spindle 3rd.

5.2 M Codes Used by Control Program

M codes used by a program are divided into a main program type and macro type. When an M code used by a program and a movement code are in the same block, the movement code is executed prior to the M code.

Note

1: M00, M01, M02, M06, M30, M98 and M99 cannot be specified together with other M codes, or an alarm is issued. When these codes are in the same block with other non-M codes, the non-M codes are executed prior to the M codes.

2: This kind of M codes include the codes that direct the CNC to perform the internal operation in addition to sending the M codes themselves to the machine, e.g. the M code to disable the block prereading function. Moreover, the codes to send the M codes themselves to the machine (without performing the internal operation) can be specified in the same block.

5.2.1 Program End and Return（M30, M02）

When M30（M02）in the program is executed in auto mode, the auto mode is cancelled. The blocks following them are not executed and the spindle and cooling are stopped. Meanwhile, the workpiece machined number increases by 1. Whether the control returns to the beginning of the program after M30 is executed is set by bit parameter N0:33#4; whether the control returns to the beginning of the program after M02 is executed is set by bit parameter N0:33#2. If M02 and M03 are in a subprogram, then the control returns to the program calling the subprogram after they are executed and proceeds to the following blocks.

5.2.2 Program Dwell（M00）

In Auto running, the automatic operation pauses after a block containing M00 is executed. Meanwhile, the previous modal information will be saved. The automatic operation is continued by pressing Cycle Start key, which is equivalent to pressing down key Feed Hold.

5.2.3 Program Optional Stop（M01）

Automatic operation is stopped optionally after a block containing M01 is executed. If the “Optional Stop” switch is set to ON, M01 is equivalent to M00; if the “Optional Stop” switch is set to OFF, M01 is ineffective. See *OPERATION*.

5.2.4 Subprogram Calling（M98）

M98 is used to call a subprogram in a main program. Its format is as follows:

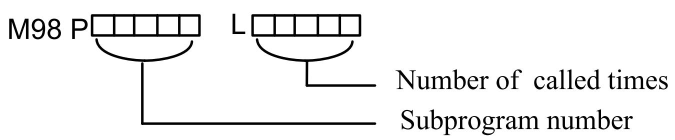


Fig. 5-2-4-1

5.2.5 Program End and Return（M99）

Command format:

M99 P○○○○

Block number（0001～9999）to be executed after returning to the main program , the leading zero can be omitted.

Function: After other codes in the current block (in the subprogram) are executed, the system returns to the main program, and executes the block specified by P. When P is not input, the system returns to the main program to call the next block following the current subprogram's M98. When M99 is used to end the main program, the current program is repetitively executed.

1. In auto mode, if M99 is executed at the end of the main program, the control returns to the program beginning to continue automatic operation. Meanwhile, the following blocks are not to be executed, and the number of the machined workpieces is not accumulated.
2. If M99 is executed at the end of a subprogram, the control returns to the main program and proceeds to the next block following the subprogram block.

Chapter 6 Spindle Function S Code

By using an S code and the numerical values behind it, the code signal can be converted to the analog signal and then sent to the machine, for controlling the machine spindle. S is a modal value.

6.1 Spindle Analog Control

When the bit parameter NO.1#2 SPT=0, the spindle speed is controlled by the analog voltage which is specified by address S and the numerical values behind. See *OPERATION* in the manual for details.

Command format：S_

Explanation:

1. Only one S code can be specified in a block.
2. The spindle speed is specified directly by address S and a numerical value behind it. Unit: r/min. e.g. For M3 S300, it means the spindle is rotated at a speed of 300 r/min.
3. If a move code and an S code are specified in the same block, they are executed simultaneously.
4. The spindle speed is controlled by an S code followed by a numerical value.

6.2 Spindle Switch Value Control

When the bit parameter NO.1#2 SPT=1, the spindle speed is controlled by the switch value, which consists of an address S and a two-digit number behind it.
Three mechanical gears for the spindle are provided when the spindle speed is controlled by the switch value. For the correspondence between S codes and spindle speed as well as the number of spindle gears, please see the manual provided by the machine tool builder.

Command format：S01（S1）；

S02（S2）；

S03（S3）；

Explanation:

There are 8 gears in the software at present, and 3 gears in the ladder diagram. When S codes beyond the codes above are specified, the system displays “Miscellaneous function being executed”.

6.3 Constant Surface Speed Control G96/G97

Command format：

Constant surface speed control: G96 S_ Surface speed（m/min or inch/min）

Constant surface speed control cancel: G97 S_ Spindle speed（r/min）

Constant surface speed controlled axis G96 P_ P1 X；P2 Y；P3 Z；P4 4^{th} axis

Function: The number following S is used to specify the surface speed (relative speed between tool and workpiece). The spindle is rotated so that the surface speed is constant regardless of the tool position.

Explanation:

1. G96 is modal. After it is specified, the program enters the constant surface speed control mode and the specified S value is assumed as a surface speed.
2. G96 code must specify the axis along which constant surface speed control is applied. It can be cancelled by G97 code.
3. To execute the constant surface speed control, it is necessary to set a workpiece coordinate system, then the coordinate value at the center of the rotary axis becomes zero.

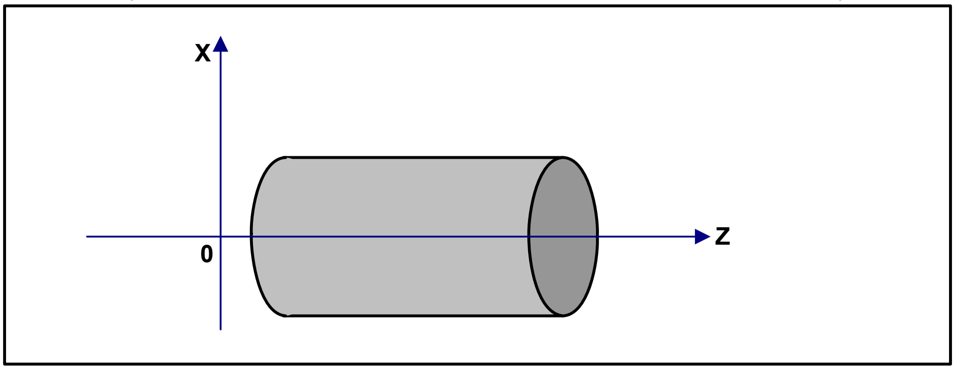


Fig. 6-3-1 workpiece coordinate system controlled by constant surface cutting speed

4. When constant surface speed control is applied, if a spindle speed higher than the value specified in G 92 S_, it is clamped at the maximum spindle speed. When the power is switched on, and the maximum spindle speed is not yet set, in G96, S=0 till M3 or M4 appears in the program.

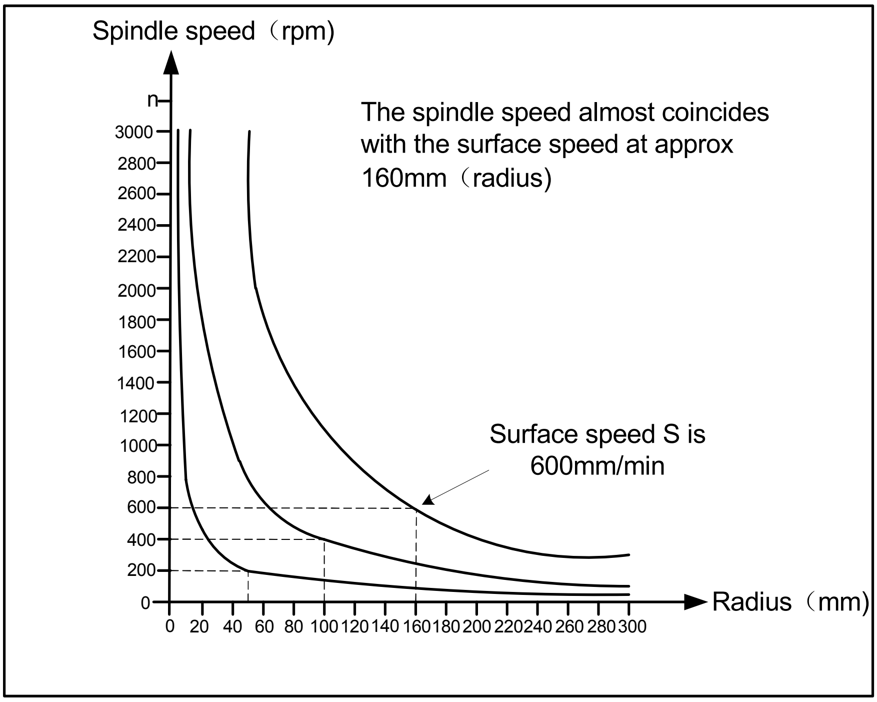


Fig. 6-3-2 Relation between workpiece radius, spindle speed and surface speed

5. Surface speed specified in G96 mode:

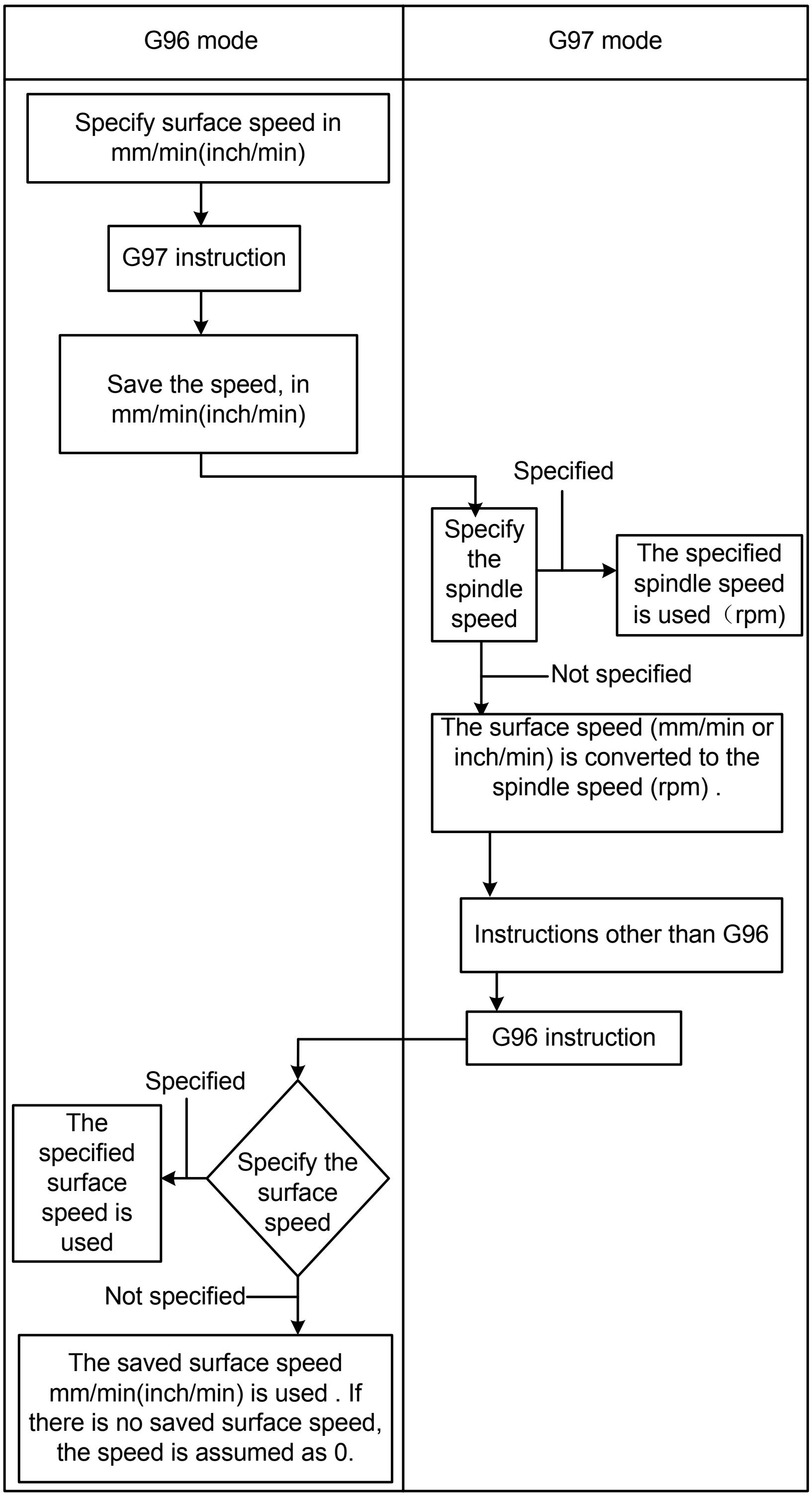


Fig. 6-3-3

6. G96 relevant parameter setting: bit parameter No.37#2 sets a reference coordinates (0: end point, 1: current point) to count G96 spindle speed when G0 rapidly positions, bit parameter No.37#3 sets G96 spindle speed clamped（0：before the spindle override，1：after the spindle override），bit parameter No.61#0 sets whether to use the constant surface speed control.

Restrictions:

1. Because the response problem in the servo system may not be considered when the spindle speed changes, and the constant surface speed is also effective during threading, it is recommended to cancel the constant surface speed by G97 before threading.
2. In a rapid traverse block specified by G00, the constant surface speed control is not made by calculating the surface speed by a transient change of the tool position, but is made by calculating the surface speed based on the position at the end point of the rapid traverse block, on the condition that cutting is not performed during rapid traverse. Therefore, the constant surface cutting speed is not used.
3. In flexible tapping, rigid tapping or deep hole rigid tapping, use G97 to cancel the constant surface cutting speed, otherwise, random teeth or broken screw tap exists.

Chapter 7 Feed Function F Code

The feed functions are used to control the feedrate of the tool. The functions and control modes are as follows:

7.1 Rapid Traverse

G00 code is used for rapid positioning. The traverse speed is set by data parameters P88~P92. An override can be applied to the traverse speed by the OVERRIDE adjusting keys on the operator panel, which are shown as follows:

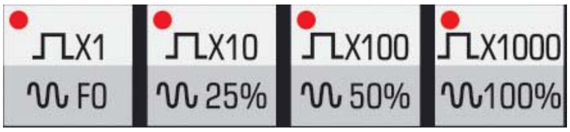


Fig. 7-1-1 Keys for rapid traverse override

F0 is set by data parameter P93.

The acceleration of rapid positioning (G0) can be set by data parameters P105~123. It can be properly set depending on the machine and the motor response characteristics.

Note: G00 block, the feedrate code F is invalid even if it is specified. The system performs positioning at the speed specified by G0 instead.

7.2 Cutting Feedrate

The tool feedrates in linear interpolation (G01) and circular interpolation(G02，G03) are specified with the numbers after F code in mm/min. The tool is moved by the programmed feedrate. An override can be applied to the cutting feedrate using the override keys on the operator panel (Override range: 0％～200%).

In order to prevent mechanical vibration, acceleration/deceleration is automatically applied at the beginning and the end of the tool movement respectively. The acceleration can be set by data parameters P125~P128.

The minimum cutting feedrate is set by data parameter P96, and the maximum cutting feedrate in the forecast mode is set by P97. If it is smaller than the lower limit, the cutting feedrate is clamped to the lower limit.

The cutting feedrate in auto mode at power-on is set by data parameter P87.

The cutting feedrate can be specified by the following two types:

A) Feed per minute (G94): it is used to specify the feed amount per minute after F code.

B) Feed per revolution (G95): it is used to specify the feed amount per revolution after F code.

7.2.1 Feed per Minute（G98）

Command format: G98 F_

Function: It specifies the tool feed amount per minute. Unit: mm/min or inch/min.

Explanation:

1. After G98 is specified (in feed per minute mode), the feed amount of the tool per minute is

directly specified by a number after F.

2. G98 is a modal code. Once specified, it remains effective till G99 is specified. The default at power-on is feed per minute mode.
3. An override from 0% to 200% can be applied to feed per minute with the override keys or band switch on the operator panel.

7.2.2 Feed per Revolution（G99）

Command format: G99　F_

Function: Feed amount per revolution. Unit: mm/r or inch/r

Explanation:

1. This function is unavailable until a spindle encoder is installed on the machine.
2. After specifying G99 (feed per revolution mode), the feed amount of the tool per revolution is directly specified by a number after F.
3. G99 is a modal code. Once specified, it keeps effective till G98 is specified. The default feedrate per revolution during initialization is 0.
4. An override from 0% to 200% can be applied to feed per revolution with the override keys or band switch on the operator panel.

Note 1：When the spindle speed is low, feedrate fluctuation may occur. The lower the spindle speed is, the more frequently the feedrate fluctuation occurs.

Note 2：For G99 feed per rev, the system executes its max. speed F500，an alarm occurs when it exceeds the max. speed.

7.3 Tangential Speed Control

The cutting feed usually controls the speed in the tangential direction of the contour path to make it reach the specified speed value.

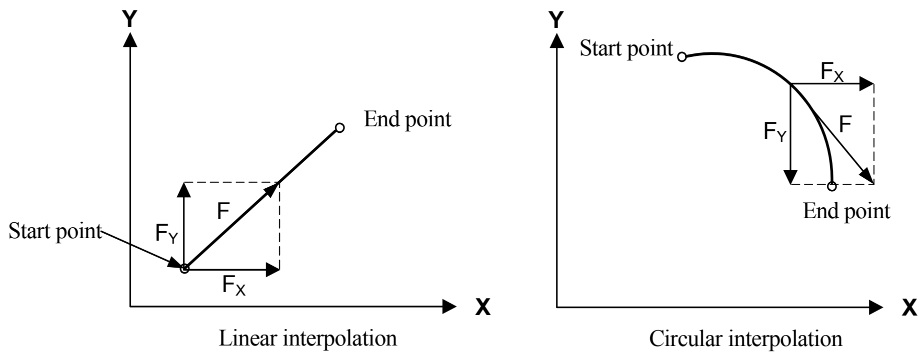


Fig. 7-3-1

$$F=\sqrt{Fx^2+Fz^2}$$

F： tangent direction's speed

Fx： X-axis speed

Fz： Z-axis speed

7.4 Keys for Feedrate Override

The feedrate in MANUAL mode and AUTO mode can be overridden by the override keys on the operator panel. The override ranges from 0～200%(21 gears with 10% per gear). In AUTO mode, if the feedrate override is adjusted to zero, the feeding is stopped by the system with 0 cutting override displayed. The execution is continued if the override is readjusted.

7.5 Automatic Acceleration/Deceleration

The system enables the motor to perform acceleration/deceleration control at the beginning and the end of the movement, which thus obtains a stable start and stop. In addition, the automatic acceleration/deceleration can also be applied when the moving speed is changed, the speed thus can be changed steadily. Therefore, the acceleration/deceleration needs not to be considered during programming.

Rapid traverse: Pre-acceleration/deceleration （ 0 : linear type ; 1 : S type ）
Post acceleration/deceleration（0:linear type;1:exponential type）
Cutting feed: Pre-acceleration/deceleration （ 0 : linear type ; 1 : S type ）
Post acceleration/deceleration（0:linear type;1:exponential type）
MANUAL feed: Post acceleration/deceleration（0:linear type;1:exponential type）
(Set the common time constant for each axis by parameters)

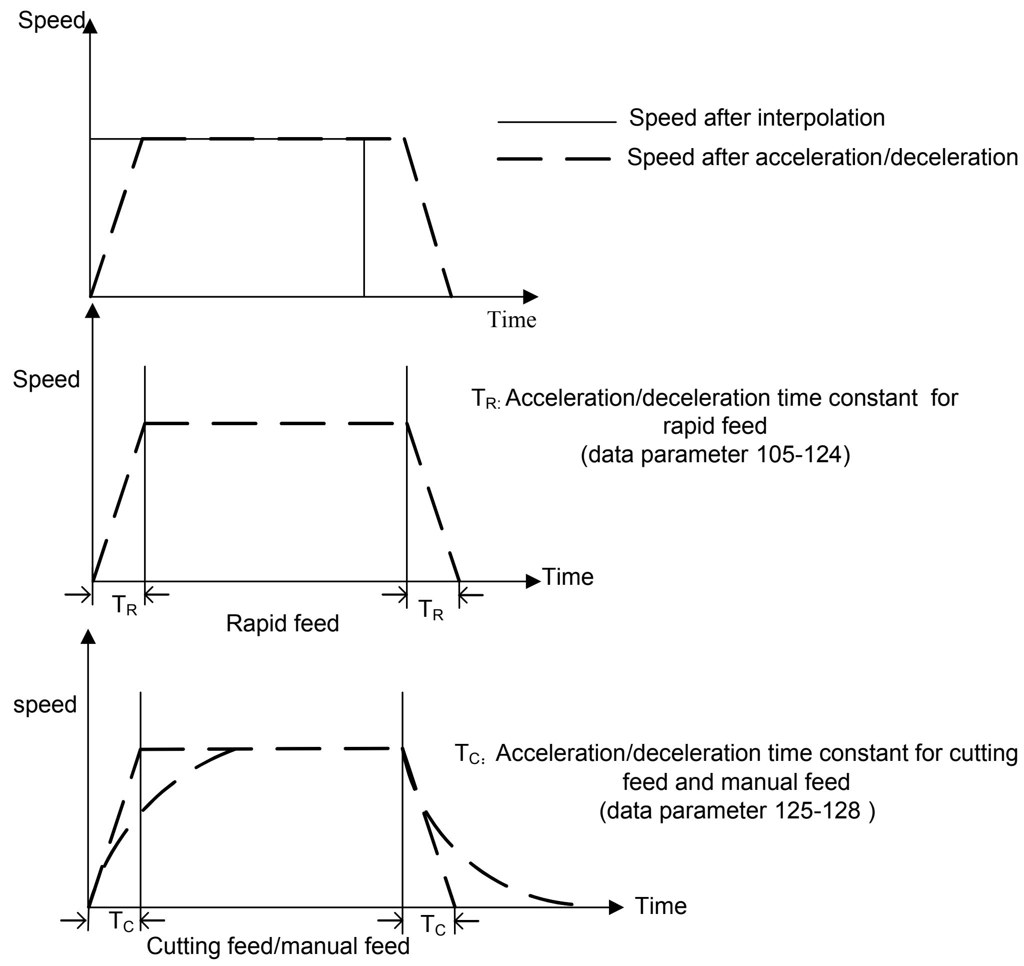


Fig. 7-5-1

Ⅰ Programming

7.6 Acceleration/Deceleration at the Corner in a Block

Example: If a block containing only Y movement is followed by a block containing only X movement, the latter X block accelerates as the former Y block decelerates. The tool path is as follows:

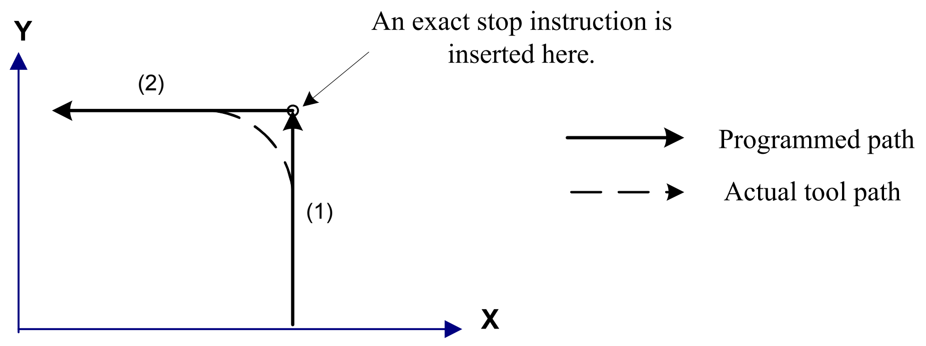


Fig. 7-6-1

If an exact stop code is inserted, the tool is moved along the real line as in the above figure by the program, otherwise the bigger the cutting feedrate is, or the longer the time constant of the acceleration/deceleration is, the bigger the arc at the corner is. For circular code, the actual arc radius of the tool path is smaller than the arc radius specified by the program. The mechanical system permitting, reduce the acceleration/deceleration time constant as far as possible to minimize the error at the corner.

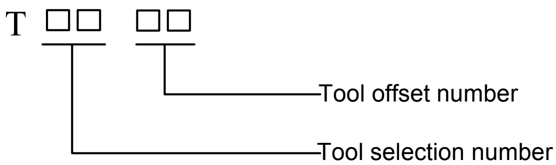
Chapter 8 Tool Function

8.1 T Command Format Meaning

By specifying a numerical value (up to 4 digits) following address T, the tools on the machine can be selected.

Two or more than T codes cannot be in the same block. If no alarm occurs when codes in the same group are in the same block, the last T code takes effect. Refer to the manual provided by the tool machine builder for the digits after address T and the corresponding machine operation of T code.

T code represents the following meanings:



a) Tool selection

The tool selection is performed by specifying a T code relative to a tool number. Relationship between a tool selection number and a tool is referred to the machine manufacture.

b) Tool offset number

It is used to select an offset value relative to an offset number. An offset value is input by keyboard unit. A corresponding offset number has two offset amount, one of which is used to X and another of which is used to Z. Refer to Operation, Tool Compensation Display, Modification and Setting.

Table 8-1-1

Offset number	Offset amount	
	X offset amount	Z offset amount
01	0.040	0.020
02	0.060	0.030
03	0	0
..	.	.
..	.	.
..	.	.

When a T code is specified and its offset number is not 00, the tool offset is valid.

When an offset number is 00, the tool offset function is cancelled.

The offset value range is set below:

mm input：-9999.999 mm～9999.999mm

8.2 Tool Offset

Offsetting X, Z is for a programmed path. T code specifies an offset number’s offset value which is added or subtracted at the end of each block.

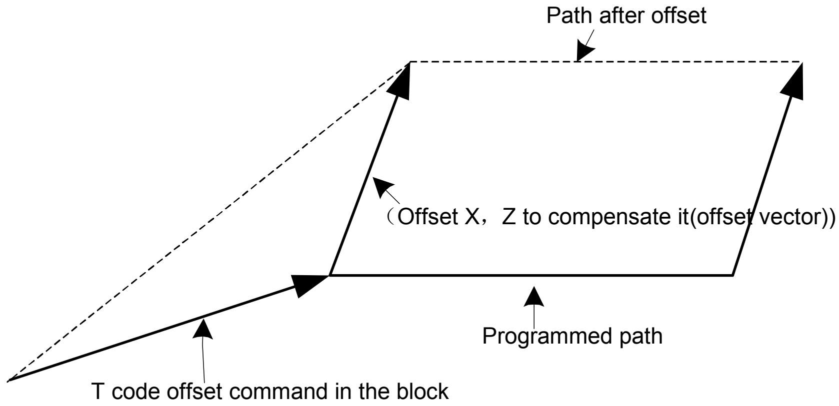


Fig. 8-2-1

a）Offset vector

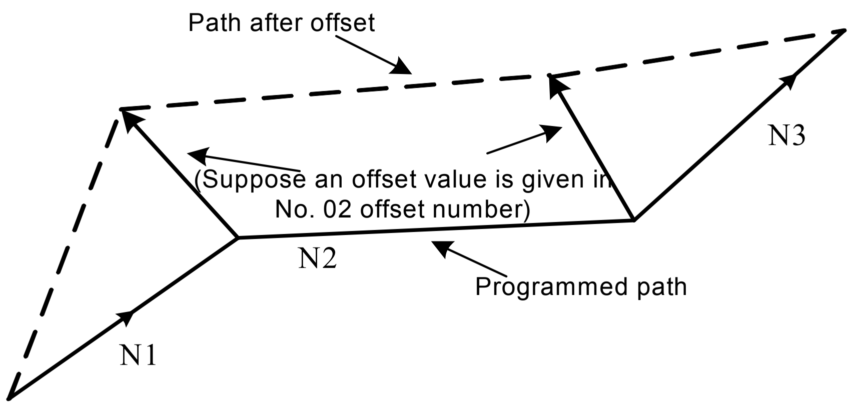
In the above figure, a vector which offsets X, Z is called an offset vector. A compensation takes effect to offset the vector.

b）Offset cancel

When T code's offset number selects 00, the offset is called. When the cancel is performed at the end of a block, the offset vector is zero.

N1 G01 U50.0 W100.0 T0202；
N2 W100.0：
N3 U0.0 W50.0 T0200；

Offset path

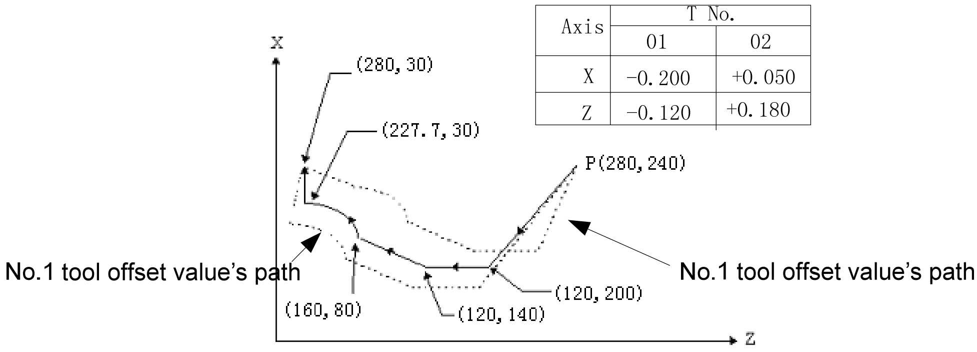


(Suppose an offset value is given in No. 02 offset number)

Fig. 8-2-2

8.3 Programming Example

Tool nose offset（Z，X）	tool
Tool #1……B（0.120，0.200）	01
Tool #2……C(-0.180，-0.050)	02



Axis	T No.	
	01	02
X	−0. 200	+0. 050
Z	−0. 120	+0. 180

Fig. 8-3-1　tool offset value compensation example -2

(Programming example 1)

G00 X280.0 Z240.0；

G00 X120.0 Z200.0 T0101；

G01 Z140.0 F30；

X160.0 Z80.0；

G03 X227.7 Z30.0 R53.81；

G00 X280.0 T0100；

#1 tool nose is the same as the program's programmed path.

（Programming example 2）

Rewrite the example 1 to the #2 tool's tool nose path is the same as the programmed path.

Ⅱ OPERATION

Chapter 1 Operation Panel

1.1 Panel Layout

GSK980TC3 series including GSK980TC3, GSK980TC3-V separately use horizontal and vertical structure, divided into LCD area, edit keyboard area, soft key functional area and machine control area, which is shown bellows:

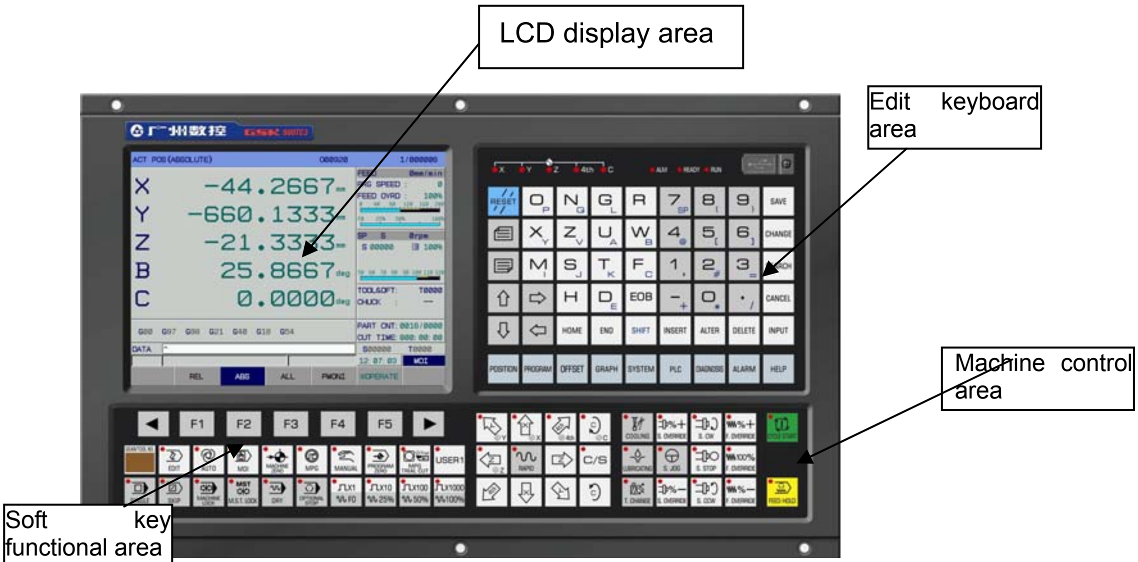


Fig. 1-1-1 GSK980TC3 panel

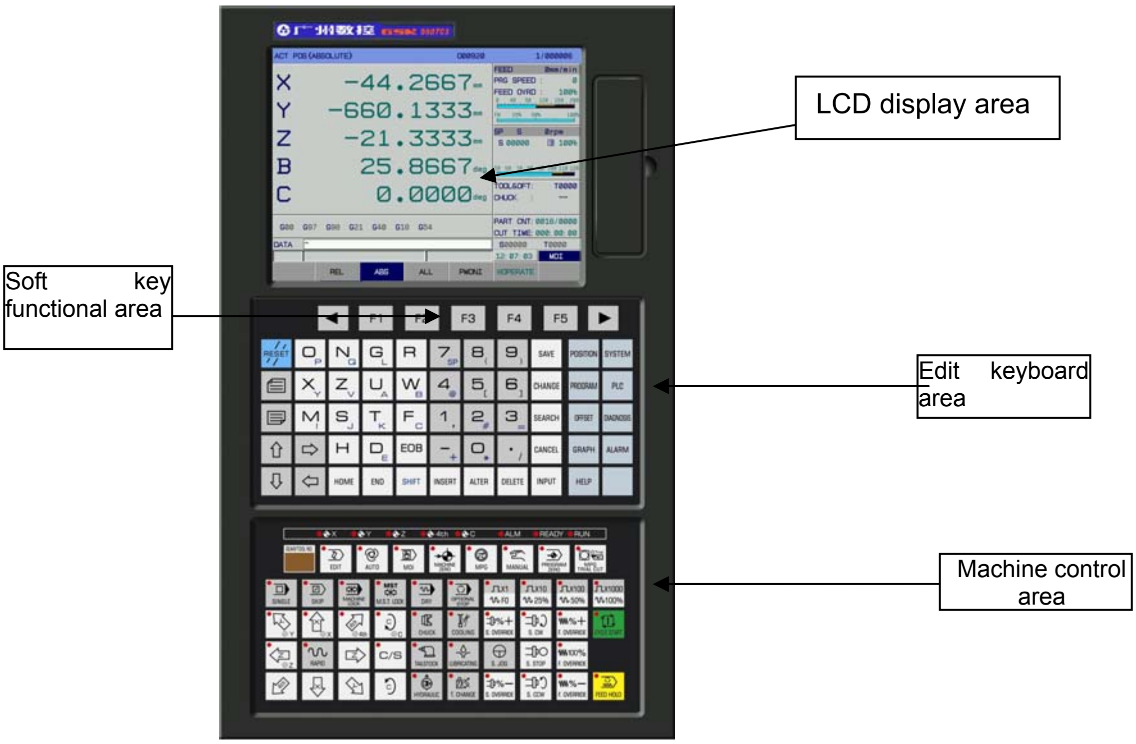


Fig. 1-1-2 GSK980TC3-V panel

1.2 Panel Function Explanation

1.2.1 LCD（Liquid Crystal Display）Display Area

GSK 980TC3, GSK980TC3-V use a color 8.4 inch LCD with resolution 800×600.

1.2.2 Editing Keyboard Area

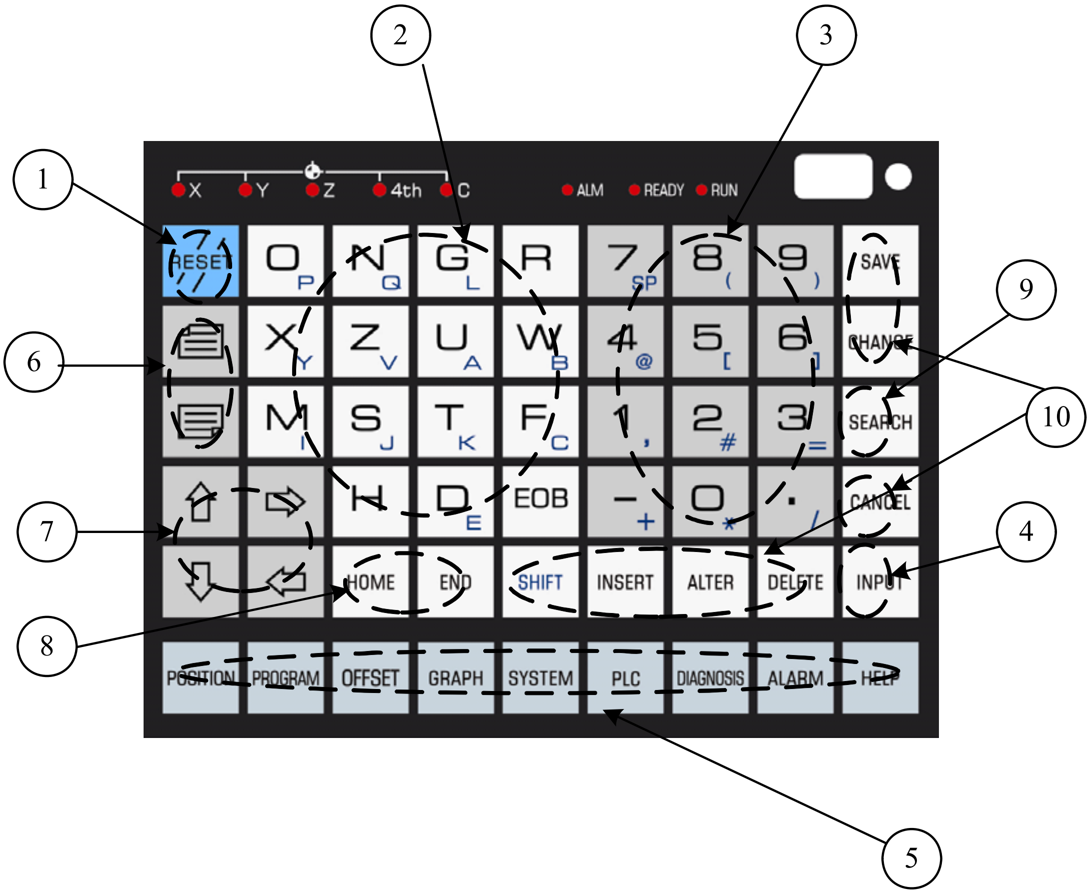


Fig.1-2-2-1 GSK980TC3 editing keyboard area

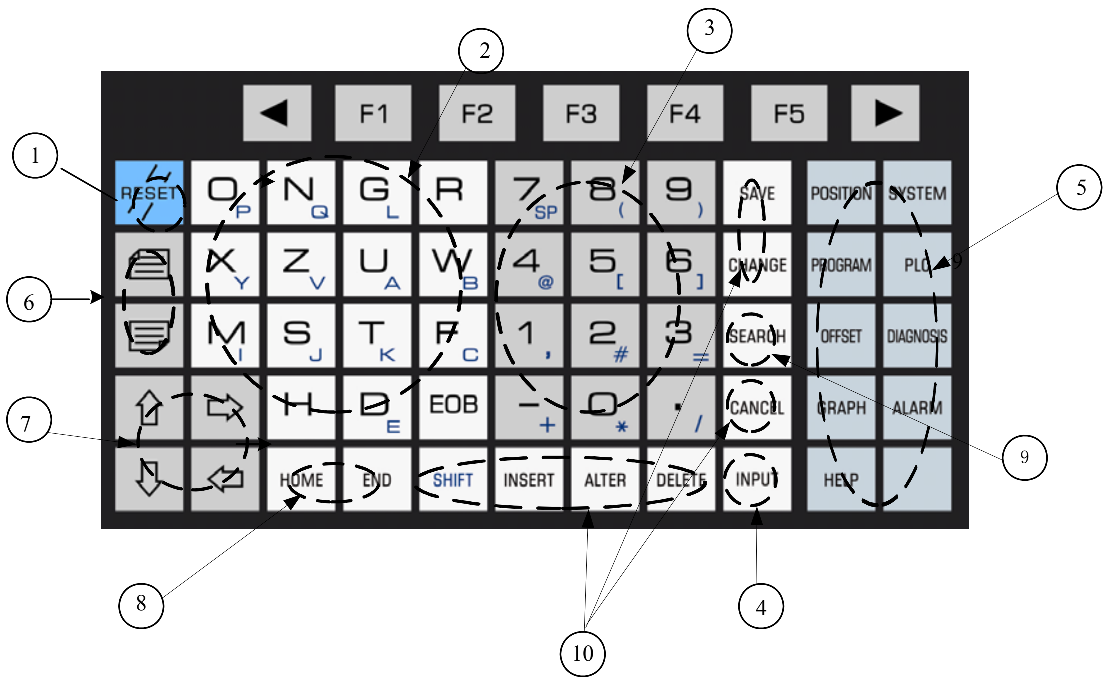


Fig.1-2-2-2　　GSK980TC3-V editing keyboard area

The functions of the keys on the editing keyboard area are divided into 10 small areas, which are explained as follows:

No.	Designation	Explanation
1	Reset key	For system reset, feed and output stop
2	Address key	For inputting addresses in MDI mode
3	Number key	For inputting numerical values in MDI mode
4	Input key	For Inputting numerical values, addresses or data into the buffer area; confirming the operation result
5	Screen operation key	By pressing any of the keys, the corresponding page is entered. See chapter 3 for details.
6	Page key	For page switching in the same display mode, and page down/up in the program
7	Cursor key	For moving the cursor in different directions
8	Editing key	For moving the cursor to the beginning or the end of a block or a program.
9	Search key	For searching data and addresses to view and modify
10	Editing key	For inserting, modifying or deleting a program or a block during programming, by using compound keys.

1.2.3　Screen Operation Keys

There are 8 display keys for operation pages and 1 display key for the help page on the panel in this system. See the figure below:

Graphic page

Alarm page

Help page

PLC page

Diagnosis page

System page

Position page

Program page

Setting page

Name	Function	Remark
Graphic page	Press this key to enter graphic page	Subpages for graphic parameters and graphic display can be viewed by switching corresponding soft keys. The center, size and ratio for the graph are set using graphic parameters
Alarm page	Press this key to enter alarm page	Subpages for a variety of alarm message can be viewed by switching corresponding soft keys.
Help page	Press this key to enter help page	Help message about the system can be viewed in this page by switching corresponding soft keys.
PLC page	Press this key to enter PLC page	The version of the PLC ladder and the configuration of system I/O can be viewed on this page, and the modification for PLC ladder is available in MDI mode.
Diagnosis page	Press this key to enter diagnosis page	The states of I/O signals on the system side can be viewed in this page by switching corresponding soft keys
System page	Press this key to enter system page	Subpages for tool offsets, parameters, macro variables and screw pitch can be displayed by switching corresponding soft keys
Position page	Press this key to enter position page	Subpages for relative coordinates, absolute coordinates and all coordinates of the current point and PLC can be displayed by switching corresponding soft keys
Pgrogram page	Press this key to enter program page	Subpages for programs, MDI, current/mode, current/time, and program directory can be displayed by switching corresponding soft keys. Program names in different pages can be viewed by pressing page keys in directory subpage.
Offset page	Press this key to enter the tool offset page	Three subprograms to be converted by a corresponding softkey, i.e., display offset page, workpiece coordinate page, macro variable page

Note: The page switch above can also be done by pressing corresponding function keys repeatedly after bit parameters NO:25#0～25#7, NO:26#6～26#7 are set. Refer to CHAPTER 3 in this manual for the explanation for each page.

1.2.4 Machine Control Area

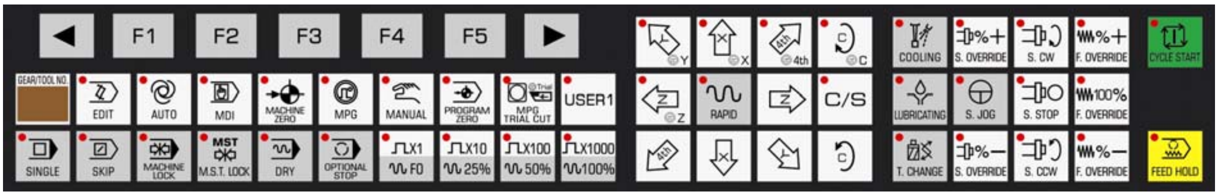


Fig. 1-2-4-1 GSK980TC3 machine control area

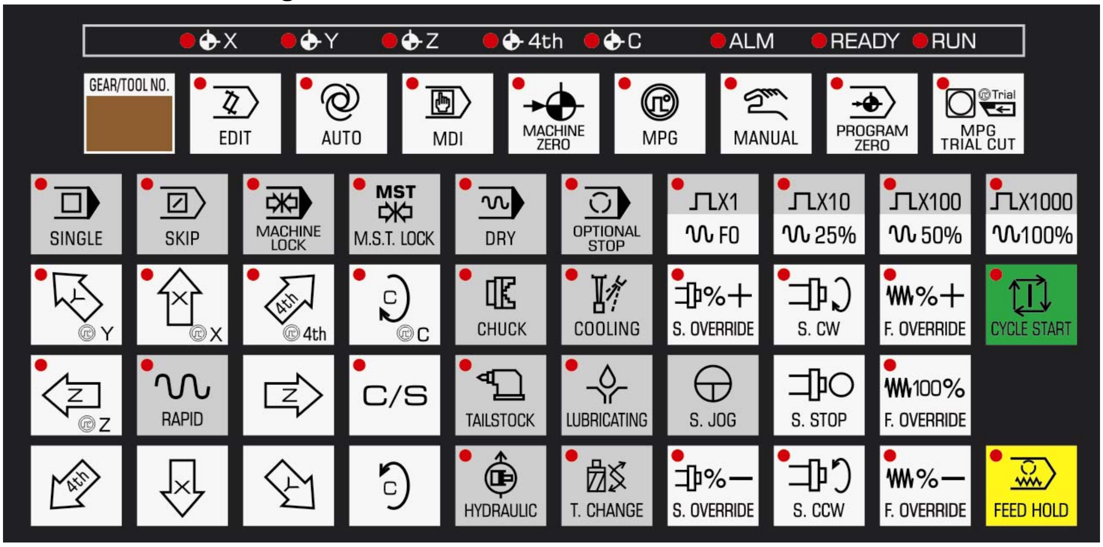
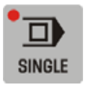
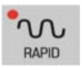
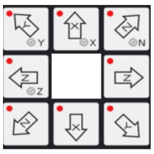


Fig. 1-2-4-2 GSK980TC3-V machine control area

The use and function definition of the basic keys for the machine control area of GSK980TC3 and GSK980TC3-V are the same.

Keys	Designation	Explanation	Remarks and operation explanation
	Edit mode key	To enter edit mode	Switching to Edit mode in Auto mode, MDI mode and DNC mode. System decelerates to stop after current block is executed
	Auto mode key	To enter auto mode	In this mode, program in internal memory is selected
	MDI mode key	To enter MDI mode	Switching to MDI mode in Auto mode, system decelerates to stop after current block is executed
	Machine zero mode key	To enter machine zero mode	Switching to Machine zero mode in Auto mode. System immediately decelerates to stop
	Program zero mode key	To enter program zero mode	Switching to Program zero mode in Auto mode. System immediately decelerates to stop
	Manual mode key	To enter manual mode	Switching to Manual Mode in Auto mode, system immediately decelerates to stop
	MPG mode key	To enter MPG mode	Switching to MPG mode in Auto mode, system immediately decelerates to stop
	Block skip key	For a block preceding with "/" sign. If it is on, the indicator lights up And the block is skipped.	Auto mode, MDI mode, DNC mode
	Single block key	For switching the execution between single block and blocks. If it is on, the indicator Lights up.	Auto mode, MDI mode, DNC mode
	Dry run switch	The indicator lights up if dry run is valid.	Auto mode, MDI mode, DNC mode
	Optional stop ON/OFF key	Whether the operation is stopped after a block containing M01 is executed.	Auto mode, MDI mode, DNC mode
	M.S.T. lock switch	M.S.T. function output is invalid if the indicator for M.S.T. function lock lights up.	Auto mode, MDI mode, DNC mode

II Operation

Keys	Designation	Explanation	Remarks and operation explanation
	Machine lock switch	The indicator lights up if it is on, and the axis movement output is invalid.	Auto mode, MDI mode, Machine zero, MPG mode, Step mode, MANUAL mode, DNC mode
	Lubricant oil switch	Machine lubricant ON/OFF	Any mode
	Cooling switch	Cooling ON/OFF	Any mode
	Spindle control keys	Spindle CCW Spindle stop Spindle CW	MPG mode, step mode, manual mode
	Spindle override keys	Spindle speed adjustment (spindle speed analog control valid)	Any mode
	Spindle JOG switch	Spindle JOG ON/OFF	Manual mode, Step mode, MPG mode
	C/S axis rotation CW/CCW	C/S axis rotation CW/CCW	Valid when the spindle is analog amount control mode in Manual mode
	Tool change switch key	Tool change switch	Valid in Manual mode, Step mode, MPG mode
	Bus switch key	Bus switch	USER1 single block runs, press the mode during running a program, the program is executed by block by block
	Rapid traverse key	Rapid traverse ON/OFF	Manual mode
	Rapid override, manual step, MPG override selection key	Rapid override, manual step, MPG override selection key	Auto mode, MDI mode, Machine zero return mode, MPG mode, Manual mode
	Manual feed key	X, Y, Z, C axis positively/ negatively moves in Manual, Step mode, the axis' positive direction is MPG selecting a axis	Machine zero return mode, Manual mode, MPG mode
	Feed hold key	Press it and the system stops the automatic run	Auto mode, MDI mode
	Cycle start key	Press it and the program automatically runs	Auto mode, MDI mode

Note 1：A block with more than 1 "/" sign at its beginning is skipped by the system even if the skip function is OFF.

Note 2：In Manual mode, the manual feedrate override is regulated by the feedrate override switch without pressing the rapid traverse key.

Note 3：Keys in <> are on the panel; keys in【 】are soft keys at the bottom of the screen;【 】is a page corresponded to the current soft key; ⊞ means there is a submenu in the menu.

Note 4：There is an user defined key USER1.

Chapter 2 System Power on/off and Safety Operation

2.1 System Power-on

Before **GSK980TC3** CNC system is powered on, the user must ensure that:

1. The machine state is normal.
2. The voltage of the power supply conforms to the requirement of the machine.
3. The wiring is correct and reliable.

The current position (relative coordinates) is displayed after system self-check and initialization.

Fig. 2-1-1

2.2 Power off

Before turning off the system, make sure that:

1. The axes X, Y, Z of the CNC are at halt;
2. Miscellaneous functions (spindle, pump, etc.) are off.
3. The CNC power is cut off prior to cutting off the machine power.

When cutting off the power, check that:

1. The LED, which indicates the cycle start on the operator panel, is off.
2. All the movable parts of the CNC machine tool are at halt.
3. Press POWER OFF button to turn off the power.

Cutting off the power in an emergency

The power should be cut off immediately to prevent accidents in an emergency situation during the machine running. However, the zero return, tool setting, etc. must be performed again because an error between system coordinates and actual coordinates may occur after power-off.

Note: See the manual provided by the machine tool builder for the machine power cut-off.

2.3 Safety Operations

2.3.1 Reset Operation

With key  pressed, the system enters the reset state:

1. All axes movement stops;
2. The M functions are ineffective;
3. Whether the G codes are saved after resetting is determined by NO:35#1～NO:35#7 and NO:36#0～NO:36#7;
4. Whether F, H, D codes are cleared after resetting is determined by NO:34#7;
5. In MDI mode, whether the edited program is deleted after resetting is determined by NO:28#7;
6. Whether the relative coordinates are cancelled after resetting is determined by NO:10#3;
7. In non-Edit mode, whether the cursor returns to the beginning of the program after resetting is determined by NO:10#7;
8. Whether macro local variables #1～#50 are cleared after resetting is determined by NO:52#7;
9. Whether macro common variables #100～#199 are cleared after resetting is determined by NO:52#6;
10. Resetting can be used during abnormal system output and coordinate axis action.

2.3.2 Emergency Stop

If the Emergency Stop button is pressed during machine running, the system enters into emergency state and the machine movement is stopped immediately. Release the button (usually rotate the button towards left) to exit the state.

Note 1: Confirm the faults have been removed before releasing the Emergency Stop button;

Note 2: Perform Reference Point Return again after releasing the Emergency Stop button to ensure the coordinate position is correct.

In general, the emergency stop signal is a normal closed signal. When the contact point is open, the system immediately enters into the emergency stop state and emergently stops the machine. The connection for the emergency stop signal is as follows:

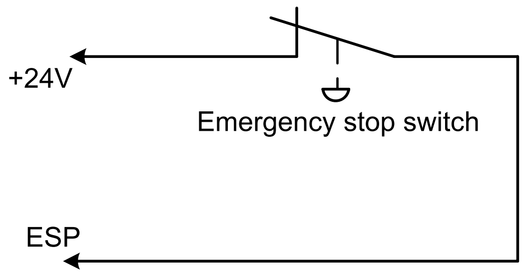


Fig. 2-3-2-1

2.3.3 Feed Hold

Users can suspend the execution pressing key 

during the machine running. Please note that the execution is not suspended in rigid tapping codes and cycle codes until the current code is executed.

2.4 Cycle Start and Feed Hold

The keys 

and 

are used for the program start and dwell operations in Auto mode, MDI mode and DNC modes. Whether the external start and dwell is used is set by PLC address **K5.1**.

Note 1：Switch Auto/MDI mode. Before the current block is executed, the cycle start is valid, <Feed hold> key is pressed, the feed hold is invalid.

Note 2：Switch Auto/MDI mode into Edit mode. Before the current block is executed, the cycle start is invalid, <Feed hold> key is pressed, the feed hold is invalid.

Note 3：Switch Auto/MDI mode into the Machine Zero Return, Step, Manual, MPG mode. Press the feed hold key and the feed hold function is invalid.

Note 4：When the cycle start key is valid, the Auto/MDI mode is switched into the Edit mode, and the feed hold key is pressed before the current block is executed, the feed hold function is invalid.

2.5 Overtravel Protection

Overtravel protection must be employed to prevent the damage to the machine due to the overtravel of the X, Y, or Z axis.

2.5.1 Hardware Overtravel Protection

The overtravel limit switches are fixed at the positive and negative maximum stroke of the machine X, Y and Z axes respectively. If the overtravel occurs, the moving axis decelerates and stops after it touches the limit switch. Meanwhile, the overtravel alarm is issued.

Explanation:

Overtravel in auto mode

In Auto mode, if the tool hits the stroke limit switch during the movement along an axis, all the axis movements are decelerated to stop with the overtravel alarm being issued. The program execution is stopped at the block where the overtravel occurs.

Overtravel in Manual mode

In Manual mode, if any axis contacts the stroke limit switch, all axes will slow down immediately and stop.

2.5.2 Software Overtravel Protection

The software stroke ranges are set by the data parameters P66~P73, with the machine coordinates taken as the reference values. Overtravel alarm occurs if the moving axis exceeds the setting software stroke. Whether the stroke check is performed after power-on and before manual reference point return is determined by bit parameter N0:11#6 (0: No, 1: Yes). Whether the overtravel alarm is issued before or after the overtravel when the software limit overtravel occurs is set by bit parameter N0:11#7 (0: before, 1: after). After the overtravel occurs, move the axis out of the overtravel range in the reverse direction in Manual mode to release the alarm.

2.5.3 Overtravel Alarm Release

Method to release the hardware overtravel alarm: In manual or MPG mode, press key 

on the panel, then move the axis in the reverse direction (for positive overtravel, move negatively; for negative overtravel, move positively).

Chapter 3 Page Display and Data Modification and Setting

3.1 Position Display

3.1.1 Four Types of Position Display

Press key 

to enter position page, which consists of 【REL】, 【ABS】, 【All】and 【PMONI】. The four subpages can be viewed using corresponding soft keys, as is shown below:

1) Relative coordinate: It displays the position of the current tool in the relative coordinate system by pressing soft key【REL】. (See fig. **3-1-1-1)**:

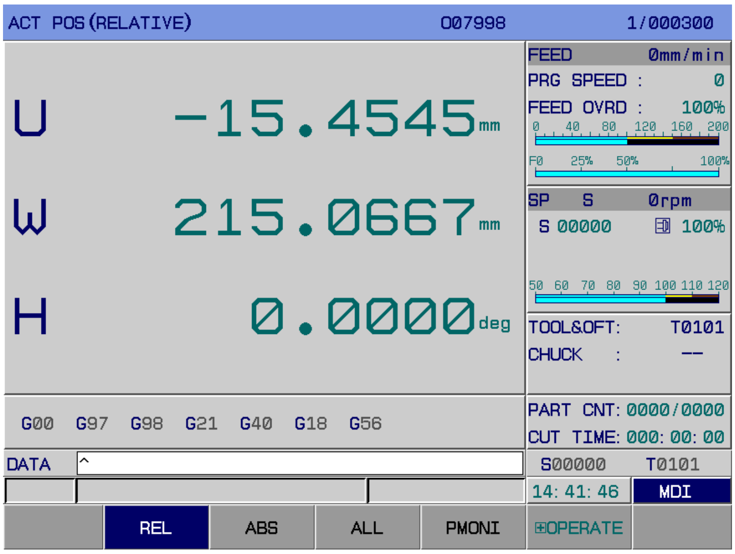


Fig. 3-1-1-1

2) Absolute coordinate: It displays the current position of the tool in absolute coordinate system by pressing soft key【ABS】(see Fig.3-1-1-2).

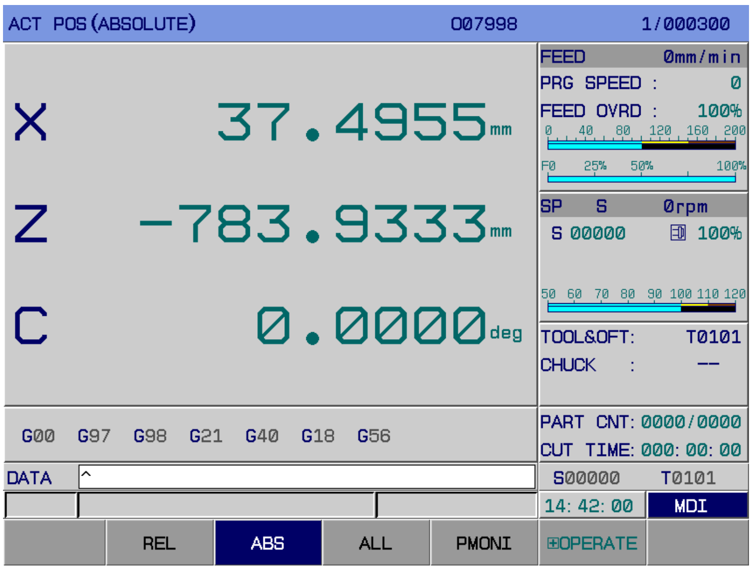


Fig. 3-1-1-2

3) ALL: It enters【ALL】page by pressing soft key【ALL】, displaying the following items:

(A) The position in relative coordinate system;
(B) The position in absolute coordinate system;
(C) The position in machine coordinate system;
(D) The offset amount (displacement) in MPG interruption;
(E) Speed component;
(F) Remaining distance (only displayed in Auto, MDI and DNC mode)

The display is shown below（Fig.3-1-1-3）:

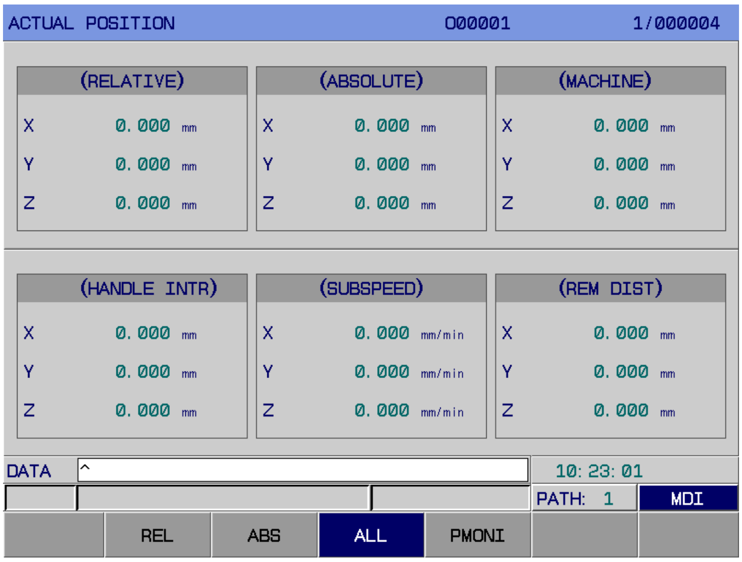


Fig. 3-1-1-3

4）In【ALL】page, pressing soft key【**OPERATION**】to display the following items:（See Fig. 3-1-1-4）

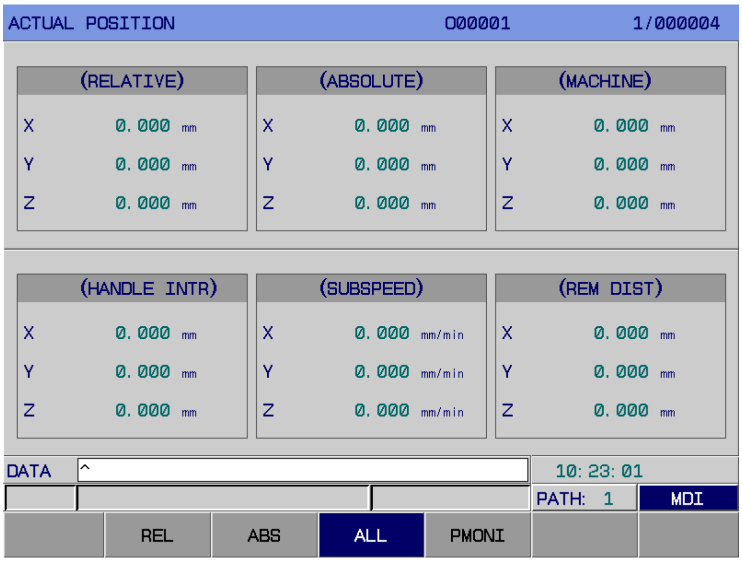


Fig. **3-1-1-4**

5）Monitor mode

It enters 【PMONI】 page by pressing soft key 【PMONI】. In this mode, the absolute coordinates, relative coordinates of the current position as well as the modal message and blocks of the program being executed can be displayed (See Fig. 3-1-1-5):

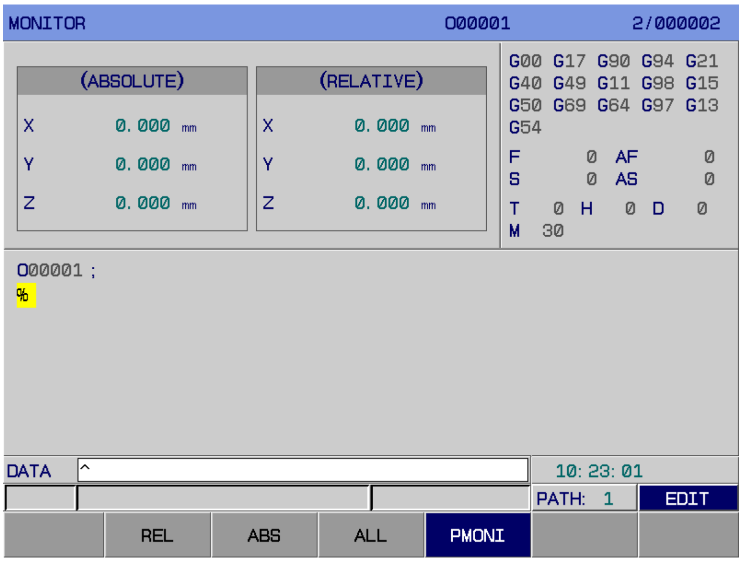


Fig. **3-1-1-5**

Note 1: Whether the modes are displayed in 【PMONI】 page can be set by parameter NO:23#6. When BIT6=0, the machine coordinates are displayed in the position where the modal codes are displayed.
Note 2: In <MACHINE ZERO>, <STEP>, <MANUAL>, <MPG>, <MACHINE ZERO>, <EDIT>modes, the intermediate coordinate system is a relative one; while in <AUTO>, <MDI> modes, it is the distance to go.

II Operation

3.1.2 Display of Cut Time, Part Count, Programming Speed, Override and Actual Speed

The programmed speed, actual speed, feedrate and rapid override, G codes, tool offset, part number, cut time, spindle override, spindle speed, tools etc. can be displayed on the subpages【REL】and【ABS】of page <POSITION> (see Fig.3-1-2-1) :

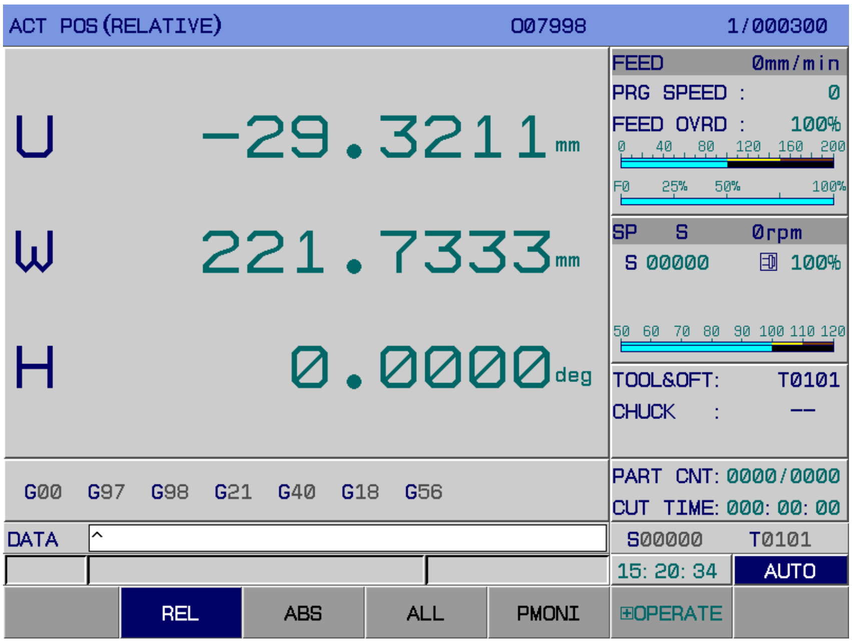


Fig. **3-1-2-1**

The meanings of them are as follows:

Speed: The actual cutting speed overridden.

Programmed speed: Speed specified by F code.

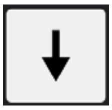
Feedrate override: Feed override selected by feedrate override keys.

Rapid override: Rapid override selected by rapid override keys.

G codes: The values of the G codes in the block being executed.

Part count: “0002/0000”，“0002” is the machined part quantity, “0000” is the required machining total part quantity. When M30 or M02 is executed in Auto mode, the count increases by 1, in other modes, the count does not increase; when the machined part quantity is more than or equal to the required machining part quantity, the system sends the signal F61#1=1，and the application method of the signal is defined by the machine tool manufacturer.

Cut time: Time counting starts after Auto run starts, with a unit of “hour: minute: second”.

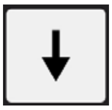
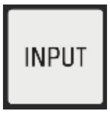
S00000: Specify the speed. Press key  in Page ACT POS(RELATIVE), locate the cursor to “S 0000”: in this way S value can be changed (range:0~value set by data parameter P258).

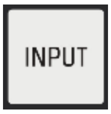
Tool offset: A tool number specified T code in program.

Note: The part count is memorized after power-down.

Ways to clear part count and cut time:

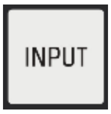
1）Switch to POSITION page, select MDI mode

2）Press key  to locate the cursor to the PRT CNT item, input data and press key 

for confirmation; if key 

is pressed directly, the part count will be cleared.

3）Shift to CUT TIME by keys Up and Down.

4）Press key 

to clear the CUT TIME.

Note 1: To display the actual spindle speed, an encoder must be applied to the spindle.
Note 2: The actual speed= the programming speed F × override; The speed of each axis is set by data parameters P88～P92 in G00 mode and it can be overridden by rapid override; the dry run speed is set by P86.
Note 3: The programming speed for feed per revolution is displayed when the block involving feed per revolution is being executed.

3.1.3 Relative Coordinate Clearing and Halving

The steps for clearing relative coordinate position are as follows:

1) Enter any page that displays the relative coordinates (Fig. 3-1-3-1);

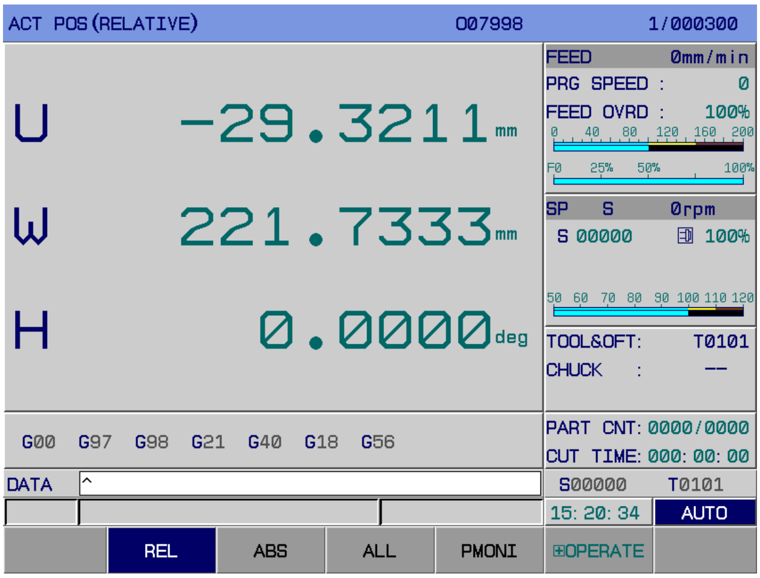
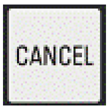


Fig. 3-1-3-1

2） **Clearing operation**: Press and hold key “X” till X in the page flickers, then press key  to clear the relative coordinate in X axis; (Fig. 3-1-3-2)**:**

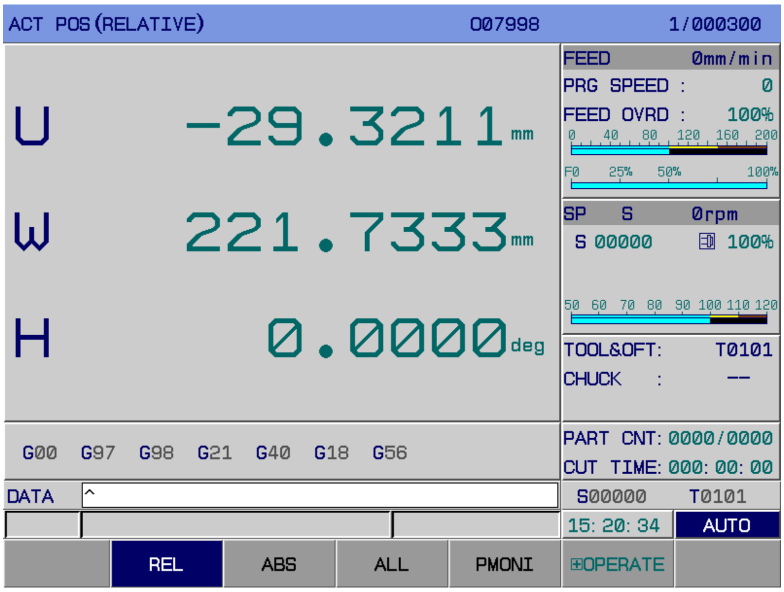
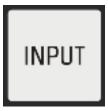
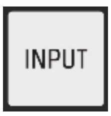


Fig. 3-1-3-2

3）**Halving operation:** Press and hold key “X” till “X” in the page flickers, then press key  to halve the relative coordinate in X axis. (The relative coordinate of the axis is divided by 2);

4）**Coordinate setting**: Press **X**” and “**U**”, and “X” in the page flickers, input the data to be set and press key  for confirmation, then the data will be input into the coordinate system.

5）Z clearing method is the same as the above.

3.1.4 Bus Monitor Page Display

When the system uses the Ethernet bus communication mode, 

is pressed to enter the position page. Press【  】 to enter 【PMONI】page. In the page, the system simultaneously displays the current position's machine coordinates, multi-coil position, single-coil position, grating position, motor speed, motor load(% is a percentage of rated load). The page is convient to debug the machine and monitor the servo's current run state. (See Fig. 3-1-5)

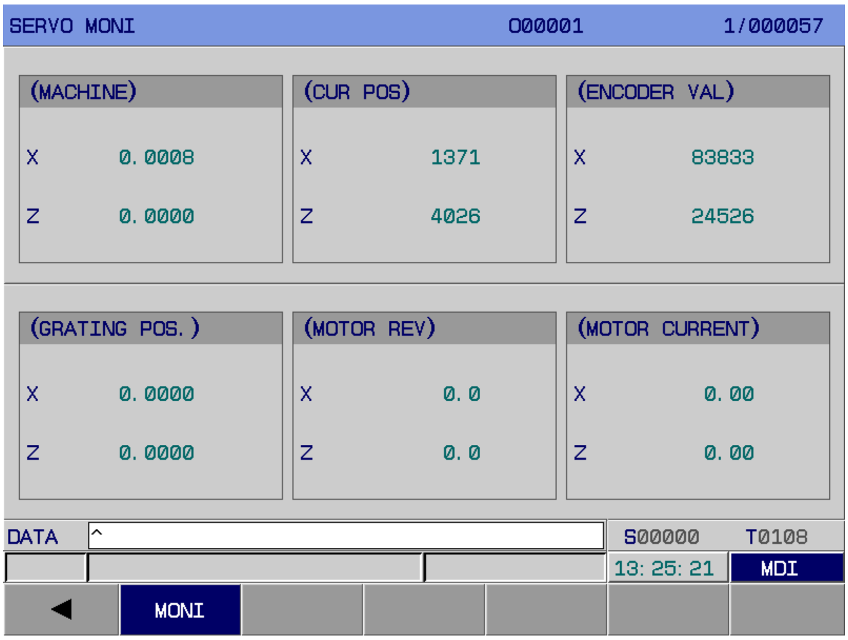


Fig. **3-1-4**

3.2 Program Display

Press key 

to enter program display page which consists of 5 subpages: 【⊞PRG】,【MDI】and 【DIR】. They can be viewed and modified by corresponding soft keys (See Fig.3-2-1).

1） Program display

Press soft key【⊞PRG】to enter program page. In this page, a page of blocks being executed in the memory can be displayed (See Fig. 3-2-1).

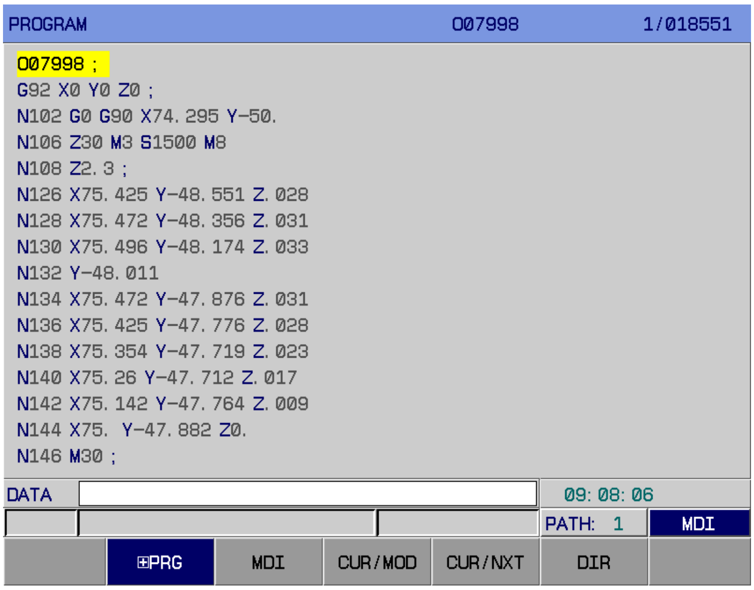


Fig. **3-2-1**

Press soft key 【⊞PRG】 again, and the program EDIT and modification page is entered (see Fig.3-2-2):

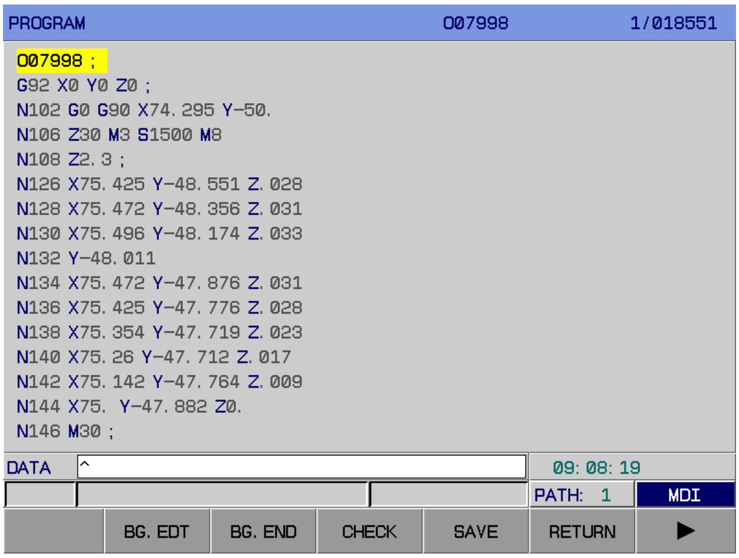


Fig. **3-2-2**

Press【▶】to enter the next page 

Press【▶】to enter the next page 

Press【◀】to return to the previous page 

Note: The 【CHECK】 function can only be performed in Auto mode.

【B. EDIT】and 【B. END】are used only in AUTO and DNC mode (background edit function). Functions of 【B.EDIT】are the same as the program edited in <EDIT> mode (See Chapter Ten "Program Edit"). Save the editing by 【B. END】or exit the background EDIT page by 【RETURN】 after editing.

2） MDI input display

Press soft key 【MDI】 to enter MDI page. In this mode, multiple blocks can be edited and executed. The program format is the same as that of the editing program. MDI mode is applicable to simple program testing operation (see Fig. 3-2-3) .

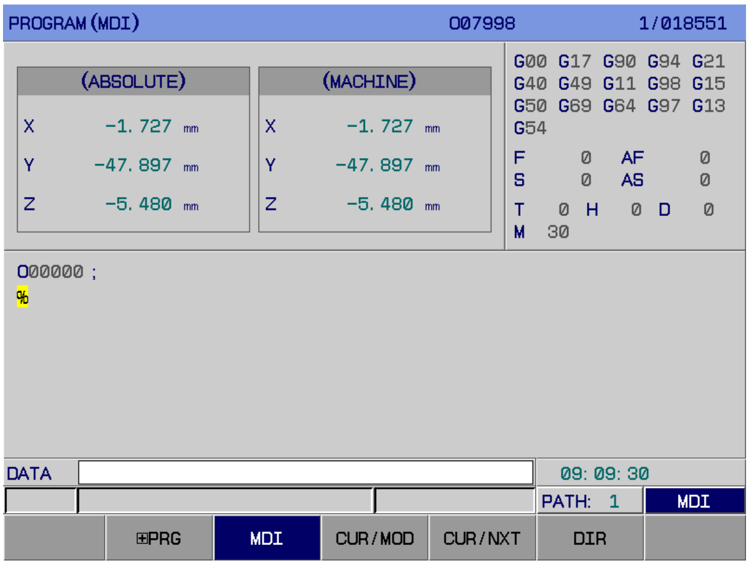


Fig. 3-2-3

3) Program (DIR) display

Ⅰ. Press soft key【DIR】to enter program (DIR) page, the contents of which are displayed as follows (Fig.3-2-4):

(a) PRG USED: The saved programs (including subprograms) /maximum number of the programs that can be saved.

(b) MEM USED: The capacity occupied by the saved programs /the total capacity for program storage.

(c) PROGRAM DIR: The sequence numbers of the saved programs are displayed in sequence.

(d) Previewing the program where the cursor is located

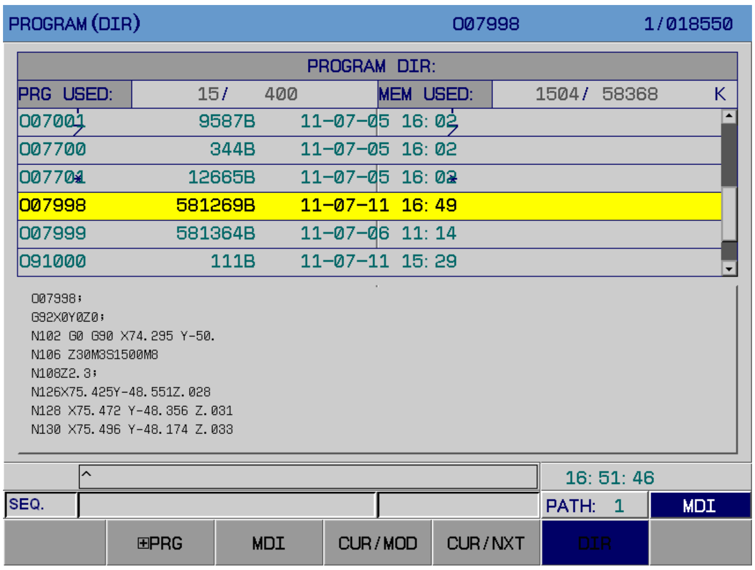


Fig. 3-2-4

Note: The program numbers in memory can be displayed by the page keys. The program names with more than 6 digits or irregular formats cannot be previewed.

3.3 System Display

Press key 

to enter system page, which consists of four subpages: 【CNC SETTING】, 【⊞PARA】, 【PITCH】,【⊞DATA】, 【⊞BUS】,【⊞TIME LIMIT STOP】, which can be displayed by pressing the corresponding soft key.

3.3.1 CNC Display, modification and setting

3.3.1.1 Modification and Setting for Password Authority

To avoid of machining programs and CNC parameters being modified viciously, GSK980TC3 provides authority setting function, and its password level is divided into 5, including 1 level (end-user level), 2 level (the system debugging level), 3 level (machine manufacture level), 4 level (system manufacture level) and 5 level (machining operation level). The system defaults the lowest level after power on (see Fig. 3-3-1-1-1).

1 level：modify some of the CNC's state parameters, data parameters.

2 level：modify the CNC's state parameters, data parameters, tool offset data and pitch.

3 level, 4 level：modify the CNC's state parameters, data parameters, offset data and transmitting PLC ladder diagram.

5 level：No password. Modify offset data, macro variables, executing operations by machine the operation panel. Must not modify the CNC's state parameters, data parameters and pitch data.

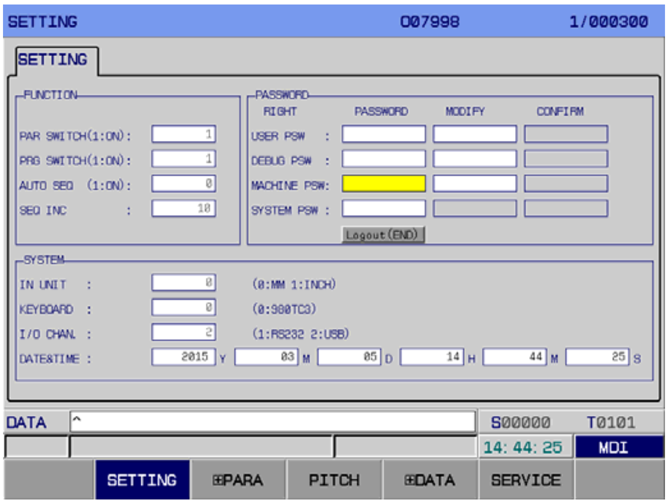
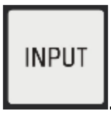
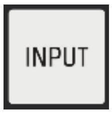


Fig. 3-3-1-1-1

1）After the system enters the page in <MDI MODE>, the cursor positions to the target position.

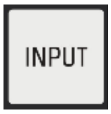
2）Input the corresponding password and press 

. When the input password is correct, the system prompts "Password Correct".

3）When the password needs to modify, 0~6 digits or letters are input. Press 

to confirm the input.

4）After modification, press 

, and the cursor moves to "Cancel（END）" button, and the system prompts："press<Input>key to confirm the cancel！"，After 

is pressed to confirm the cancel, the system prompts:" Cancel Completes!". At the same time, the cursor returns to the password toolbar. The password is automatically cancelled after power-off.

3.3.2 Display, Modification and Setting for Parameters

3.3.2.1 Parameter Display

1）**Bit parameter page**. Press soft key【⊞PARA】to enter parameter page.（See **3-3-2-1-1**）:

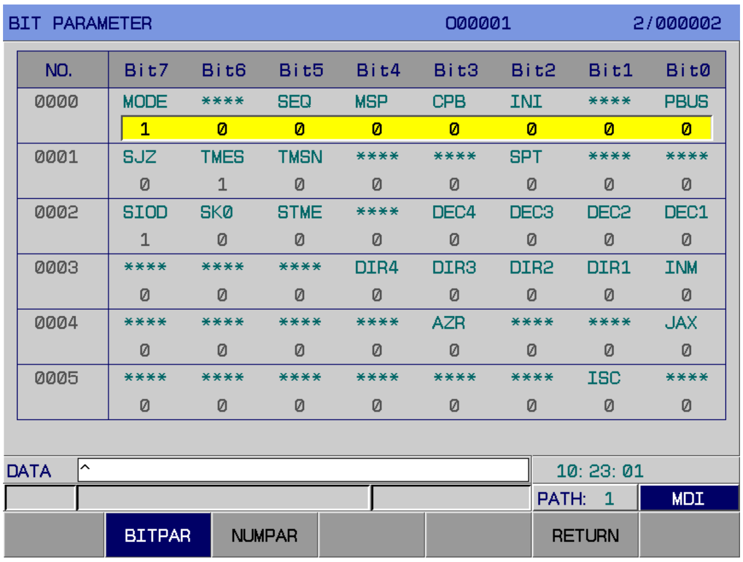
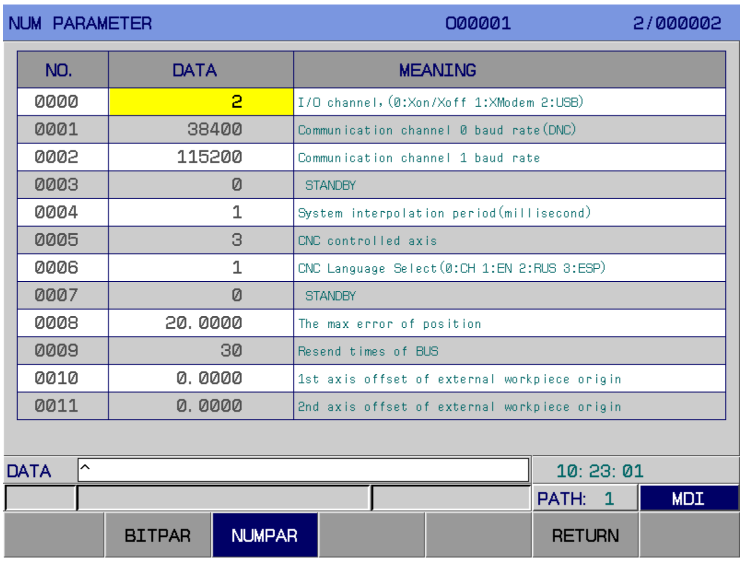


Fig. 3-3-2-1-1

Refer to APPENDIX 1 PARAMETERS for details.

2）**Number parameter page**　Press soft key【NUMPAR】to enter this page. (See fig. 3-3-2-1-2)**:**



NUM PARAMETER 000001 2/000002

NO.	DATA	MEANING
0000	2	I/O channel,(0:Xon/Xoff 1:XModem 2:USB)
0001	38400	Communication channel 0 baud rate(DNC)
0002	115200	Communication channel 1 baud rate
0003	0	STANDBY
0004	1	System interpolation period(millisecond)
0005	3	CNC controlled axis
0006	1	CNC Language Select(0:CH 1:EN 2:RUS 3:ESP)
0007	0	STANDBY
0008	20.0000	The max error of position
0009	30	Resend times of BUS
0010	0.0000	1st axis offset of external workpiece origin
0011	0.0000	2nd axis offset of external workpiece origin

Fig. 3-3-2-1-2

Refer to APPENDIX 1 PARAMETERS for details.

3） Common parameters

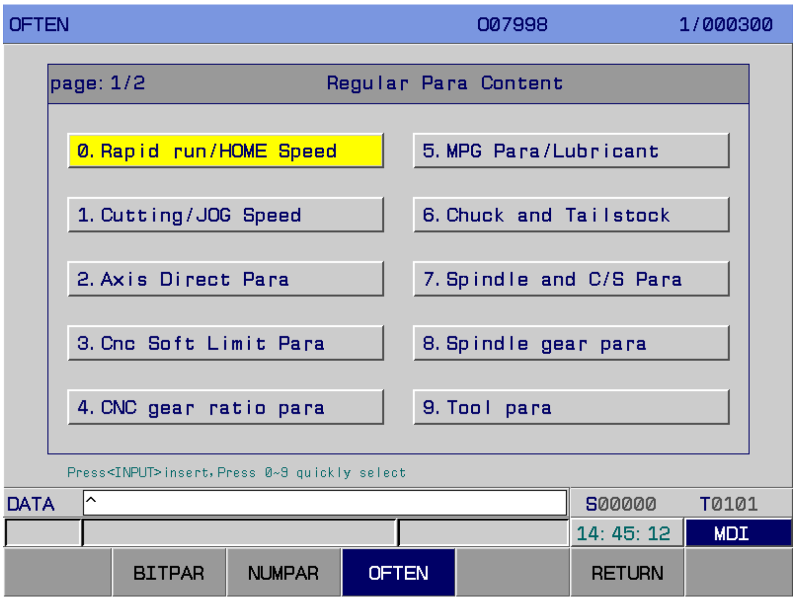


Fig. 3-3-2-1-3

In the above 【Common parameter】page, select the required parameter catalog number to enter the corresponding page. (See Fig. 3-3-2-1-4):

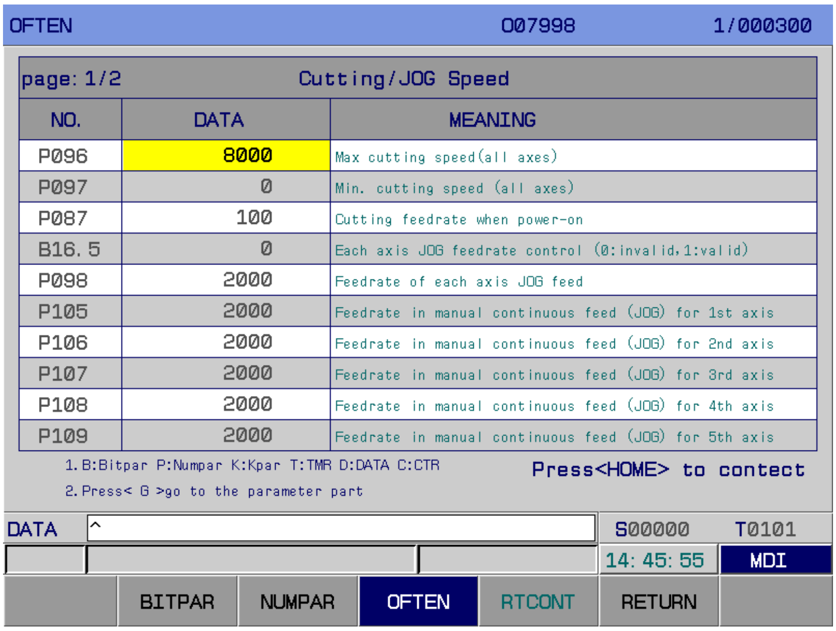


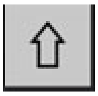
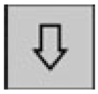
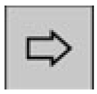
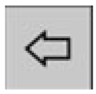
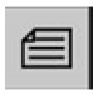
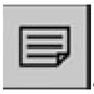
Fig. 3-3-2-1-4

Concrete operation methods:

1）Select MDI mode.

2）Press 

, and then press【NUMPAR】 to enter【Common Parameter】display page.

3）Select the parameter catalog by pressing , , , , , .

4）Move the cursor to the parameter number to be modified:

Method 1：press<Input> to enter the selected, press page up/down to display the page in which the set parameter is; press the direction key to move the cursor and to position the required parameter position to modification.

Method 2：press the corresponding digit key, and then press <Input> key，press page up/down to display the page in which the set parameter is; press the direction key to move the cursor and to position the required parameter position to modification.

5）Press <HOME> key or press【Return to catalog】to return to the common parameter catalog.

6）Press <G> key to skip the parameter page in which the parameter is selected.

7）Letters in front of the parameter number represent different parameter type.（B: bit parameter，P: data parameter，K:K parameter，T:T parameter，D:DATA parameter，C:CTR parameter）

8）Modify the maximum/minimum value range of data parameters in the NUMPAR parameter.

Note: Catalog of common parameters and parameter included in the catalog from the CNC's correspondingly allocated files can be freely selected by the user.

II Operation

3.3.2.2 Modification and Setting Parameter Values

1) Select MDI mode;

2) Press key 

to enter <SETTING> page, turn on the parameter switch. Set the parameter switch to 1.

3) Press key 

, then the soft key【⊞NUMPAR】to enter 【BITPAR】or【PAR】or【COMPAR】display page.

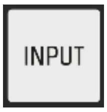
4) Move the cursor to the parameter number to be modified:

Method 1: Press page keys to display the parameter to be set; then move the cursor to the place to be modified;

Method 2: Press key 

to search after inputting the parameter number.

5) Input a new parameter value using number keys (corresponding passwords are required for modifying parameters of different levels).

6) Press key 

for confirmation, then the parameter value is input and displayed.

7) Turn off the parameter switch after setting all the parameters.

Note: There is no search function in【COMPAR】page.

3.3.3 Display, Modification and Setting for Screw Pitch Offset

3.3.3.1 Pitch Offset Display

Press soft key【PITCH】to enter pitch offset page, which is shown as follows (Fig. 3-3-3-1-1):

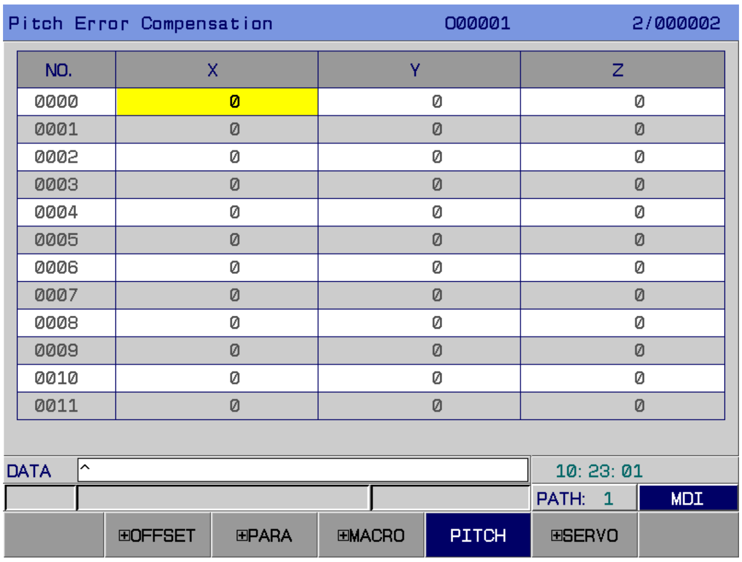


Fig. 3-3-3-1-1

II Operation

3.3.3.2 Modification and Setting for Pitch Offset

1) the pitch error compensation number is set by data parameters P226～P230, and the pitch error offset interval is set by data parameters P226～P230.

2) In <MDI> mode, input the offset value for each point in turn.

Note: Refer to Ⅳ 4 Installation and Connection of “GSK980TC3 CNC System PLC, Installation and Connection User Manual” for the setting of pitch offset.

3.3.4 Backup, Restoration and Transmission for Data

Press soft key【⊞DATA】to enter SETTING (DATA DEAL) page. The user data (such as ladder, ladder parameters, system parameter values, tool offset values, pitch offset values, system macro variables, custom macro programs and CNC part programs) can be backup (saved) and restored (read); and the data input and output via PC or U disk are also available in this system. The part programs saved in CNC are not affected during the data backup and restoration.（See Fig.3-3-4-1）

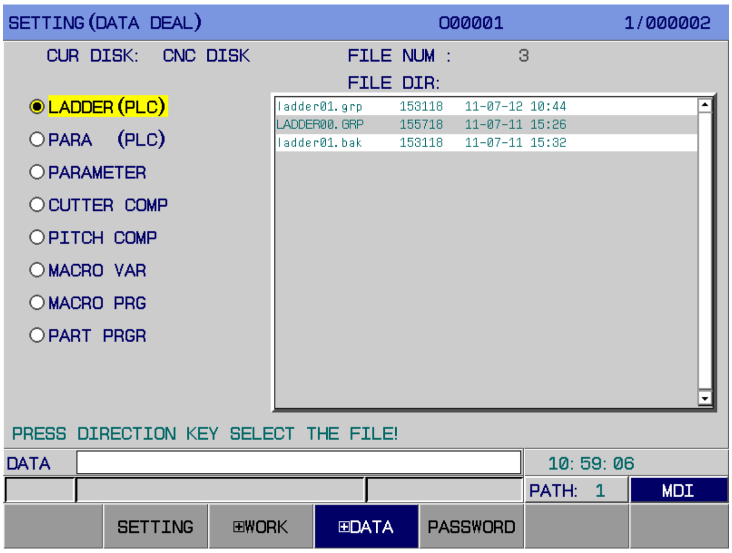


Fig. 3-3-4-1

Operation:

1. Set the password for a corresponding level in password page pressing soft key【PASSWORD】. For the corresponding password levels of the data, please see to **3.3.1.1 setting and modification for password authority** in this passage:

2. Press soft key【⊞DATA】twice to enter the DATA DEAL page, as Fig 3-3-4-2 below is shown:

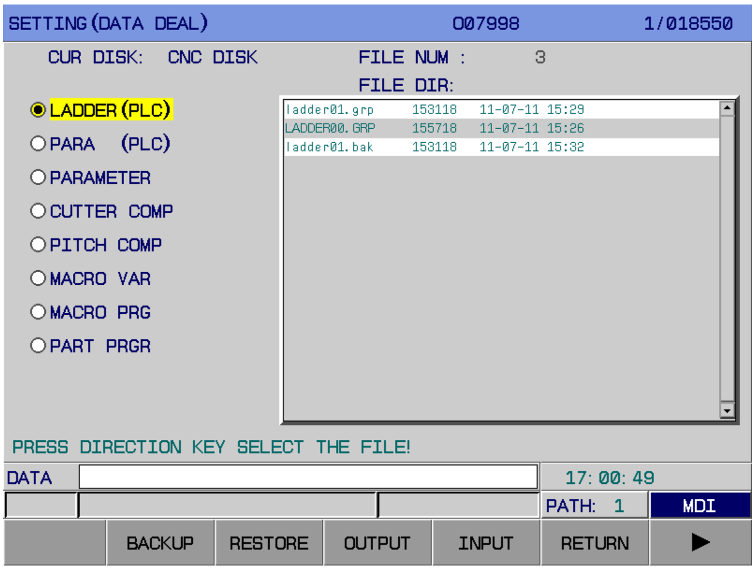


Fig. 3-3-4-2

Press【▶】to enter the next page

The functions of the operations are shown in the table below (table 3-3-4-1):

Table 3-3-4-1

Item	Function explanation
Data backup	It is available to backup the data saved in the system disk such as ladder (PLC), parameters (PLC), system parameter values, tool offset values, pitch offset values, and system macro variables separately. After the backup, the system will create a backup file with file extension .bak.
Data restoration	It is available to restore the data saved in the system disk such as ladder (PLC), parameters (PLC), system parameter values, tool offset values, pitch offset values, or system macro variables separately. The operation reads the backup file saved in the system firstly and then recovers the data.
Data output	This operation can output the data saved in the system disk to the external storage devices.
Data input	This operation can input the data saved in the external storage devices to the system disk.
One key backup (OTB)	It can backup a variety of data items to the system disk simultaneously.
One key restoration (Ghost)	It can restore the backup files of multiple data items simultaneously.
One key output	It can copy multiple data items saved in the system disk to a U disk simultaneously.
One key input	It can copy multiple data items to the system disk from a U disk simultaneously.

3. Press ↑ and ↓ to select the target file, press ← and → to switch between data item directory and file directory.

4. Press corresponding soft keys to perform operations such as backup, recovery, output, input, one key backup, one key recovery, one key output and one key input.

Note:

1) When I/O channel is set to "U Disk", the functions of soft keys Data Output and Data Input are the same.
2) When performing data output/input operation, ensure the setting for the I/O channel is correct. When using a U disk, set the I/O channel to 2; when using transmission software via PC, set the I/O channel to 0 or 1.
3) The contents of One Key Output/Input are determined by password authorities. See table 3-3-1-1 for the correspondence between data items and password authorities.
4) Relevant parameters

   Bit parameter N0:54#7: for setting whether one key output/input is valid for part programs in debugging-level authority or above.

   Bit parameter N0:27#0: for setting whether the editing for subprograms with program numbers from 80000-89999 is forbidden.

   Bit parameter N0:27#4: for setting whether the editing for subprograms with program numbers from 90000-99999 is forbidden.
5) There are concerned operation prompts in the system during data processing, the contents of which are shown as follows (table 3-3-4-2).

Table 3-3-4-2

No.	Prompt message	Cause	Handling
1	Once key operation completed	Operation succeeded	Transmission is completed
2	One key operation completed, system prompts: Copy after modifying parameters	The input/output operation of the macro program has been performed, but the parameters concerned in the system have not been set.	Skip the input/output operation of this file.
3	One key operation completed, system alarm: Parameters taking effect after power-off are modified.	The update for the ladder and ladder parameters has been executed, which requires power-on again.	Transmission is completed, please turn on the power again.
4	File reading failed	File error	Interrupt the input/output operation
5	File writing failed	File error	Interrupt the input/output operation
6	File copy failed	File error	Interrupt the input/output operation
7	Large file, please use DNC	The part program is greater than 4M	Interrupt the input/output operation
8	Insufficient storage capacity	The storage capacity is not enough.	Interrupt the input/output operation

File LADCHI**.TXT is invalid after it is transmitted to the system until the power is turned off and on again.

3.3.5 Bus Servo Parameter Display, Modification and Setting

Press【⊞Bus Allocation】to enter the bus page. The bus page is shown below (See Fig. **3-3-5-1**）:

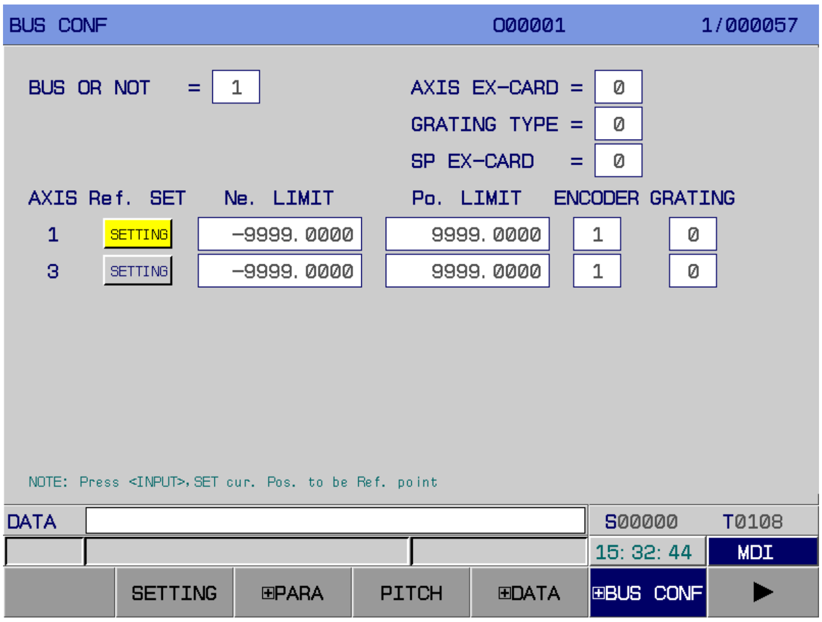


Fig. 3-3-5-1

【⊞ **bus allocation**】operation explanation:

Press 【⊞ **bus allocation**】soft key to enter the bus allocation page as Fig. 3-3-5-1 to search or modify parameters:

1. Enter <MDI> operation mode;
2. Press the move cursor UP, DOWN, LEFT, RIGHT to move it to the required change item;
3. Modify it according to the following explanation:

1) Bus

2) Encoder type

0：incremental 1：absolute

Note: No：20#6 sets to use an absolute encoder or not.

3) Select permissive max. deviation

Note: the system defaults 50.000mm，and P450~P454，P455~P459 can set a deviation.

X, Z simultaneously returns to the machine zero 0：no expansion cards 1：have expansion cards

Note: No：0#6 can set to use the bus servo card or not.

4) Grating type

0：incremental 1：absolute

Note: No：1#0 can set to use an absolute grating rule or not.

5) Spindle expansion card

0：no expansion cards 1：have expansion cards

Note: No：1#1 sets the spindle driver to be the bus control mode or not.

6) Absolute zero setting

a) Firstly, set the system terminal's gear ratio, feed axis' direction and zero return direction.

b) In MDI mode, "Bus or not" in the bus allocation page is set to 1, "Encoder Type" is set to 1, manual moving each axis sets the machine zero position.

c) Move the cursor to the soft key SET. According the prompt, press <INPUT>key twice, and the zero return indicator lights, the current position record of each axis' motor absolute encoder is taken as the machine zero. The system is turned off and then turned on, the zero return indicator is OFF. The user can manually set negative border and positive border according to the actual machine's max. stroke, which makes the current machine's absolute coordinate offsets forward or backward one value, No.61#6 is set to 1, and the positive/negative limit is valid.

Setting range：-99999.9999~99999.9999， P66~P85 sets each axis' positive/negative border.

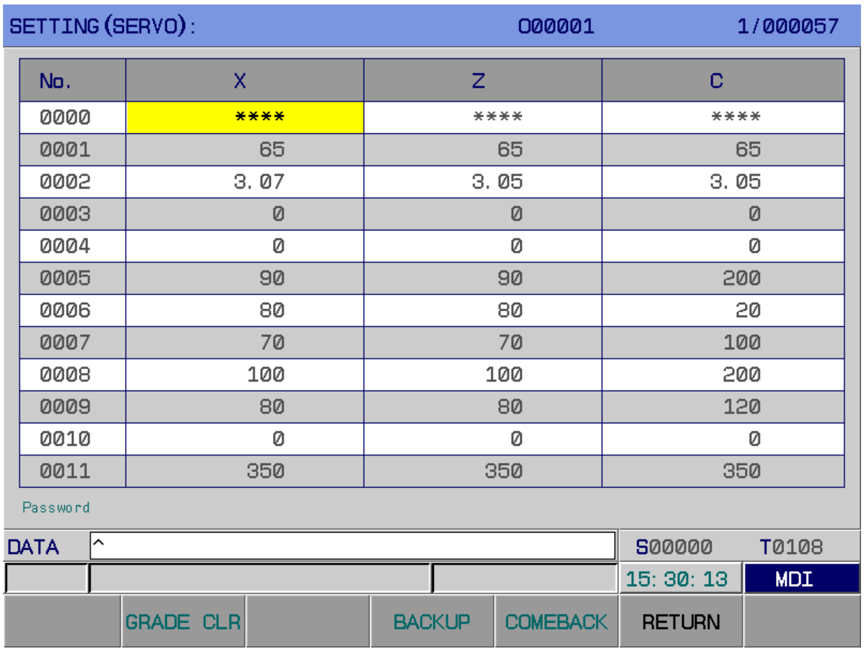
d) Whether to allocate with grating. Each axis is set to allocated with grating or not, 0：no grating，1：have grating.

Note 1：After the machine zero is set, modifying each axis' zero return direction, feed axis movement direction, servo and system gear ratio cause zero missing, the machine zero must be reset.

Note 2：After the machine zero is reset, it influences other reference points, the 2nd, 3rd reference point must be reset.

3.3.5.1 Servo Parameter Display

Press the soft key【⊞ Bus Allocation】 to enter the servo debugging page, then press【⊞ Servo Parameter】to enter the servo parameter page, which is shown below: (see Fig. **3-3-5-1-1**）.



No.	X	Z	C
0000	****	****	****
0001	65	65	65
0002	3.07	3.05	3.05
0003	0	0	0
0004	0	0	0
0005	90	90	200
0006	80	80	20
0007	70	70	100
0008	100	100	200
0009	80	80	120
0010	0	0	0
0011	350	350	350

Fig. 3-3-5-1-1

3.3.5.1.1 Servo Parameter Modification and Setting

1) Select <MDI> operation mode.

2) Press SYSTEM to enter【CNC Setting】 page, and set the parameter switch to "1".
3) Press SYSTEM, and then press【⊞ Buse Allocation】to enter the servo debugging page. Press【⊞ Servo Parameter】to enter the parameter setting and display page.
4) Move the cursor to the current selected axis parameter **#0**，input the corresponding password, press the input key to load the driver parameter to the system, and modify the servo parameter in【Servo Parameter】page.
5) Move the cursor to the required modified parameter number's position:
   Method 1：press page up/down to display page where the parameter to be set is; or press the direction key to move the cursor, and position the required modified parameter's position.
   Method 2：after the parameter number is input, SEARCH is pressed to position.
6) Press INPUT to confirm it. The parameter value is loaded into the driver, and the status bar displays " Driver parameter load success!"
7) Press SAVE to make the servo save the refreshed parameters, and the status bar displays " Driver parameter save success!"
8) After all parameters' setting completes, the parameter switch is OFF.

3.3.5.1.2 Parameter Setting of Servo Matched with Motor model

1) Select <MDI> operation mode.
2) Press SYSTEM to enter【CNC Setting】page, and set the parameter switch to "1".
3) Press SYSTEM, then press【⊞ Buse Allocation】 to enter the servo debugging page, at last press【⊞ Servo Parameter】to enter the parameter display page.
4) Move the cursor to the current selected axis parameter **#0**，input the password **385**，press the input key to load the driver parameter to the system, and modify the servo parameter i【Servo Parameter】page.
5) Move the cursor to the parameter **#1**，input a number matched with the motor model：
6) Press INPUT to confirm it. The parameter value is loaded into the driver, and the status bar displays " Driver parameter load success!"
7) Press SAVE to make the servo save the refreshed parameters, and the status bar displays " Driver parameter save success!"

8) After all parameters' setting completes, the parameter switch is OFF.

3.3.5.1.3 Servo Parameter Backup

1) Select <MDI> operation mode.

2) Press SYSTEM to enter【CNC Setting】page, and set the parameter switch to "1" .

3) Press SYSTEM to enter【CNC Setting】page, and input the **end-user password or higher level password**.

4) Press SYSTEM, then press【⊞ Buse Allocation】 to enter the servo debugging page, at last press【⊞ Servo Parameter】to enter the parameter display page.

5) Select【Backup】key to backup the current selected axis' parameter into the file DrvParXX.txt.（XX axis number. Such as: backup X axis, the file name: DrvPar01.txt）.

6) After all parameters' setting completes, the parameter switch is OFF.

3.3.5.1.4 Servo Parameter Recover

1) Select <MDI> operation mode.

2) Press SYSTEM to enter【CNC Setting】page, and set the parameter switch to "1" .

3) Press SYSTEM to enter【CNC Setting】page, and input the **end-user password or higher level password**.

4) Press SYSTEM, then press【⊞ Buse Allocation】 to enter the servo debugging page, at last press【⊞ Servo Parameter】to enter the parameter display page.

5) Select【Restore】key to restore the current selected axis' backup parameter file DrvParXX.txt into the servo drive.（XX axis number. Such as：backup X axis, the file name:DrvPar01.txt）.

6) Press SAVE to make the servo save the refreshed parameters, and the status bar displays " Driver parameter save success!"

7) After all parameters' setting completes, the parameter switch is OFF.

3.3.5.1.5 Servo Grade Zero

When the parameters are debugged, the servo parameter's rigid is too big to cause the machine shake. To avoid a danger, the servo grade zero function can rapidly restore the servo parameter into 0 grade initial state.

1) Select <MDI> operation mode.

2) Press SYSTEM to enter【CNC Setting】page, and set the parameter switch to “1” .

3）Press SYSTEM to enter【CNC Setting】page, and input the **end-user password or higher level password**.

4) Press SYSTEM, then press【⊞ Buse Allocation】 to enter the servo debugging page, at last press【⊞ Servo Parameter】to enter the parameter display page.

5) Select【Grade Zero】to restore all servo axis parameters into 0 grade parameters.

6) Press SAVE to make the servo save the refreshed parameters, and the status bar displays “ Driver parameter save success!”

7) After all parameters’ setting completes, the parameter switch is OFF.

3.3.5.2 Servo Debugging

To ensure the servo debugging function really responding the servo performances, cancel the drive side’s gear ratio and the system side’s all compensations (including pitch error compensation and backlash compensation).

3.3.5.2.1 Page Constitution

Press【⊞ Servo Debugging】 to enter the servo debugging tool page. The page is shown below（See Fig. 3-3-5-2-1-1~Fig.3-3-5-2-2）.

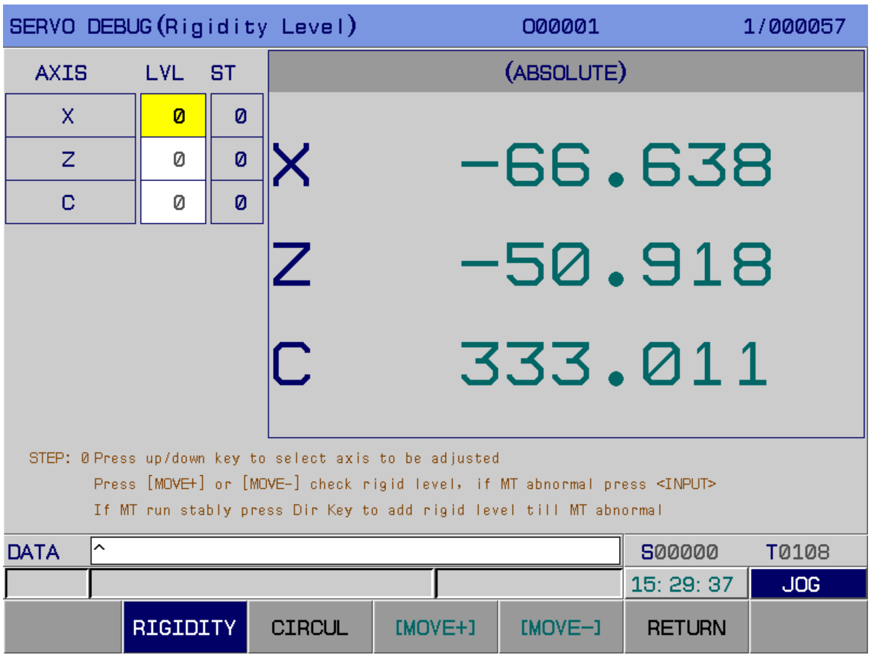


Fig. 3-3-5-2-1 rigid grade page

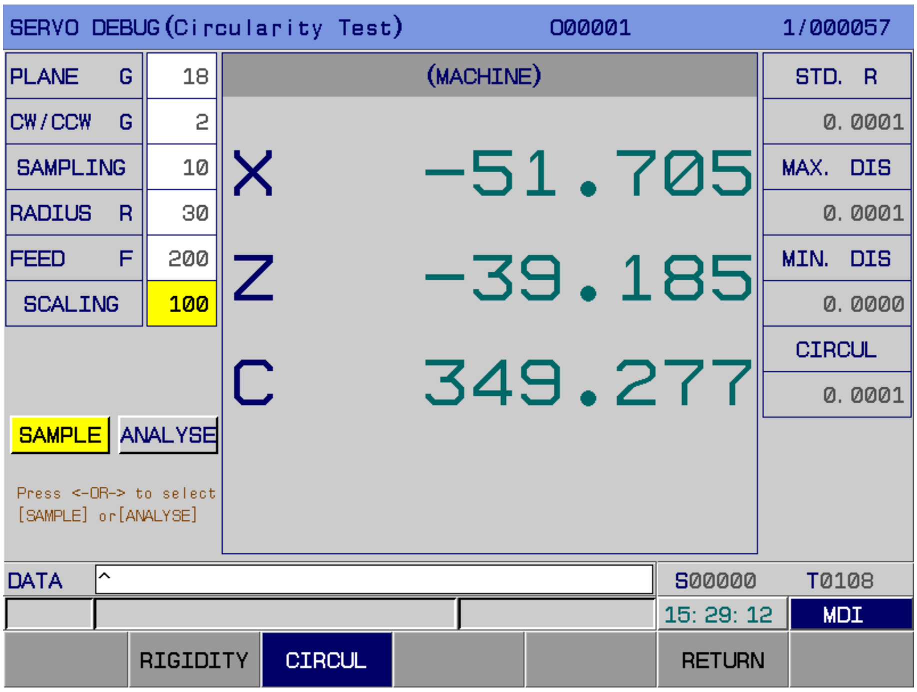


Fig. 3-3-5-2-2 Circular degree test page

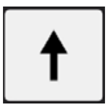
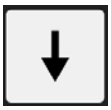
Note: The coordinate axis display in the servo debugging page is determined by the system's controlled axis quantity and the smaller number in the bus servo's salve station.

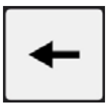
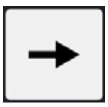
3.3.5.2.2 Function Introduction

1. Rigid grade and parameter optimization operation function
   The function is to set the servo parameter to the servo performance's optimal state.
2. Circular degree
   The circular degree tests can simulate a circumference cutting movement circle, and hereby collect the motor mask position message to judge each servo axis response's synchronism of the machine tool.

3.3.5.2.3 Operation Explanation

1. **Rigid grade debugging operation**
   Explanation: debug and set the rigid grade. The system can perform one axis one time.
   Operation key:

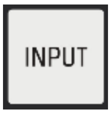
A.  and  key: selection axis. (note: once the system enters the optimizing process, the user cannot use the UP, DOWN direction key to change the current operating axis.)

B.  and  key: reduce or increase the current axis' rigid grade, press every time the rigid grade to reduce or increase one grade;

C. **【Axis move+】** and **【axis move-】**soft key: positively or negatively move at the speed set by P393 the currect axis to some distance set by P392. Before entering the optimizing processing, repetitively press【axis move+】 and 【axis move-】soft key movement axis to view whether the motor vibrates or is abnormal. Must not continuously press 【axis move+】 and 【axis move-】 to get the motor characteristic data after entering the optimizing process.

Note 1：After entering the optimizing process, press【axis move+】and【axis move-】soft key movement axis and sample data.

Note 2：The non-specialized person must not change P392 and P393，otherwise, it maybe cause optimization unsuccessfully.

D. 

key: confirm the operation or enter the next step;

E. 

key: cancel some operation or return to the previous step;

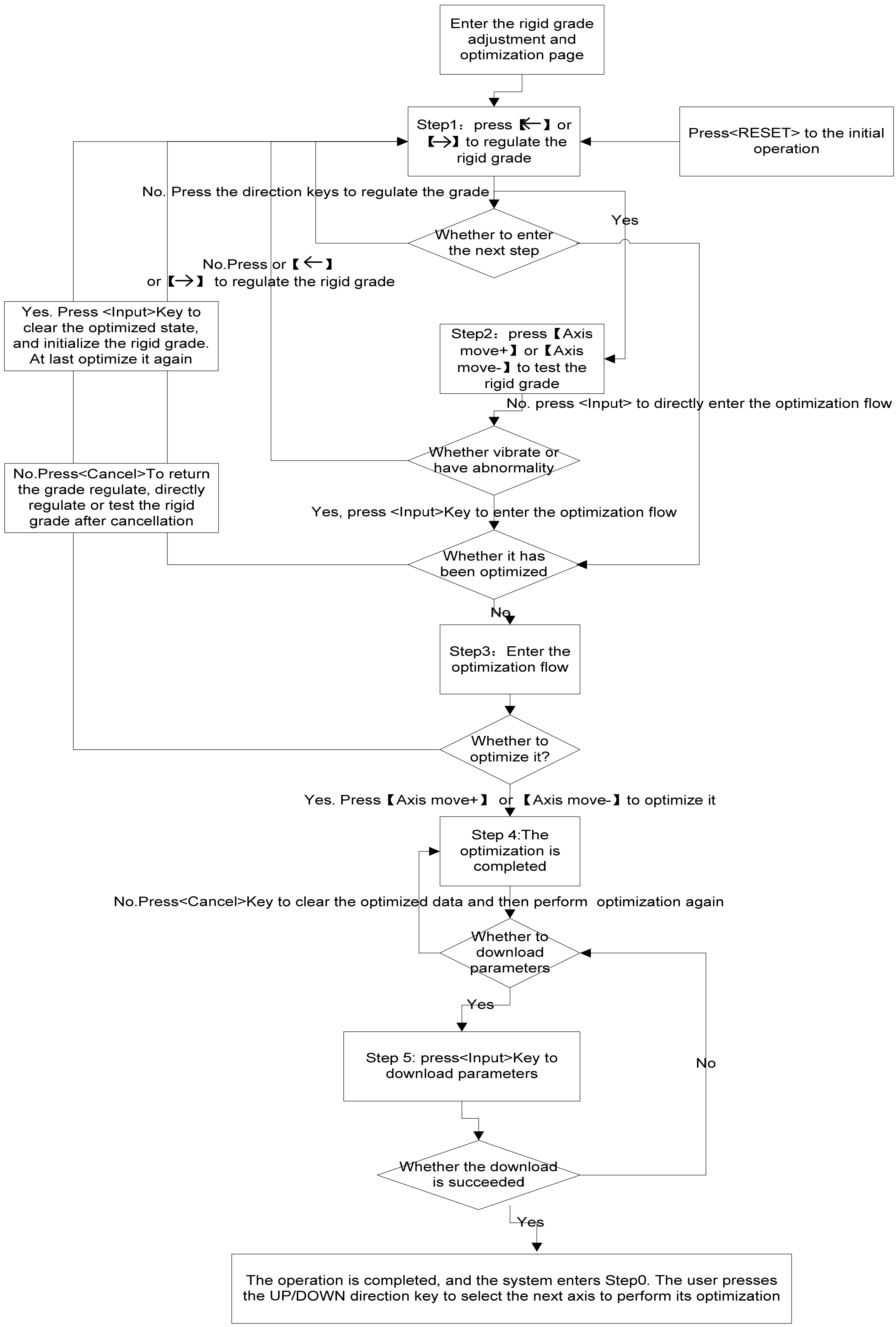
F. 

key: reset the operation and return to the initial operation step;

G. 

key: save operations.

Operation process is shown below:



II Operation

2. Circular test

Operation key：

A. Number key：input various parameter values;

B. ↑ and ↓ key：select a parameter item;

C. ← and → key：select functions(collect and analysis)；

D. INPUT key：input parameter values or confirm and execute operations;

E. INPUT key：clear data and reset to the initial state.

Parameter items:

A. plane：select a test plane G18;

B. CW/CCW circle： select the circle direction G02, G03;

C. Sampling period：set the sampling period, which is set by the circle radius and feedrate. The bigger the radius is, the longer the sampling period is; the slower the feedrate is, the longer the sampling period;

D. feedrate：move speed when test;

E. magnification times：the circular degree analysis is a multiple of error magnifying.

Operation steps:

Step 1: After setting various parameters, press ← or → to select the sampling function;

Step 2： Press INPUT to start the circular movement and start sampling data. After sampling, press ← or → to select the analysis function.

Step 3: Press INPUT to the analysis function and output the circular degree data and draw circular error distribution diagram, which is shown below:

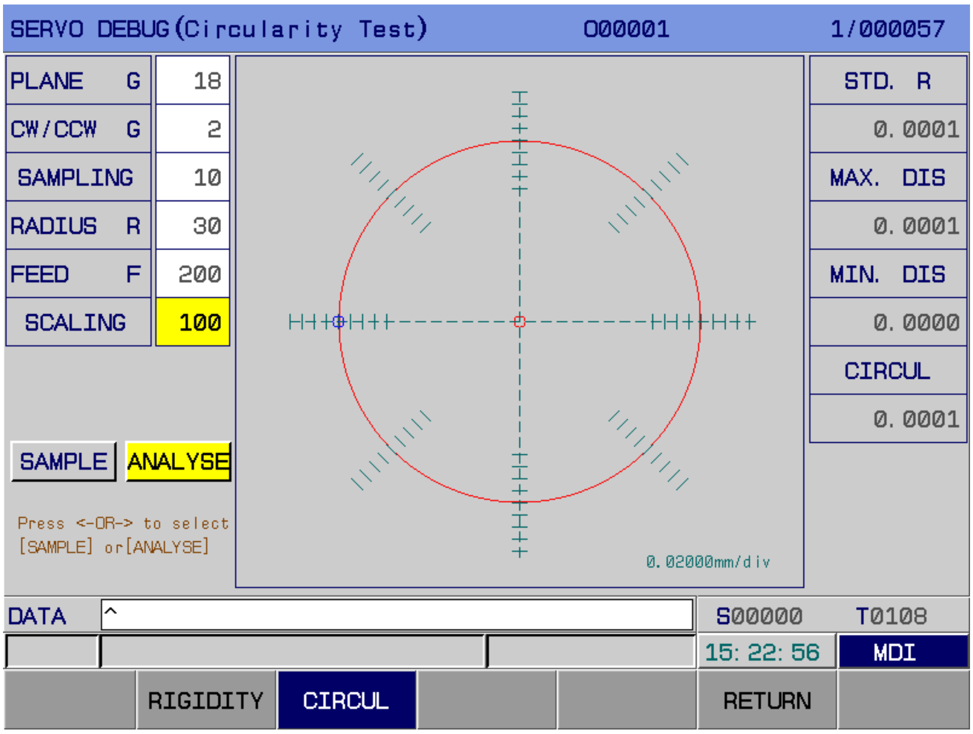
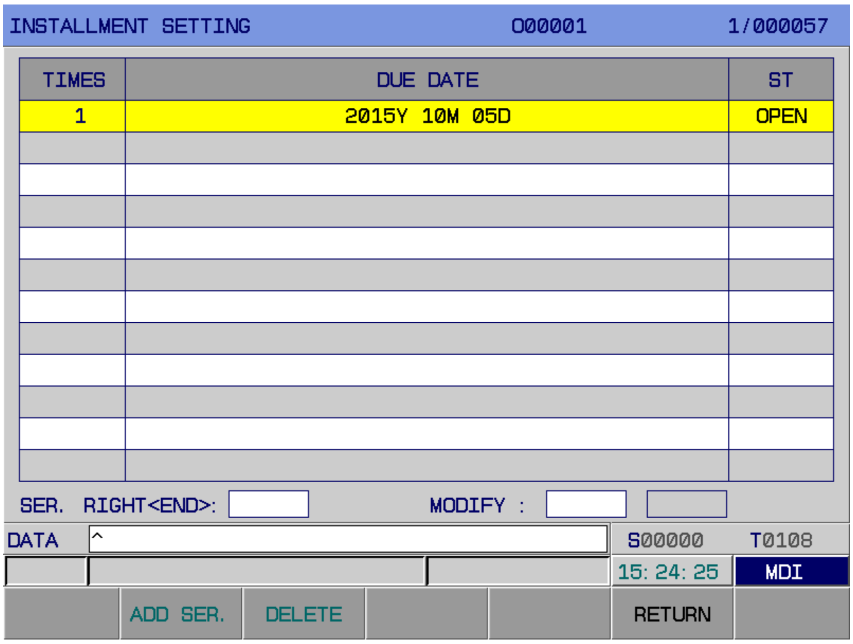


Fig. 3-3-5-2-3-1

Note: After debugging the rigid grad and parameter optimizing function, use the circular degree test function to test the current feed axis' synchronism. Each plane's circular degree test within 6u is taken as a better synchronism of current each feed axis, and the parameter debugging basically succeeds.

3.3.6 Time Limit Stop Display

Press【▶】, and then press【Time limit Stop】to enter the time limit stop display page which is divided into staging authority and password modification function. A new password can be modified by the staging authority, which is shown below (see Fi.g3-4-6-1):



3-3-6-1

3.3.6.1 Staging Authority Setting and Modification

1）After entering the page in <MDI mode>, move the cursor to position to the target position.

2)After getting the machine manufacturer's password, press END to input the password, then press INPUT to confirm the password. If the password is correct, the system prompts "Password correctness".（the password is vacancy when delivery, and the user directly input it.）

3）When the authority password is modified, 0~6 digits number or letters are input, and INPUT is pressed to confirm the input.

4）The system prompts "Please input a new password again".

5） After inputting the new password, press INPUT. When the twice inputs are the same, the system prompts "the password modified, please save the new". The operation password is modified successfully.

Note: When the twice inputs are different, the system prompts "the new password and the confirmation password are inconsistent, this moment, please input the new password".

Press 【Time limit stop】again to enter the time limit stop page which includes【Newly increased periods】,【Delete periods】function, and the concrete contents are shown below: (see Fig. **3-4-6-2**)：

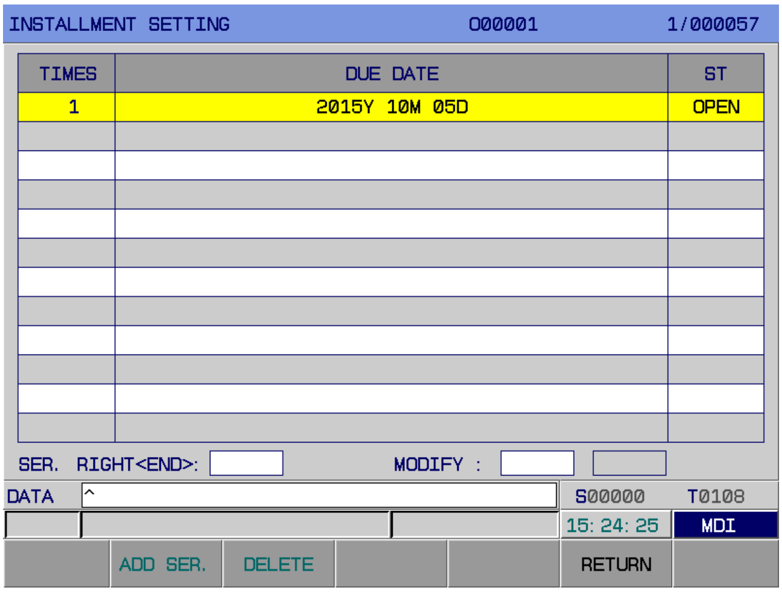


Fig. **3-3-6-2**

In the figure, press 【Newly increased periods】page to enter the operation page.(See Fig. **3-4-6-3**）

II Operation

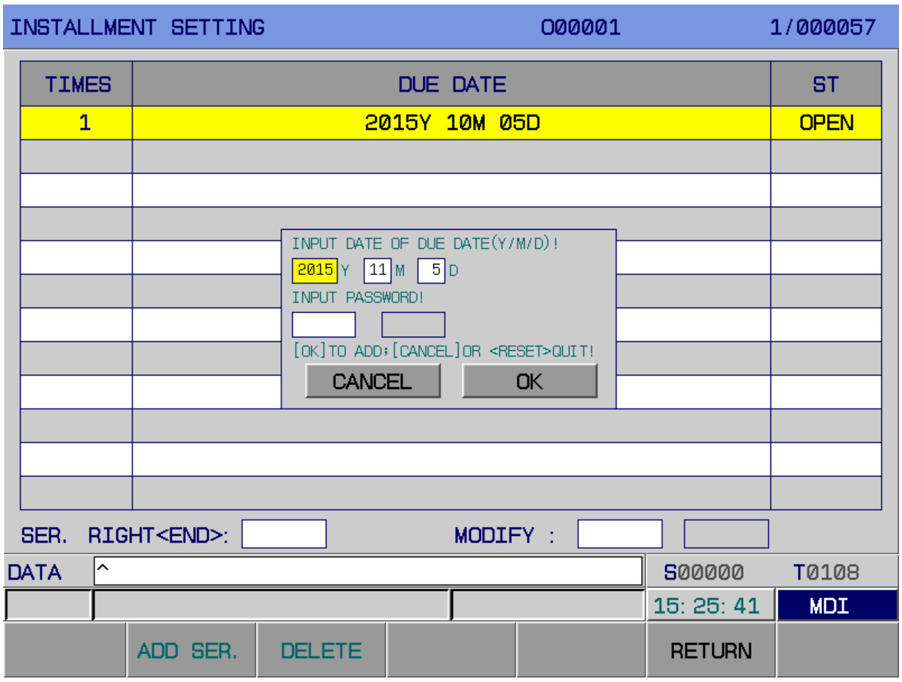
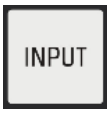


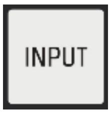
Fig. **3-3-6-3**

Concrete operations:

1） After entering the page in <MDI mode>, move the cursor to position to the target position.

2） After getting the machine manufacturer's authority, set the stop data and password. Set up to12 periods. Set the password: such as password 1, password 2 …….

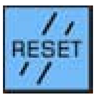
3） Input the password, press 

to confirm the input, and the system prompts "Input the new password again". When the input is correct, the cursor automatically jumps to "Confirmation", press 

to set the newly increased periods successfully. Press 

not to newly increase the periods.

4） In【Newly increased】, press 

or 

to exit.

Note: When the twice inputs are different, the system prompts "the new password and the confirmation password are inconsistent, this moment, please input the new password".

【Delete periods】 page display:

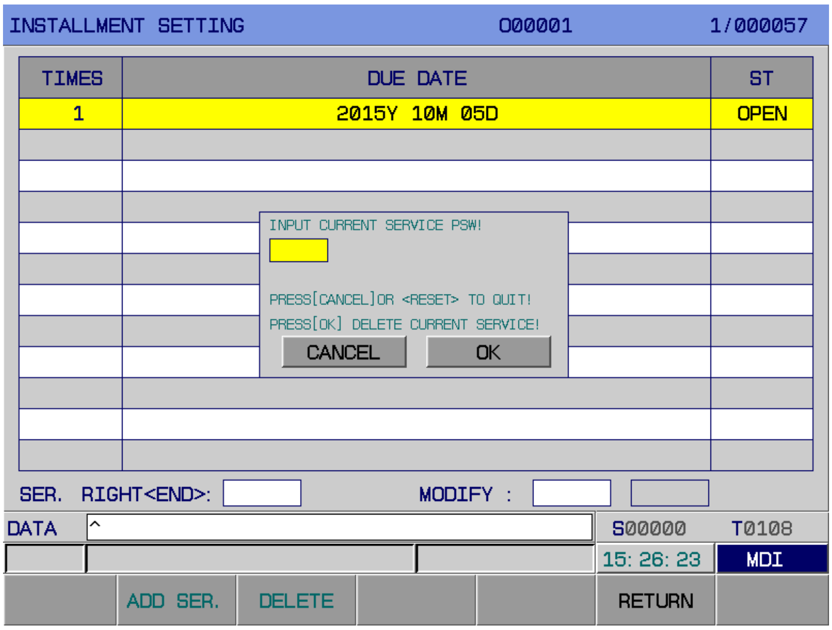
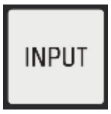
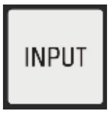


Fig. **3-3-6-4**

Operation steps:

1） After entering the page in <MDI mode>, move the cursor to position to the required deleted period position.

2） Press “Delete periods” and pop-up a dialog of time limit stop period.

3） Input the stop limit password, press  to confirm the input. When the input is correct, the cursor automatically jumps to “Confirmation”, press  to delete the selected time limit stop periods. When the password is incorrect, the deletion cannot be done successfully.

4） In【Delete periods】, press  or  to exit.

3.4 Tool Offset Display

Press  to enter the offset message display page which is divided into three subpages【⊞Offset】, 【⊞Workpiece coordinates】, 【⊞Macro variable】, press the corresponding soft key to view or modify messages. The concrete content is shown below: (See Fig. **3-4-1**)：

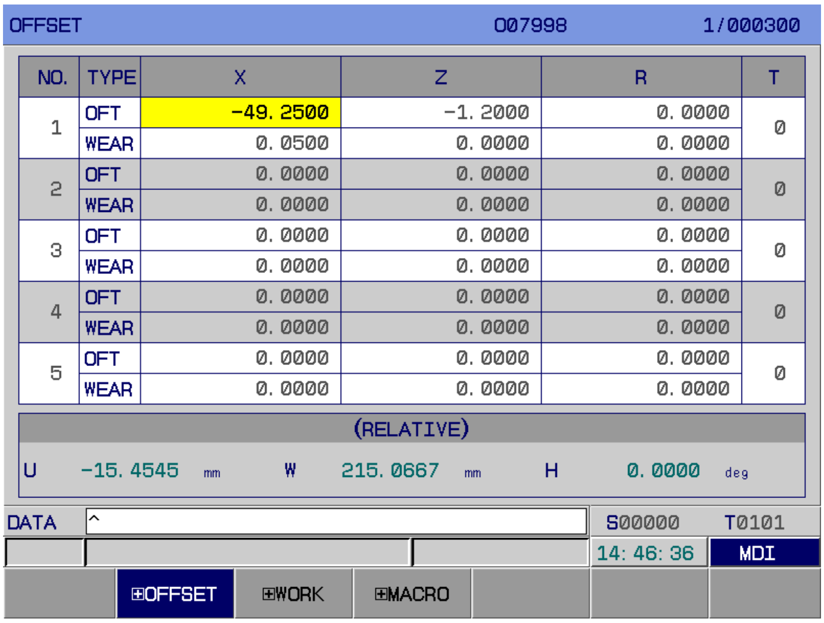


Fig. **3-4-1**

3.4.1 Offset Display, Modification, Setting

Press【⊞Offset】to enter the offset display page as follows (See Fig. **3-4-1-1-1**）:

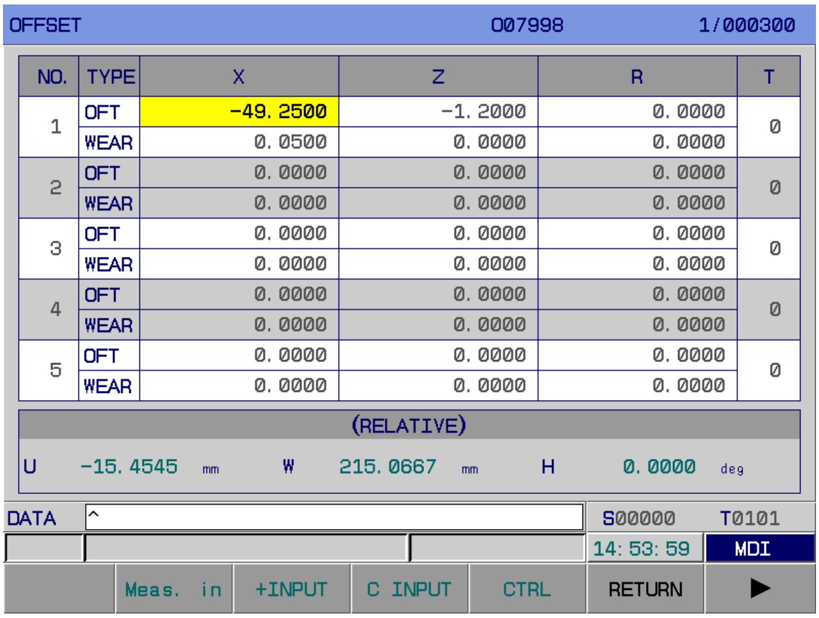


Fig. 3-4-1

Press ▶ to enter the next page

to directly input the compensation amount or add/subtract the current position's value.

3.4.1.1 Offset Value's Setting

Set the offset in the offset page:

1) Press 【⊞Offset】to enter the offset display page.

2) Move the cursor to the required input compensation number's position.

Method 1：press page up/down to display the page where the required modified compensation amount is; press the direction key to move the cursor to the required modified compensation number's position.

Method 2：after inputting the compensation number, press SEARCH to position.

3) Input digital value.

A. Offset line:

（1） Absolute axis: press the address keys（X Y, Z V corresponding axis name's key）, input the digital value, press INPUT on the panel or F1【Measure input】soft key（displayed value=absolute coordinates －input value）; the system automatically clears the current wear value after input.

（2）Relative axis: press the address keys（U A, W B corresponding axis name's key）, input a digital value, press INPUT on the panel or F2【+input】soft key（displayed value=current numerical value +input value）;

（3）Absolute axis: axis: press the address keys（X Y, Z V corresponding axis name's key）, press F3【C input】soft key（displayed value=relative value – wear value）;

（4）Press R on the panel, input a digital value，press <Input>key.（displayed value=input value）;

（5）Press T K on the panel, input a digital value（0～9 integer）, press INPUT key on the panel. （displayed value=input value）

B. Wear line：

Relative axis：press U A, W B corresponding axis name's key）, input a digital value, press INPUT on the panel or F2【+input】soft key（displayed value=current numerical value +input value）

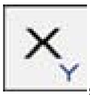
Note 1：Taking example of T：only input T K on the panel, other letters are invalid.

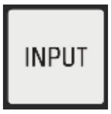
Note 2：All operations are mistaken for formats except for the above operations in the offset page.

Note 3：Data color becomes after offset modification, the cursor restores after it is moved or the page is switched.

3.4.1.2 Offset Value Modification

1）Move the cursor to the required changed tool offset number's position according to the method described in Section 3.4.2.1.

2）Require to change X tool offset value, input U. For Z, input W; or select the required changed data position, press the address key（， and other corresponding axis name's key_ to input the digital value.

3）Press  on the panel or F2【+input】. Add the current tool offset value to the input incremental value. The result is taken as the new tool offset value to display.

Example: the set X tool offset value is 5.678

Input the increment U1.5 by keyboard

The new set X tool offset value is 7.178（=5.678+1.5）

Note 1：When the tool offset is changed, the new offset amount does not take effect immediately. It becomes valid till its compensation number T code is executed.

Note 2：The user can modify the tool compensation value during the program running, but the modification is done before the tool compensation number runs.

3.4.2 Workpiece Coordinate Setting Page

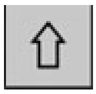
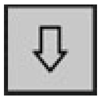
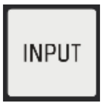
Press【⊞WORK】 to enter the coordinate system setting page, and the displayed content is shown below (See Fig. **3-4-2-1**）.

Fig. 3-4-2-1

Another 50 additional workpiece coordinate systems can be used besides the 6 standard workpiece coordinate systems (G54～G59 coordinate systems), as is shown in Fig. 3-4-2-2. Each coordinate system can be viewed or modified by page keys. See section 4.2.7 Additional workpiece coordinate system in PROGRAMMING for details about its operation.

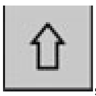
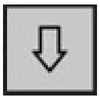
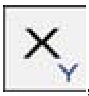
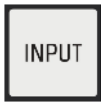
Fig. 3-4-2-2

3.4.2.1　Directly Input (Digit)

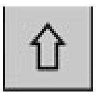
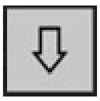
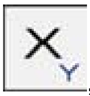
After entering the workpiece coordinate system page, press  ,  to move the cursor to the input position, press the input digital value, press 

to directly input the digit. (as Fig. 3-4-3-2)

3.4.2.2　Two Methods to Input Coordinates:

Method 1: press ,  to move the cursor the input position, press the address key (,  , or other corresponding axis name keys), input digits; press 

and the CNC automatically completes count and input the result data;

Method 2: press  ,  to move the cursor the input position, press the address key (,  , or other corresponding axis name keys), input digits; press the soft key 

and the CNC automatically completes count and input the result data;

Method to search a coordinate system:

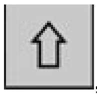
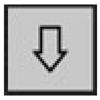
1) In any mode, press key 

to search after inputting a coordinate system, e.g. inputting "G56".

2) In any mode, by inputting "P6" or "P06" and then pressing key 

, the cursor will be located in the additional workpiece coordinate system "G54 P06".

3.4.2.3 A Workpiece Coordinate System Modification

After the system entering the workpiece coordinate system page, press ,  to the required data position to modify, input a digit, press F3【+Input】, add the current workpiece coordinate system's offset value to the input value's increment, and its result is taken as a new coordinate system's offset value.(Value=current value + input value) as Fig. 3-4-3-3-1.

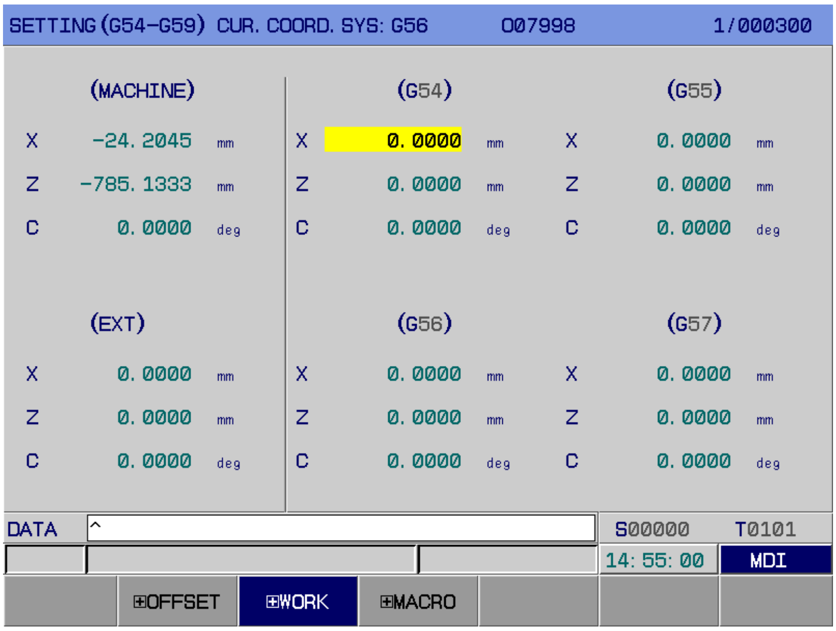
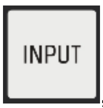


Fig.3-4-2-3-1

3.4.2.4 Clearing a Workpiece Coordinate System

After the system enters the workpiece coordinate system setting page, the user presses 

to clear all workpiece coordinate system's offset values, and then the system prompts to require to press 

, and the deletion is completed. When the deletion is cancelled, 。 <Cancel>key is pressed to cancel the deletion; the user can use the direct input method in this chapter to set any coordinate system's axis offset value to 0.

3.4.3 Macro Variables Display, Modification and Setting

3.4.3.1 Macro variable display

Press soft key【⊞MACRO】 to enter macro variable page, which consists of two subpages:【CUSTOM】 and 【SYSTEM】. Both of them are available to be viewed and modified by corresponding soft keys, which is shown below:

1） **Customer variable page** Press soft key【CUSTOMER】to enter this page.（See **3-4-3-1-1**）:

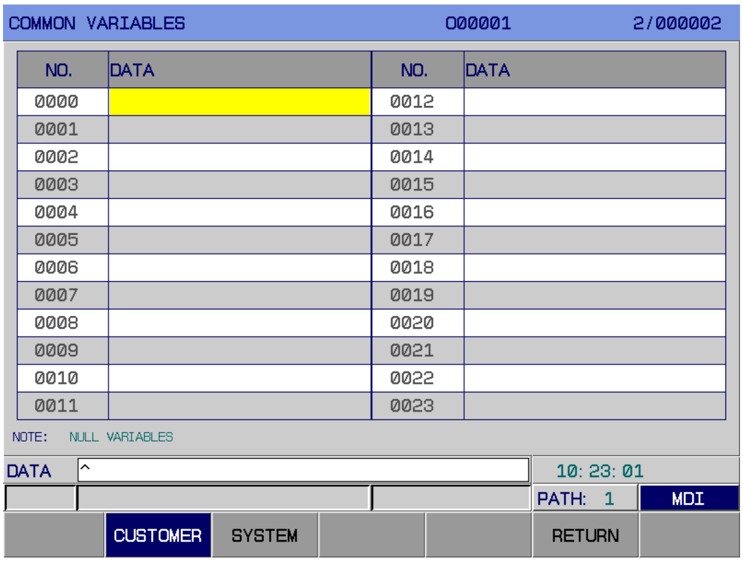


Fig. 3-4-3-1-1

2） **System variable page** Press soft key【SYSTEM】to enter this page（See Fig.**3-4-3-1-2**）:

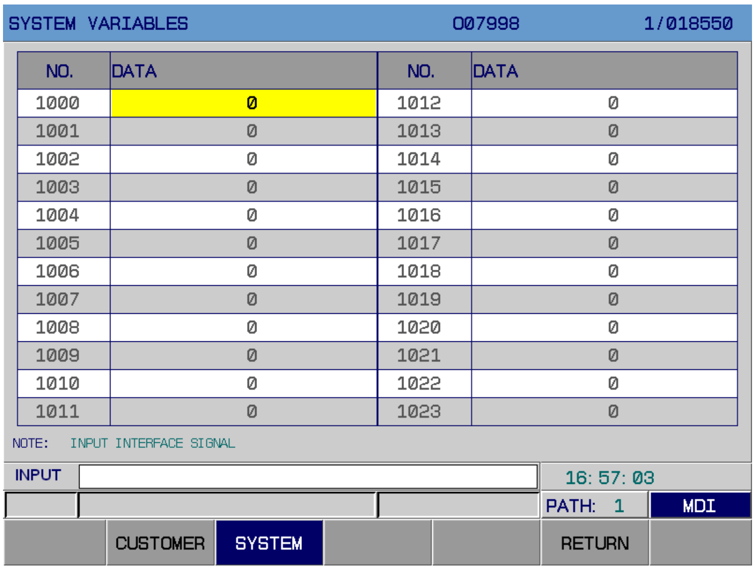


Fig. 3-4-3-1-2

Refer to Section 4.7.2 in PROGRAMMING for the explanation and use of macro variables.

3.4.3.2 Macro Variables Modification and Setting

1) Select <MDI> mode.

2) Press key 

, then soft key【⊞MACRO】to enter macro variable page.

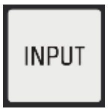
3) Move the cursor to the variable number to be modified.

Method 1: Press page keys to display the page where the variable is to be modified; move the cursor to the variable to be modified.

Method 2: Press key 

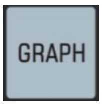
to search after inputting the variable number.

4) Input a new value using number keys.

5) Press key 

for confirmation, and then the value will be input and displayed.

3.5 Graphic Display

Press key 

to enter the graphic page which consists of two subpages:【G. PARA】and【⊞GRAPH】. They can be switched between each other by corresponding soft keys. (See Fig.3-5-1)

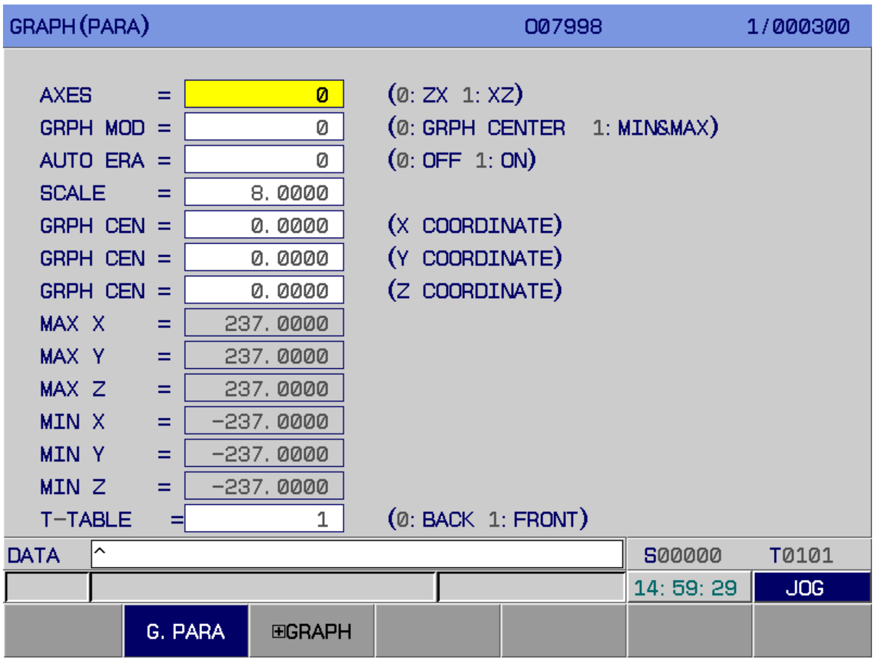


Fig. 3-5-1

1） Graphic parameter page: Press soft key【G. PARA】 to enter this page, see Fig.3-5-1.

A. Graphic parameter meaning

AXIS: set drawing plane, with 2 selection modes (0-1), shown in the 2nd line.
Graphic mode: set graphic display mode
Automatic erasion: When it is set to 1, the program graphic is erased automatically at next

cycle start-up after the program is finished.

Scale: set drawing ratio

Graphic center: set the coordinates corresponding to the LCD center in workpiece coordinate system

The maximum and minimum value: The scaling and the graphic center are automatically set when the maximum and minimum value of the axis are set.

Maximum value of X axis: the maximum value along X axis in graphics (Unit: 0.0001mm / 0.0001inch)

Minimum value of X axis: the minimum value along X axis in graphics (Unit: 0.0001mm / 0.0001inch)

Maximum value of Y axis: the maximum value along Y axis in graphics (unit: 0.0001mm / 0.0001inch)

Minimum value of Y axis: the minimum value along Y axis in graphics (Unit: 0.0001mm / 0.0001inch)

Maximum value of Z axis: the maximum value along Z axis in graphics (Unit: 0.0001mm / 0.0001inch)

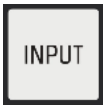
Minimum value of Z axis: the minimum value along Z axis in graphics (Unit: 0.0001mm / 0.0001inch)

Tool post selection: modifying the parameter according to the tool post type is for the sake of the graph visually reflecting the machine actuality. There are two kinds of selection mode (0-1), which is shown as the 2nd line.

B. Setting steps for graphic parameters:

a. Move the cursor to the parameter to be set;

b. Key in the value required;

c. Press key  to confirm it.

2）Graphic page Press soft key【⊞GRAPH】to enter this page（Fig. **3-5-2**）:

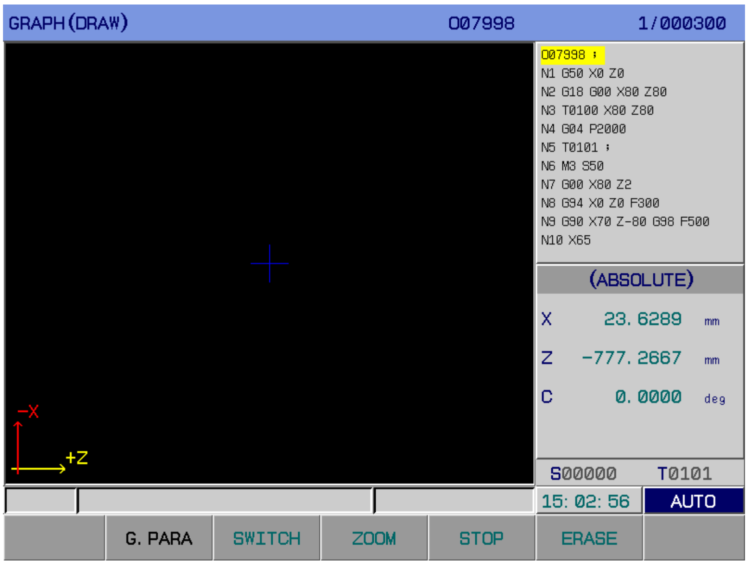


Fig. 3-5-2

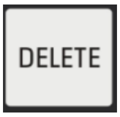
The machining path of the program being executed can be monitored in graphic page.

A Press soft key【START】or key  to enter the DRAW START mode, then sign "*" is placed in front of "S: START";

II Operation

B Press【STOP】soft key or key  to enter the DRAW STOP mode, then sign'*'is moved ahead of "T: STOP";

C Press soft key 【SWITCH】 to switch the graph display among coordinates corresponding to **0**～1;

D Press soft key【ERASE】or key  to erase the graph drawn.

3.6 Diagnosis Display

The state of DI/DO signals between CNC and machine, the signals transferred between CNC and PLC, PLC internal data and CNC internal state etc. are displayed in the diagnosis page. Refer to "GSK980TC3 CNC System PLC, Installation and Connection" for the meaning and setting of each diagnosis number.

The diagnosis of this part is used to detect the running states of the CNC interface signals and internal signals rather than modifying the states.

Press key  to enter the Diagnose page, which consists of 5 subpages: 【⊞ SIGNAL】,【SYSTEM】, 【BUS】, 【DSP】 and【⊞WAVE】. All of them can also be viewed by pressing the soft keys（See Fig. **3-6-1**）.

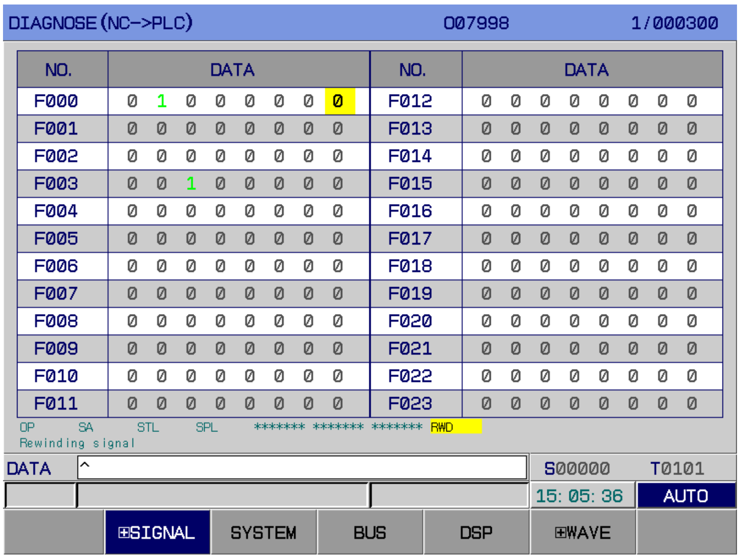


Fig. 3-6-1

II Operation

3.6.1 Diagnosis Data Display

3.6.1.1 Signal Parameter Display

Press【SIGNAL】to enter the signal diagnosis page. The page display content is shown below:（See Fig. **3-6-1-1-1~**Fig. **3-6-1-1-4**）。

1. F signal page Press soft key 【F SIGNAL】in <DIAGNOSIS> page to enter diagnosis (NC→PLC) page. See Fig. 3-6-1-1.

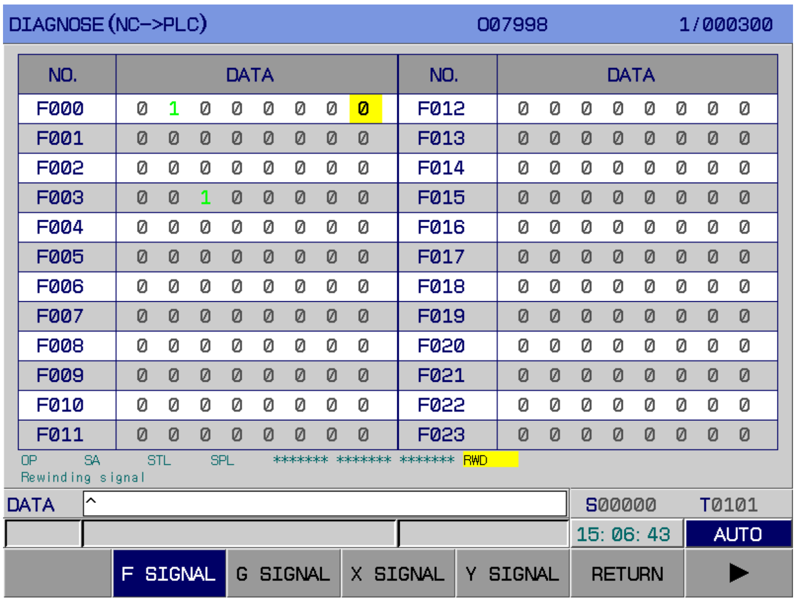


Fig. 3-6-1-1-1

This is the signal sent to PLC by CNC system. See "GSK980TC3 CNC System PLC, Installation and Connection User Manual" for the meaning and setting of each diagnosis number.

2. G signal page In <DIAGNOSE> page, press soft key【G SIGNAL】 to enter diagnosis (PMC—>CNC) page, which is shown in Fig. 3-6-1-2:

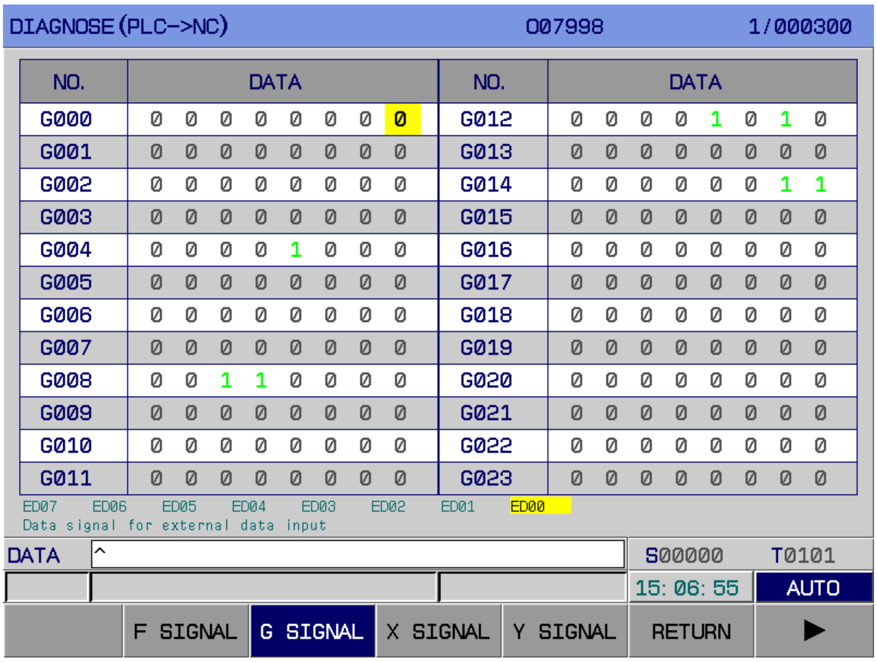


Fig. 3-6-1-1-2

This is the signal sent to CNC system by PLC. See “GSK980TC3 CNC System PLC, Installation and Connection User Manual” for the meaning and setting of each diagnosis number.

II Operation

3. X signal page Press soft key【X SIGNAL】in <DIAGNOSIS> page to enter diagnosis (MT→PLC) page, which is shown in Fig. **3-6-1-3:**

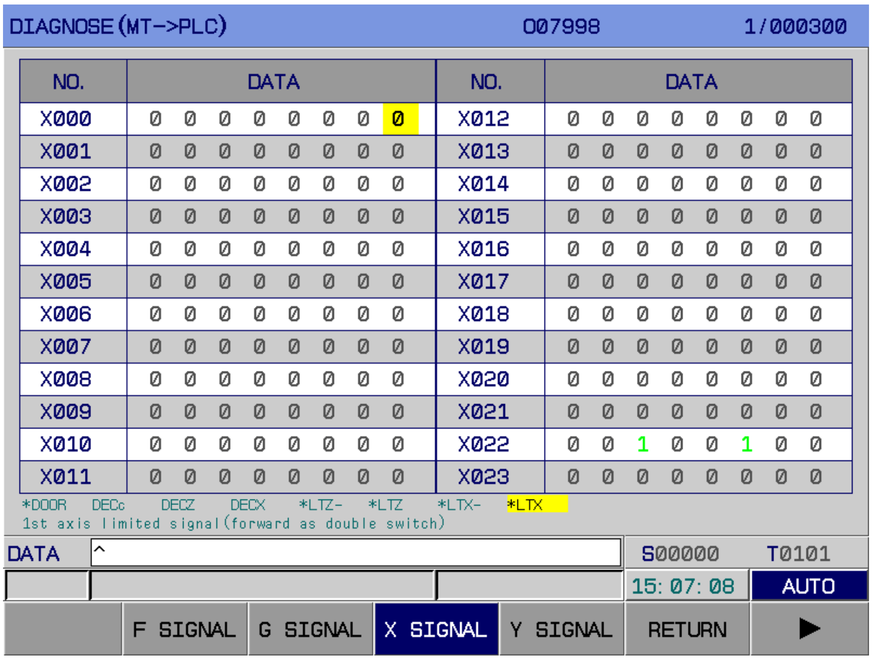


Fig. 3-6-1-1-3

This is the signal sent to PLC by CNC system. See “GSK980TC3 CNC System PLC, Installation and Connection User Manual” for the meaning and setting of each diagnosis number.

4. Y signal page Press soft key【Y SIGNAL】in <DIAGNOSIS> page to enter （PLC→MT）page, which is shown in Fig. 3-6-1-4:

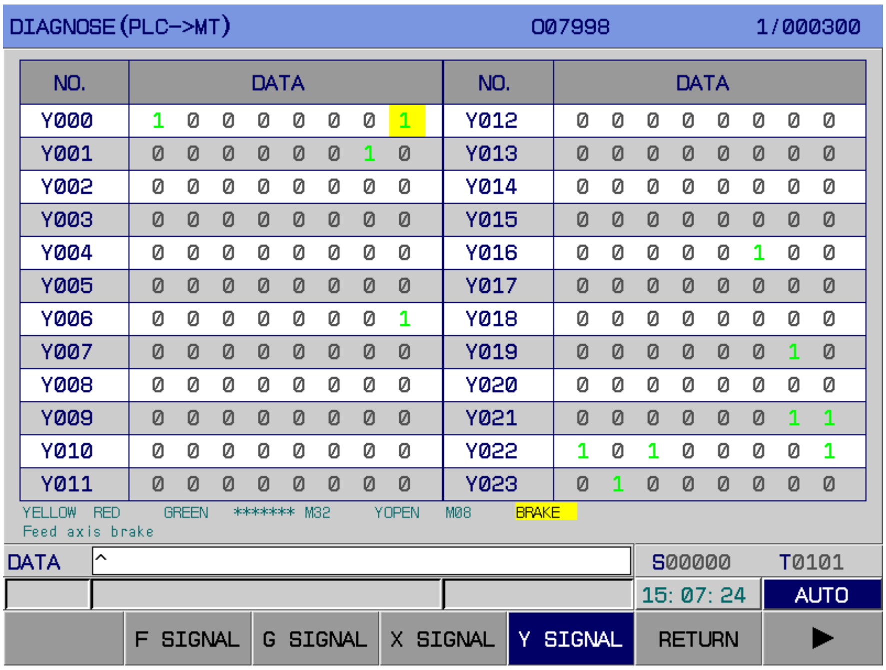


Fig. 3-6-1-1-4

This is the signal sent to PLC by CNC system. See “GSK980TC3 CNC System PLC, Installation and Connection User Manual” for the meaning and setting of each diagnosis number.

3.6.1.2 System Parameter Display

Press soft key 【SYSTEM】 to enter the system signal diagnosis page, which is shown in Fig. 3-6-1-2-1:

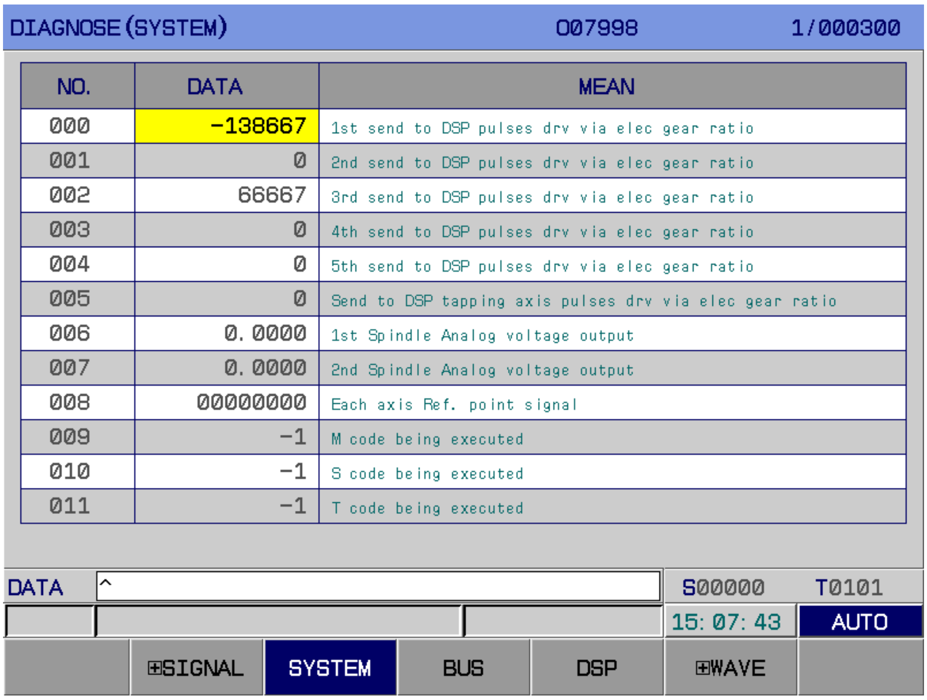


Fig. 3-6-1-2-1

3.6.1.3 Bus Parameter Display

Press soft key 【BUS】 to enter the bus signal diagnosis page, which is shown in Fig. 3-6-1-3-1:

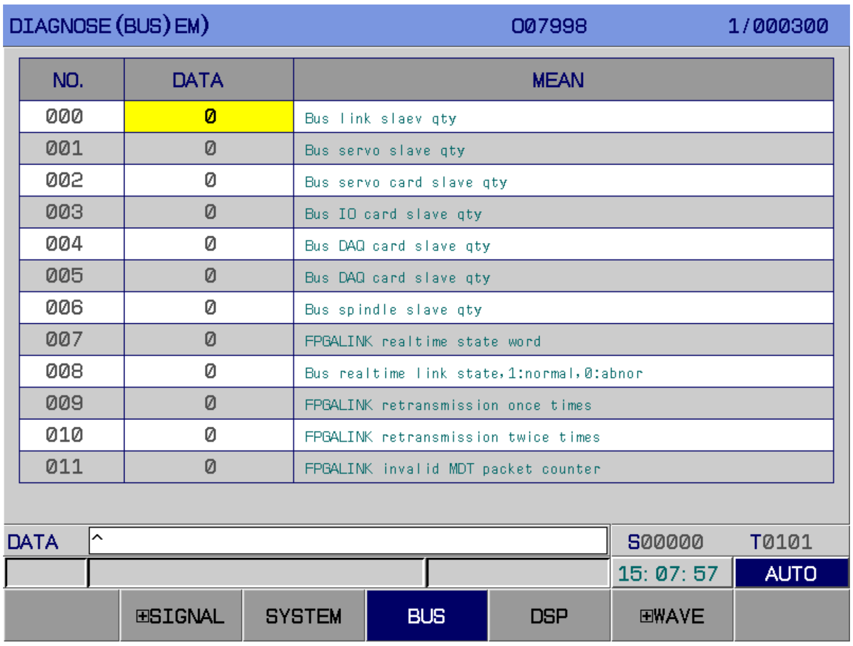


Fig. 3-6-1-3-1

3.6.1.4 DSP Parameter Display

Press soft key【DSP】to enter the system signal diagnosis page, which is shown in Fig. 3-6-1-4-1:

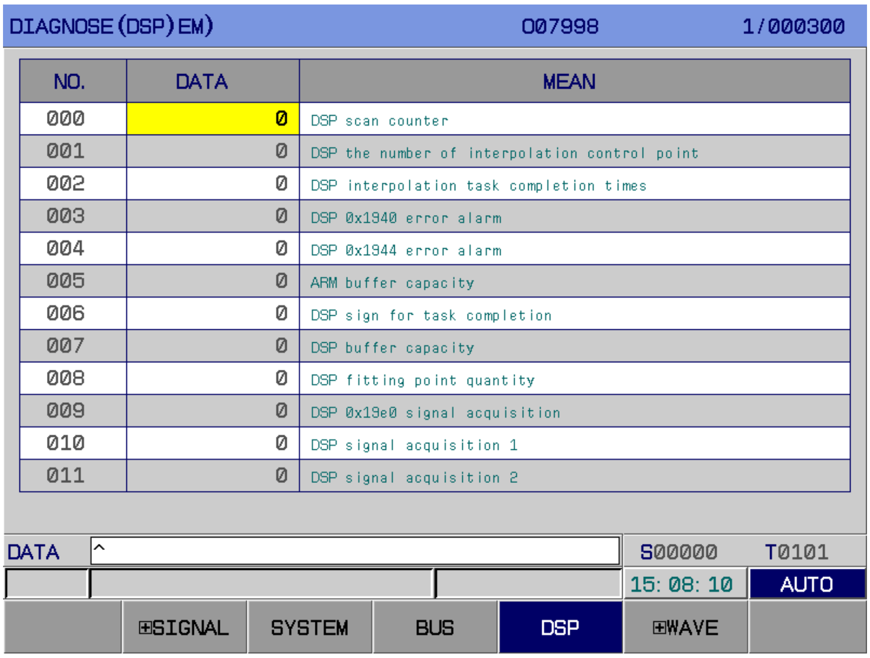


Fig. 3-6-1-4-1

3.6.1.5 Wave Parameter Display

Press soft key【WAVE】to enter the wave page, which is shown in Fig. 3-6-1-5-1.

II Operation

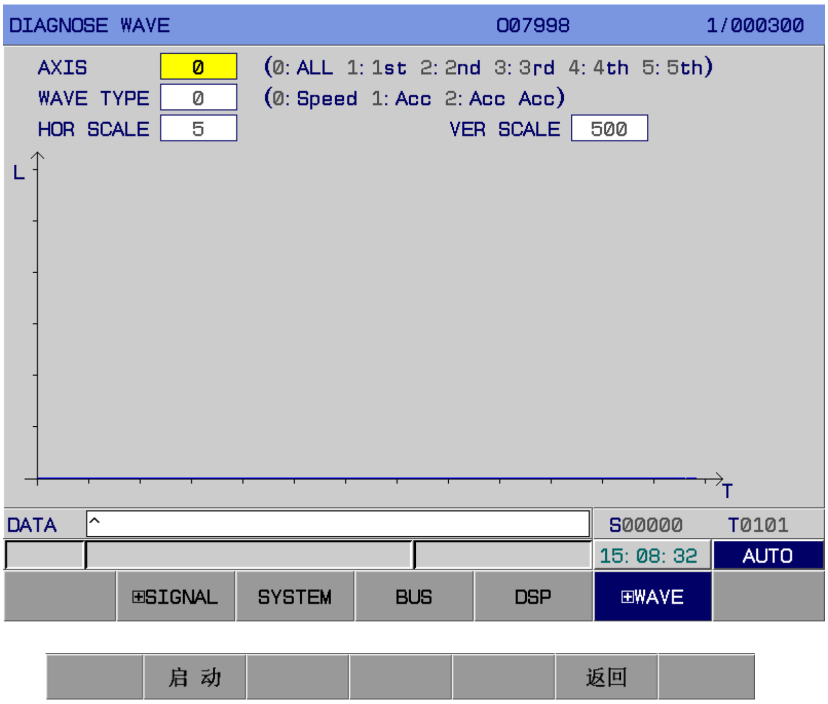
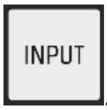


Fig. 3-6-1-5-1

AXIS: select the axis for WAVE diagnosis.

WAVE: select the waveform type.

HOR SCALE: select the graph ratio.

Data: In any mode, input corresponding data and press key 

.

Using key <START> to monitor signals, key <STOP> to stop monitoring signals.

3.6.2 Signal State Viewing

1) Press key 

to select the DIAGNOSE page.
2) The respective address explanation and meaning are shown at the lower left corner of the screen when the cursor is moved left or right.
3) Move the cursor to the target parameter address or key in the parameter address, then press key 

to search.
4) In【WAVE】page, the feedrate, acceleration and jerk of each axis can be displayed. It is easy to debug the system and find the optimum suited parameters for the drive and the motor.

3.7 Alarm Display

When an alarm is issued, “ALARM” is displayed at the lower left corner of the LCD. Press key 

to display the alarm page. There are 4 subpages: 【ALARM】, 【USER】, 【HISTORY】 and【OPERATE】, all of which can be viewed by the corresponding soft keys（See Fig.3-7-1 to Fig.3-7-4）. Whether the page is switched to alarm page when an alarm occurs can also be set by bit parameter No: 24#6.

1. Alarm page In <ALARM> page, press soft key【ALARM】to enter this page, which is shown in Fig.3-7-1.

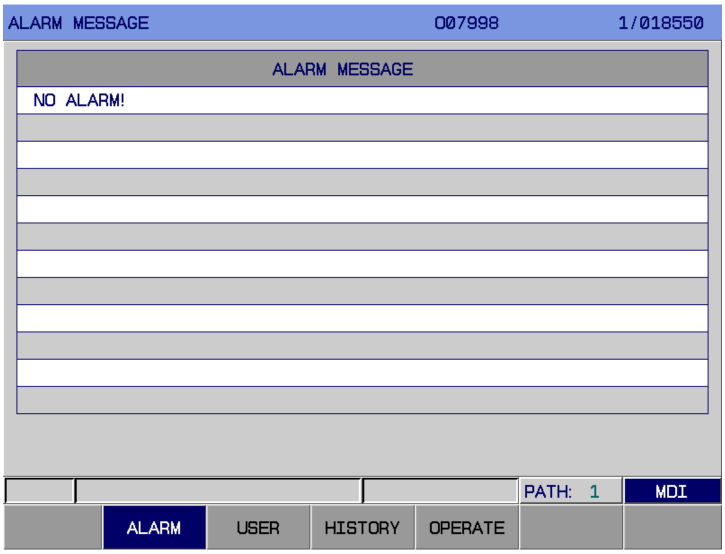


Fig. 3-7-1

In alarm page, the message of current P/S alarm number is displayed. See details about the alarm in Appendix 2.

2. User page In <ALARM> page, press soft key 【USER】 to enter external alarm page, which is shown in Fig. 3-7-2.

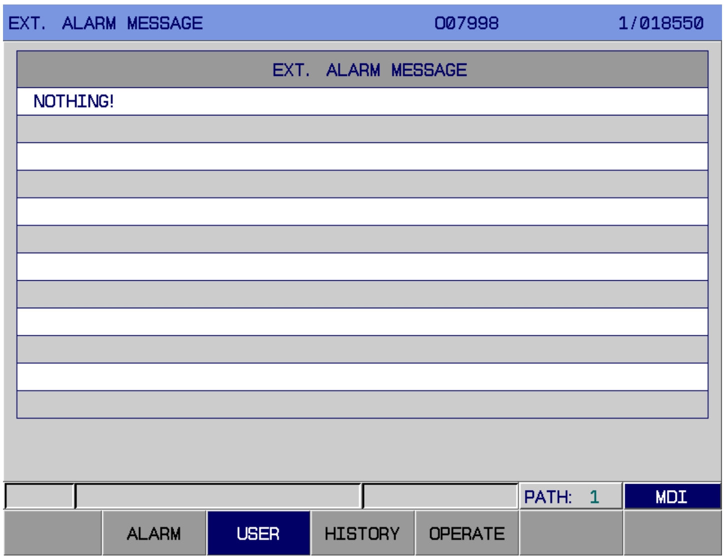


Fig. 3-7-2

See “GSK980TC3 CNC System PLC, Installation and Connection User Manual” for the details about the user alarm.

Note: The external alarm number can be set and edited by users according to the site conditions. The edited contents of the alarm are input into the system via a transmission software. The external alarm is the A of edit file LadChi.txt, and the two digits behind it are set by bit parameters 53.0~53.3. (The default is 01, i.e. the file name is LadChi01.txt)**

3. History page In <ALARM> page, press soft key 【HISTORY】 to enter this page. See Fig. 3-7-3:

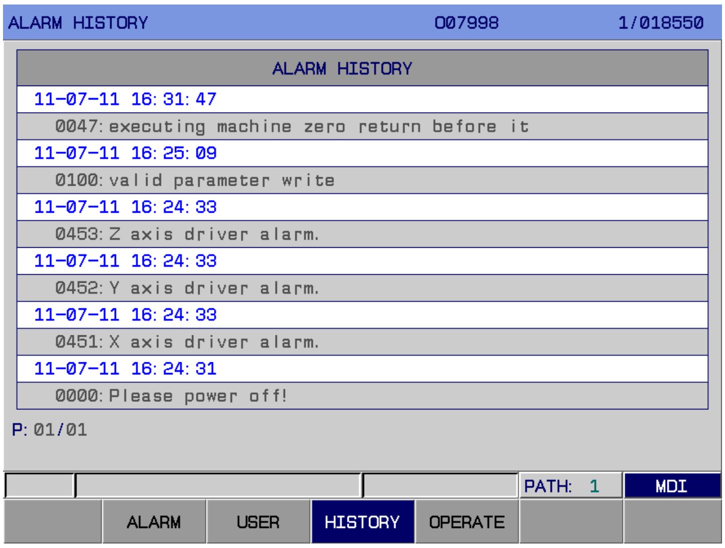


Fig. 3-7-3

In this page, the messages are arranged in chronological order for users' convenience.

4. OPERATE page In <ALARM> page, press soft key【OPERATE】to enter this page, as is shown in Fig. 3-7-4:

The OPERATE page displays the modification messages applied to the system parameters and ladders, e.g. content modification and time modification.

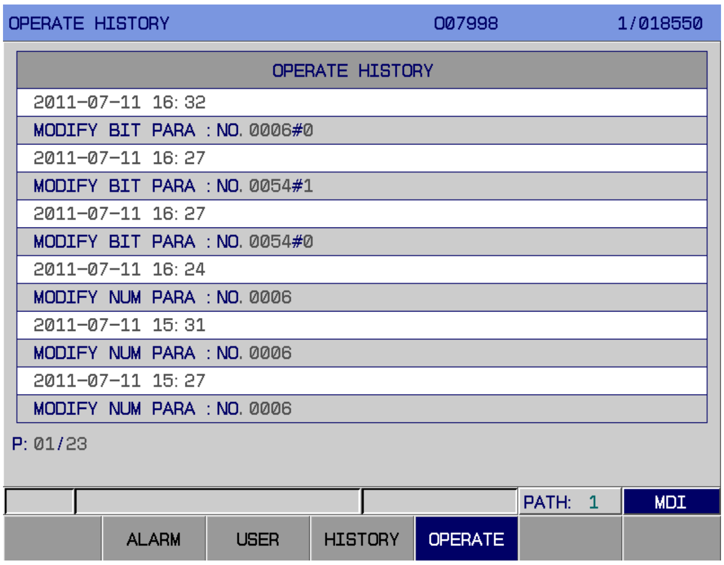


Fig. 3-7-4

OPERATE page can display 34 pages, while HISTORY alarm page can display 9 pages. The alarm time, alarm numbers, alarm messages and page numbers can be viewed using page keys.

The records of the HISTORY and OPERATE can be deleted by pressing key 

(system debugging level or above required).

3.8 PLC Display

Press the key  to display the PLC page. There are 5 subpages, including 【INFO】, 【⊞PLCGRA】, 【⊞PLCPAR】, 【PLCDGN】and【⊞PLCTRA】, which can be viewed by the corresponding soft keys（See Fig.3-8-1～Fig.3-8-5）.

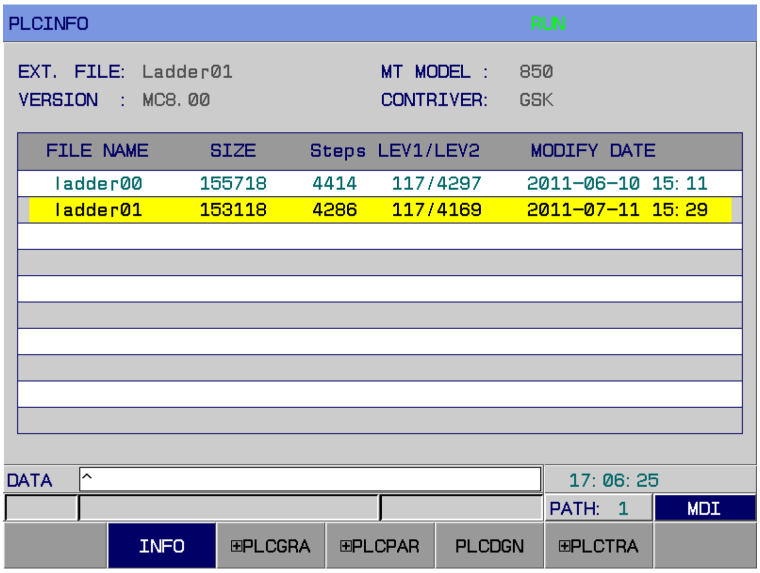


Fig. 3-8-1

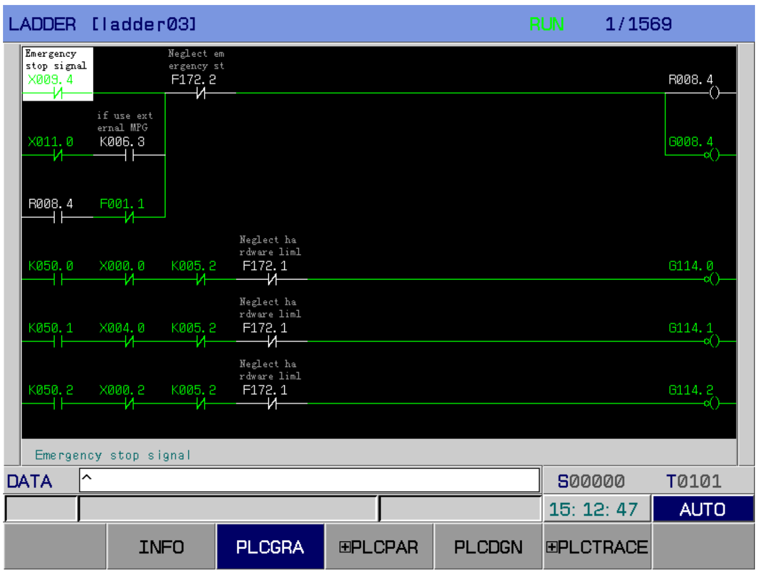


Fig. 3-8-2

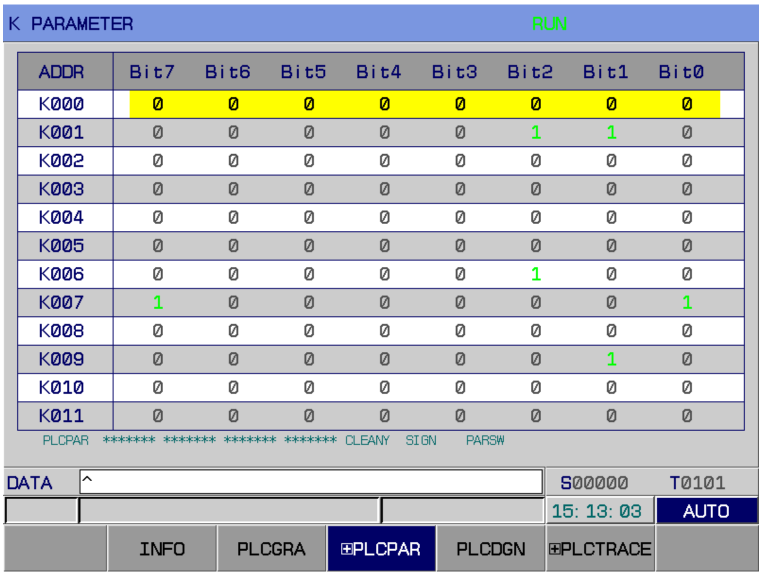


Fig. 3-8-3

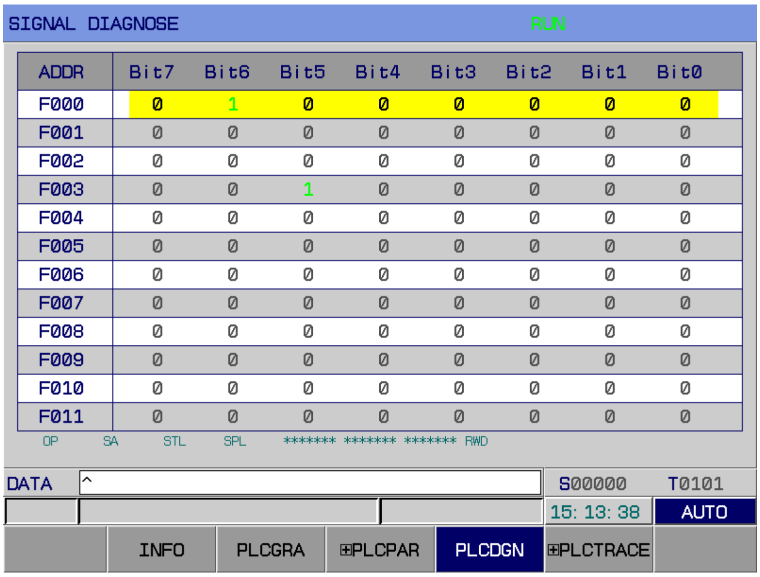


Fig. 3-8-4

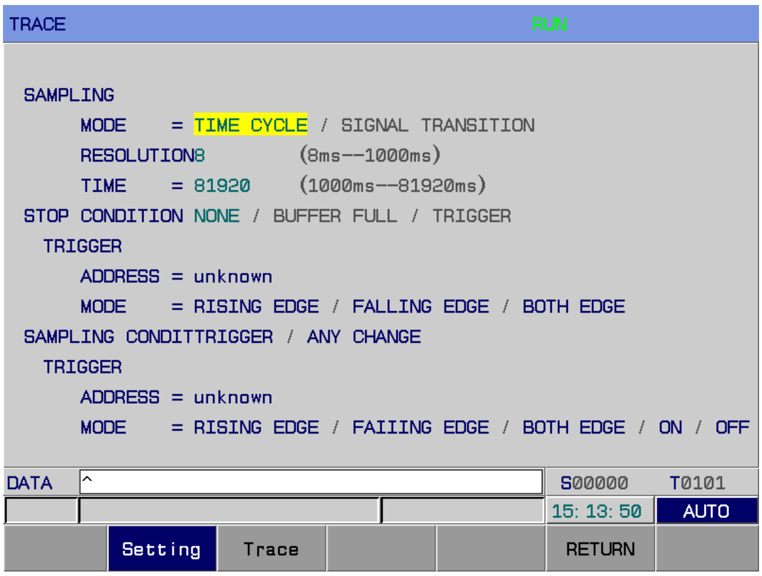


Fig. 3-8-5

II Operation

Note: Refer to GSK 980TC3 *CNC System PLC Connection and Installation User Manual* for the PLC ladder modification and relevant messages.

3.9 Help Display

Press key 

to display help page. There are 8 subpages, including 【SYS INFO】, 【OPRT】, 【ALARM】, 【G CODE】, 【PARA】,【MACRO】,【⊞PLC.AD】and【CALCULA】. All of them can be viewed by corresponding soft keys（See Fig. 3- 9- 1～3- 9- 12）.

1. System information page In <HELP> page, press soft key【SYS INFO】to enter system information page (See Fig. 3-9-1):

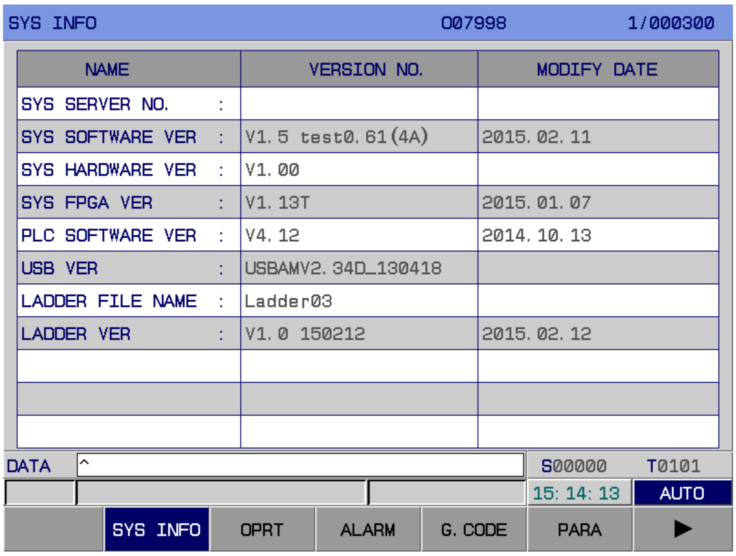


Fig. 3-9-1

2. OPRT page In <HELP> page, press soft key【OPRT】to enter this page, as is shown in Fig. 3-9-2:

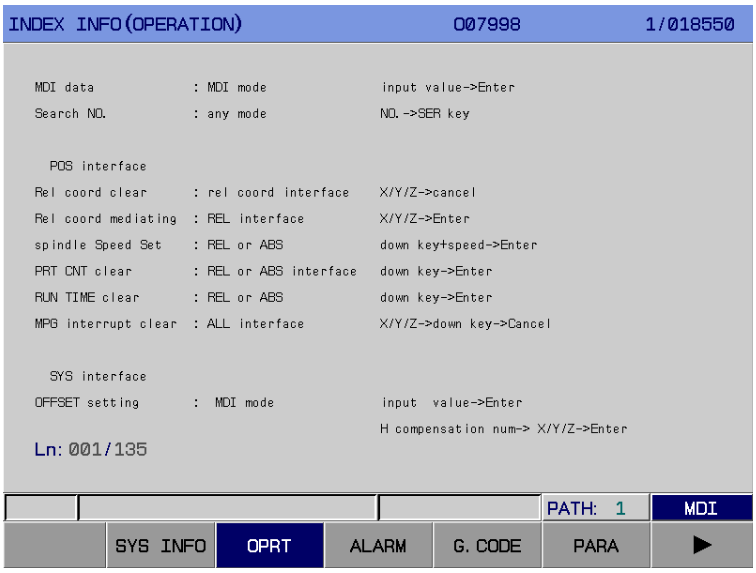


Fig. 3-9-2

The various operation steps on different pages are described in <HELP> **(**OPRT**)** page, you can get help in the HELP page if you are unfamiliar with some operations.

3. ALARM page　In <HELP> page, press soft key【ALARM】to enter this page. See Fig. 3-9-3:

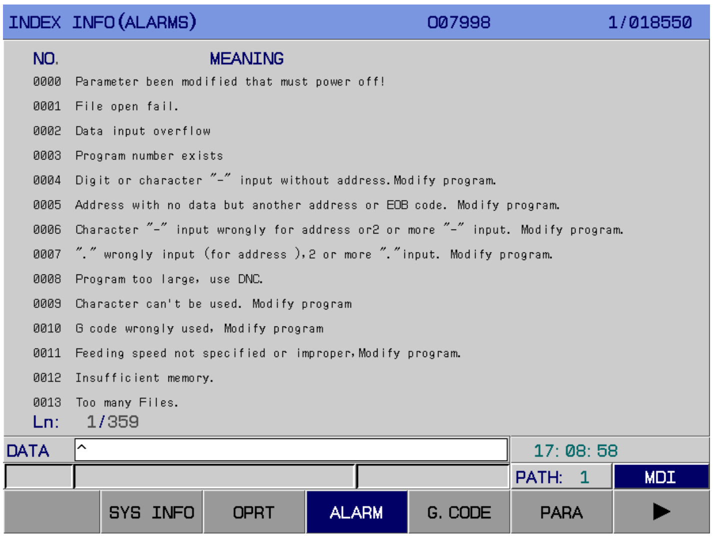


Fig. 3-9-3

The meaning and handling for each alarm number is described in this page.

4. G code page　In <HELP> page, press soft key【G. CODE】to enter this page. See Fig. 3-9-4:

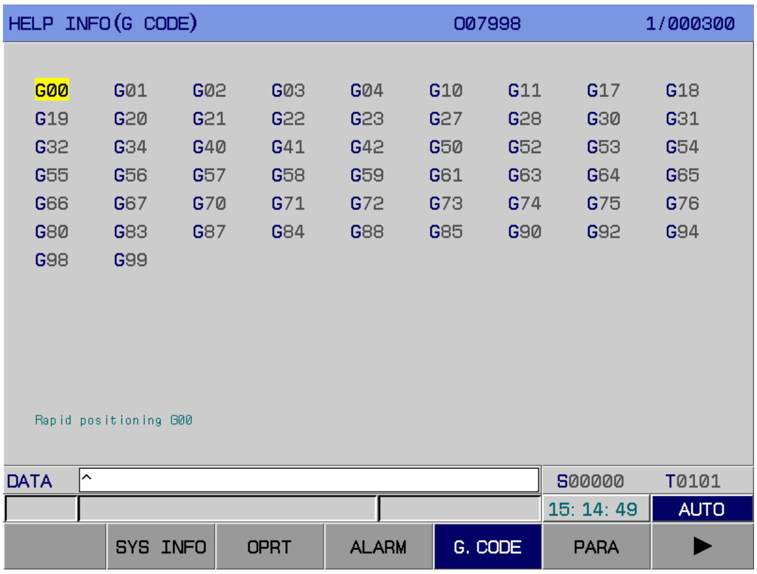
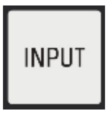


Fig. 3-9-4

The definitions of G codes used in system are shown in G code page. Move the cursor to the G code to be viewed, then its definition is shown at the lower left corner of the page (Fig. **3-9-4**). If you need to know the format and usage of a G code, press key 

on the panel after selecting the G code. Press key 

to return. See Fig. 3-9-5:

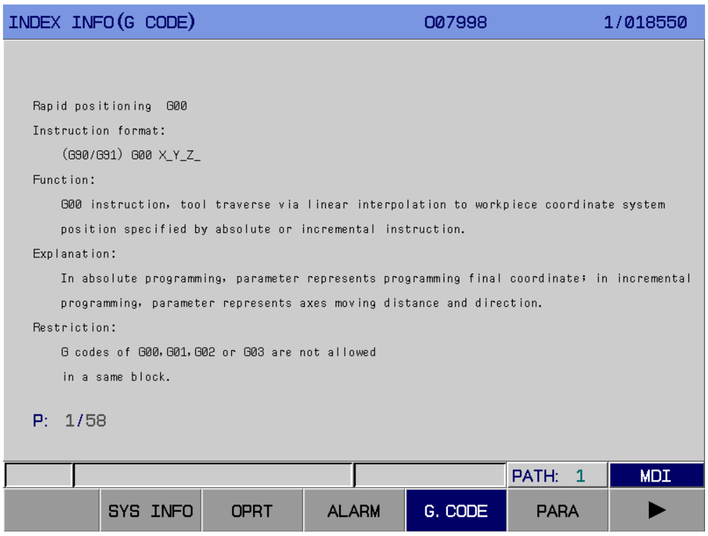


Fig. 3-9-5

The formats, functions, explanations and restrictions of codes are introduced in this page. You can find the corresponding information here if you are unfamiliar with these codes.

5. Parameter page In <HELP> page, press soft key【PARA】to enter this page, as is shown in Fig.3-10-6:

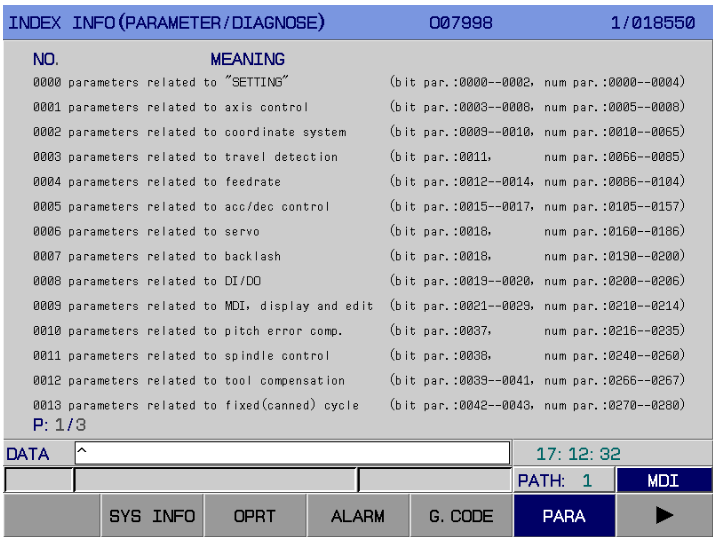


Fig. 3-9-6

The parameter setting for each function is described in the page. If you are not familiar with the setting, you can find corresponding information here.

6. Macro page In <HELP> page, press soft key【MACRO】 to enter this page, as is shown in Fig.3-10-7:

II Operation

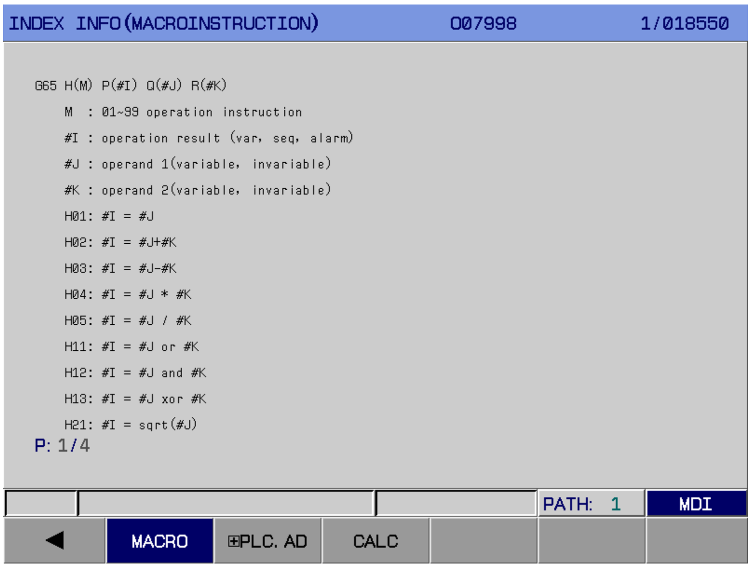


Fig. 3-9-7

The formats and a variety of operation codes of the macro codes are described in this page, and the setting ranges for local variable, common variable and system variable are also given. If you are unfamiliar with the macro code operations, you can get corresponding information here.

7. PLC.AD page In <HELP> page, press soft key【PLC.AD】to enter this page. There are four subpages, including【F. ADDR】,【G. ADDR】,【X. ADDR】and【Y. ADDR】, as is shown in figures 3-9-8～3-9-11:

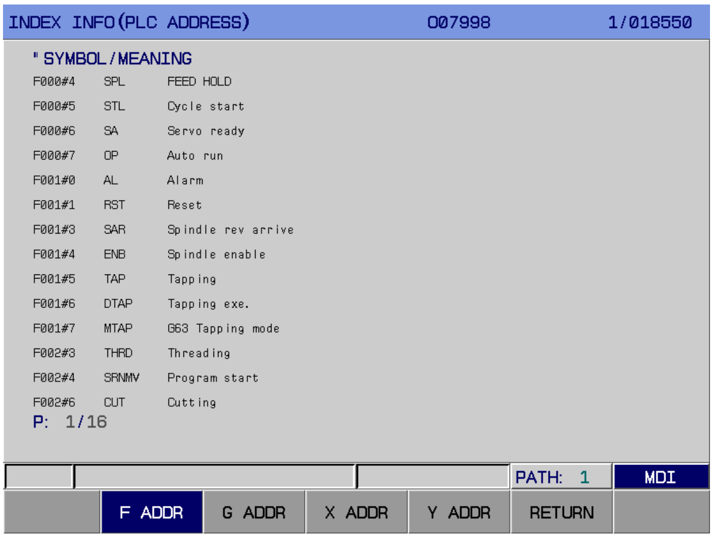


Fig. 3-9-8

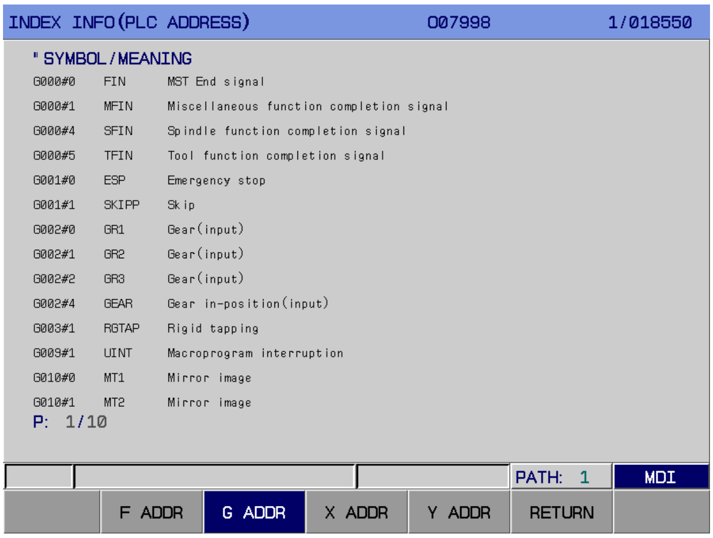


Fig. 3-9-9

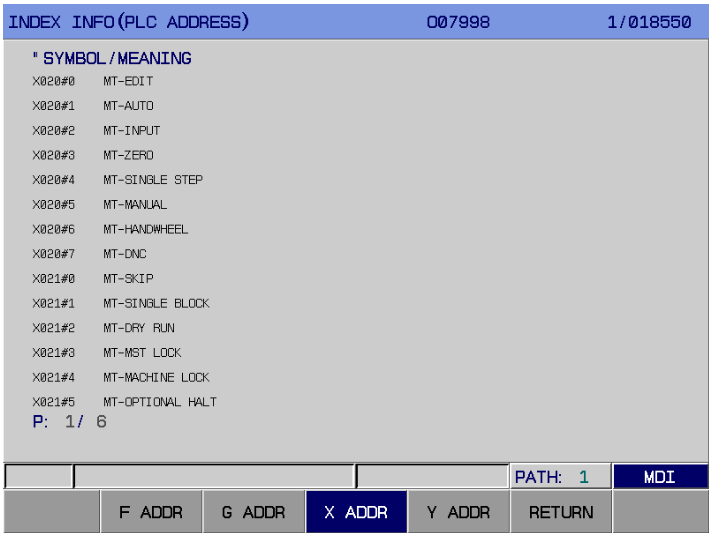


Fig. 3-9-10

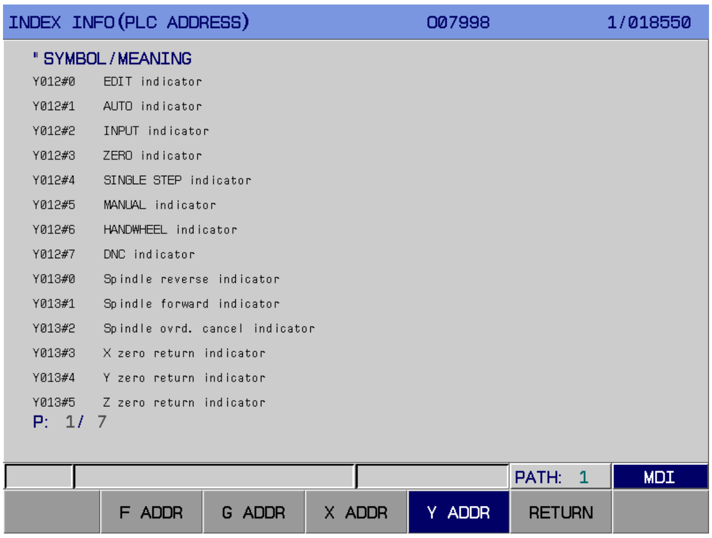


Fig. 3-9-11

The PLC addresses, signs, meanings are described in this page, and you may get the corresponding information here if you are unfamiliar with these addresses.

8. CALCULA page In <HELP> page, press soft key【CALCULA】to enter this page. See fig. 3-9-12:

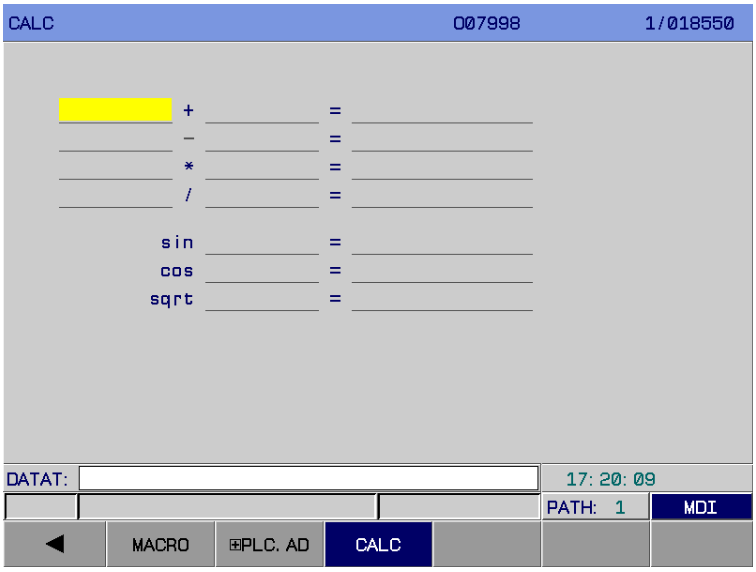


Fig. 3-9-12

Chapter 4 Manual Operation

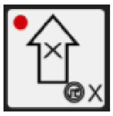
Press key 

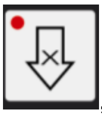
to enter Manual mode, which includes manual feed, spindle control and machine panel control, etc.

4.1 Coordinate Axis Movement

In Manual mode, each axis can be moved at MANUAL feedrate or manual rapid traverse speed separately.

4.1.1 Manual Feed

X axis can be moved in the positive or negative direction by pressing and holding key 

or , and the feedrate can be changed by feedrate override. If the key is released, the X axis movement is stopped. That of Z axis is the same. The three axes simultaneous moving is not available in this system, but the two axes simultaneous zero return is supported by the system.

Note: The manual feedrate of each axis is set by parameter P98.

4.1.2 Manual Rapid Traverse

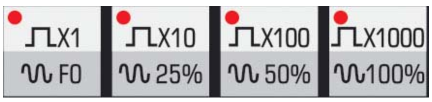
Press key 

to enter Rapid Traverse state with its indicator lighting up. Then press manual feeding keys to move each axis at the rapid traverse speed.

Note 1: The manual rapid speeds are set by the parameter P170~ P173.
Note 2: Whether manual rapid traverse is effective before reference point return is set by the bit parameter N0:12#0.

4.1.3 Manual Feedrate and Manual Rapid Traverse Speed Selection

The manual feedrate override, which can be selected by the band switch, is divided into 21 gears (0%--200%) in Manual feed mode .

In manual rapid traverse, press keys 

to select the override of the manual rapid traverse speed. The rapid override is divided into four gears, including Fo，25％，50％and 100％ (The speed of F0 is set by data parameter P93).

Note: The rapid overrides are effective for the following speed.

(1) G00 rapid traverse

(2) Rapid traverse in canned cycle

(3) Rapid traverse in G28

(4) Manual rapid traverse

Example: If the rapid traverse speed is 6m/min and override is 50%, the actual speed is 3m/min.

4.1.4 Manual Intervention

While a program being executed in Auto, MDI or DNC mode is shifted to MANUAL mode after a dwell operation, the manual intervention is available. Move the axes manually, then shift the mode to the previous one after the intervention. When key 

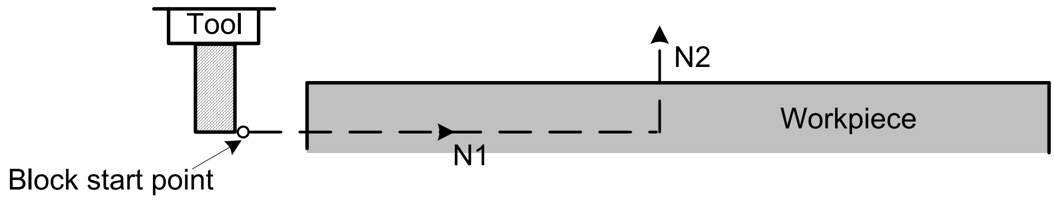
is pressed to run the program, each axis returns to the original intervention point rapidly by G00, and the program execution continues.

Explanation:

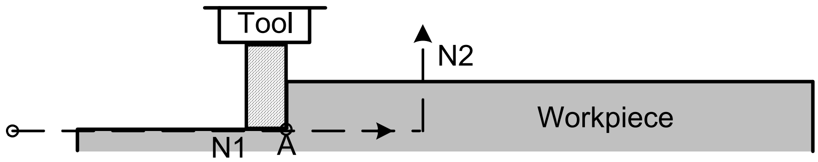
1. If the single block switch is turned on during return operation, the tool performs single block stop at the manual intervention point.
2. If an alarm or resetting occurs during the manual intervention or return operation, this function will be cancelled.
3. Use machine lock, mirror image and scaling functions carefully during manual intervention.
4. Machining and workpiece shape should be taken into consideration prior to the manual intervention to prevent tool or machine damage.

The manual intervention operations are shown in the following figure:

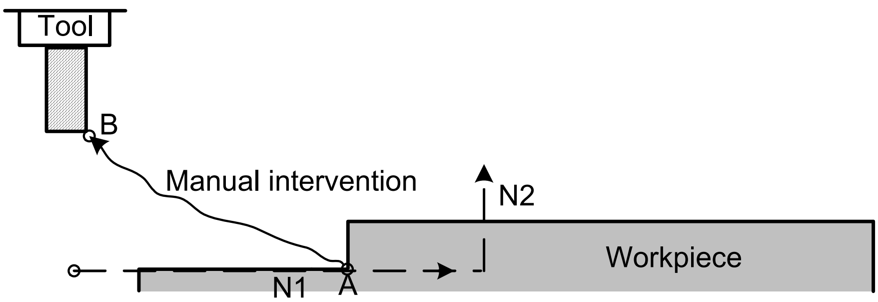
1. The N1 block cuts a workpiece



2. The tool machine is stopped by pressing the feed hold key in the middle of the N 1 block



3、After retracting the tool to point B, tool machine movement is restarted.



4. After automatic return to point A by G00 speed, the remaining moving command of block N1 is executed.

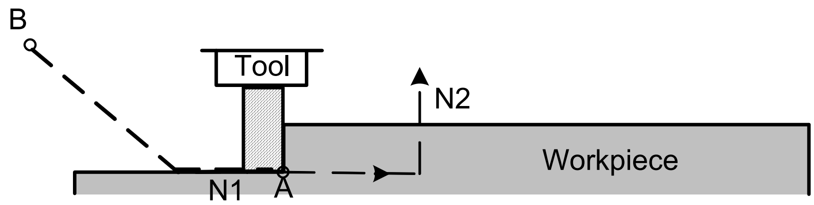


Fig.4-1-4-1

4.2 Spindle Control

4.2.1 Spindle Rotation CCW

: Specifies S speed in MDI mode; in Manual/MPG/Step mode, press this key to rotate the spindle counterclockwise.

4.2.2 Spindle Rotation CW

: Specifies S speed in MDI mode; in Manual/MPG/Step mode, press this key to rotate the spindle clockwise.

4.2.3 Spindle Stop

: In Manual/MPG/Step mode, press this key to stop the spindle.

4.2.4 Spindle Automatic Gear Shift

Whether the spindle is frequency conversion control or gear control is set by bit parameter No:1#2. If parameter No:1#2=1, the spindle auto gear shift is controlled by PLC. Three gears (gear 1 to gear 3) are available in this system, and the maximum speed of each gear is set by parameters（P246, P247, P248）respectively. NO:1#2=1: the spindle speed is changed by I/O controlling automatic gear shift. Presently, the system uses the 3-gear control (S1, S2, S3) , which can modify the ladder diagram adding the gear output. The system will automatically select the corresponding gear after executing S speed code speed.

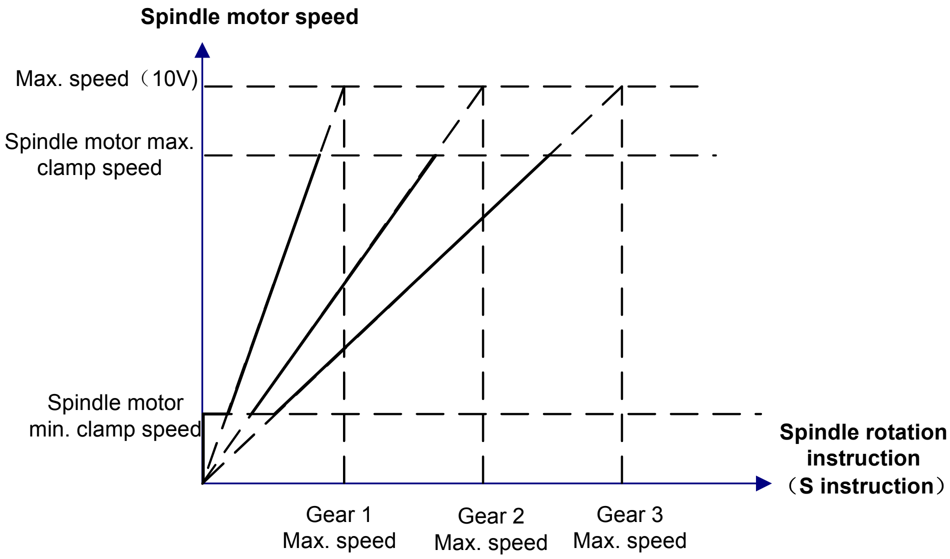


Fig.4-2-4-1

Note: When the spindle auto gear shift is effective, the spindle gear is detected by gear in-position signal and S code is executed.

4.3 Other Manual Operations

4.3.1 Cooling Control

: A compound key, used to switch between cooling ON and OFF. ON: the indicator lights up; OFF: the indicator goes out.

4.3.2 Lubricating Control

: A compound key, used to switch between lubricant ON and OFF. ON: the indicator lights up; OFF: the indicator goes out.

4.3.3 Chuck Control

: In manual mode, hit the pedal switch and the chuck releases. The chuck state displays the release in【POSITION】, and the switch is hit again, the chuck clamps, and the state displays the clamp.

4.3.4 Tailstock Control

: switch the machine tailstock forward/backward. The indicator ON means its forward, and the indicator OFF means its backward.

4.3.5 Manual Tool Change Control

: In Manual/MPG/STEP mode, press the key and the tool post rotates to the next tool. Press <PLC> soft key, and then press【**+Ladder Para**】 to enter【DATA】 page to view the total quantity.（Refer to the machine tool manufacturer's user manual）

4.4 Toolsetting Operation

Machining a part generally needs several different tools. Because of the tool installation and tool offset, when each tool rotates to the cutting position, each tool nose's position does not completely coincide. Without considering the tools' offset when programming, the system sets the tool offset's automatic creating toolsetting method, which is convenient to the toolsetting operation. After toolsetting operations, the user compiles only programs according to the part drawing and processing technology when programming without considering the tools' offset, and calls the corresponding tool compensation values in machining programs. The system sets trial cut toolsetting, machine zero return toolsetting and other toolsetting modes.

4.4.1 Trial Cut Toolsetting

When the workpiece coordinate system does not change, the trial cut toolsetting can be executed, which steps shown below: (set the workpiece coordinate system based on the workpiece end face).

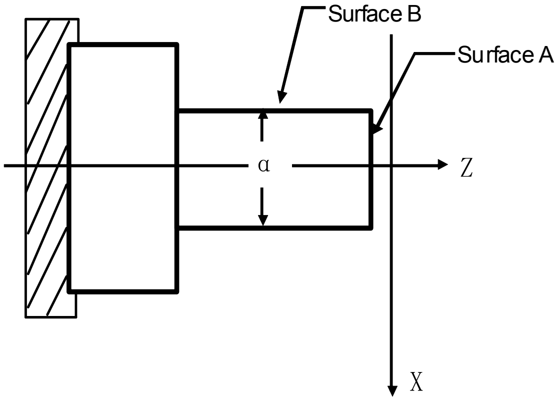
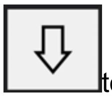
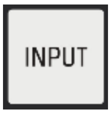
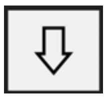
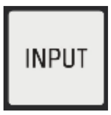


Fig. 4-4-1-1

1. Select any one tool（suppose to be No.1 tool）, without tool compensation；
2. Execute A surface cutting along X negative direction in Manual mode;
3. Release the tool along X positive direction when Z does not move, and stop the spindle rotation;
4. Press  to enter the offset page display mode, press  ,  to move the cursor to select No. 1 offset number as the tool's offset number;
5. Orderly input the address key “Z”, digital key “0” and ；
6. Execute B surface cutting along Z negative direction in Manual mode;
7. Release the tool along Z positive direction when X does not move, and stop the spindle rotation;
8. Measure the diameter "α'"（suppose α'=30）；
9. Press  to enter the offset page display mode, press  ,  to move the cursor to select No.1 offset number;
10. Orderly input the address key “X”, digital key “3”, “0” and ；
11. Move the tool to the safety tool change position;
12. Change another tool(suppose to be No. 2 tool), without tool compensation;

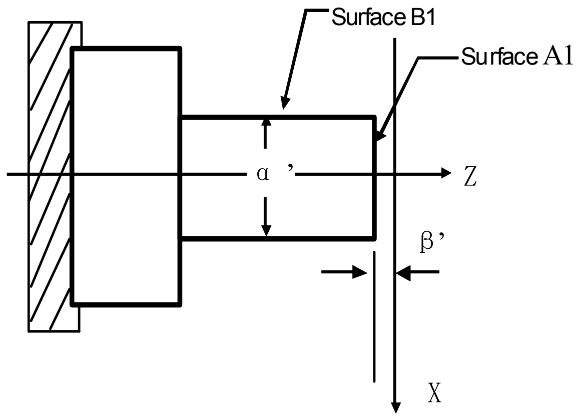
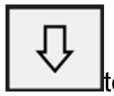
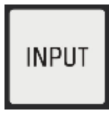


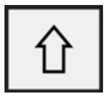
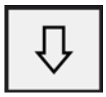
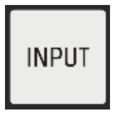
Fig. 4-4-1-2

13. Execute A1 surface cutting along X negative direction in Manual mode;
14. Release the tool along X positive direction when Z does not move, and stop the spindle rotation;
15. Measure the distance “β”（suppose β=1） between A1 surface and workpice coordinate system).
16. Press 

to enter the offset page display mode, press  ,  to move the cursor to select No. 2 offset number as the tool’s offset number;
17. Orderly input the address key “Z”, sign key “-” and 

;
18. Execute B1 surface cutting along Z negative direction in Manual mode;
19. Release the tool along Z positive direction when X does not move, and stop the spindle rotation;
20. Measure the diameter "α’"（suppose α’=28）;

    Press 

to enter the offset page display mode, press  ,  to move the cursor to select No.2 offset number;
21. Orderly input the address key “X”, digital key “2”, “8” and 

;
22. For other toolsetting method, repeat the above Step 11～22.

4.4.2 Machine Zero Return Toolsetting

Using the toolsetting method does not require a reference tool or not. When the tool wears or any one tool is regulated, the user executes the tool’s toolsetting again. Before toolsetting, return to the machine zero one time (see Chapter 9.1 about the machine zero). After power-off, executing the machine zero return one time can continue the machining, which operation is convenient.

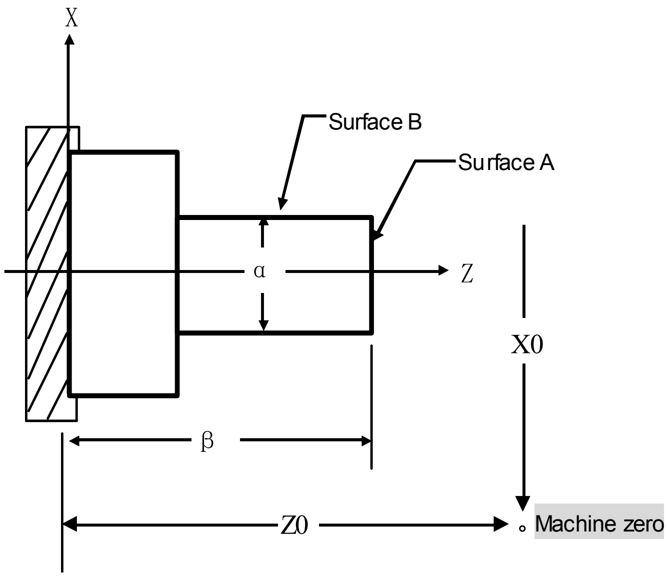


Fig.4-4-2-1

Operation step:

1） Press 

 to enter the machine zero return operation mode, press X-axis positive key and Z-axis positive key to make two return to the machine zero one time;

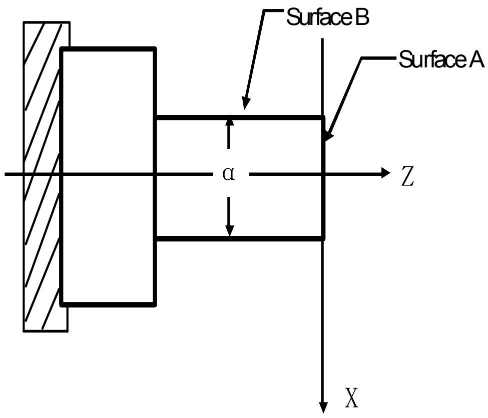


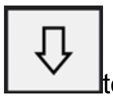
Fig. 4-4-2-2

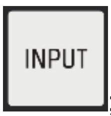
2） Select any one tool（suppose to be No.1 tool）, without tool compensation;

3） Execute A surface cutting along X negative direction in Manual mode;

4） Release the tool along X positive direction when Z does not move, and stop the spindle rotation;

5） Press 

 to enter the offset page display mode, press  ,  to move the cursor to select No. 1 offset number as the tool's offset number;

6） Orderly input the address key "Z", digital key "0" and 

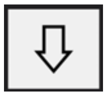
;

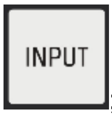
7） Execute B surface cutting along Z negative direction in Manual mode;

8） Release the tool along Z positive direction when X does not move, and stop the spindle rotation;

9）Measure the diameter "α’"（suppose α’=30）;

10） Press 

to enter the offset page display mode, press  ,  to move the cursor to select No.1 offset number;

11） Orderly input the address key “X”, digital key “3”, “0” and 

;

12） Move the tool to the safety tool change position;

13） Change another tool(suppose to be No. 2 tool), without tool compensation;

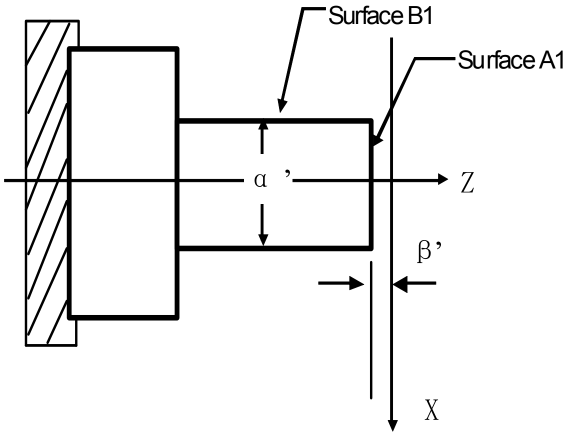


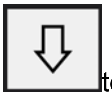
Fig.4-4-3-3

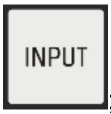
14） Execute A1 surface cutting along X negative direction in Manual mode;

15） Release the tool along X positive direction when Z does not move, and stop the spindle rotation;

16） Measure the distance “β”（suppose β=1） between A1 surface and workpice coordinate system);

17） Press 

to enter the offset page display mode, press  ,  to move the cursor to select No. 2 offset number as the tool’s offset number;

18） Orderly input the address key “Z”, sign key “-”digital key “1” and 

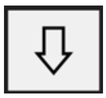
;

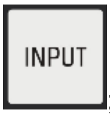
19） Execute B1 surface cutting along Z negative direction in Manual mode;

20） Release the tool along Z positive direction when X does not move, and stop the spindle rotation;

21） Measure the diameter "α’"（suppose α’=28）;

22） Press 

to enter the offset page display mode, press  ,  to move the cursor to select No.2 offset number;

23） Orderly input the address key “X”, digital key “2”, “8” and 

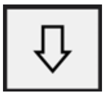
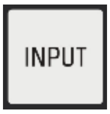
;

24） For other toolsetting method, repeat the above Step 12～23.

4.5 Tuning a Tool Compensation Value

Tuning a tool compensation value uses only U, W input.

Example: X-axis tool compensation value needs to increase 0.010mm. Press 

to enter the offset page display mode, press  ,  move the cursor to select 001’s some offset number, input U0.010, and press 

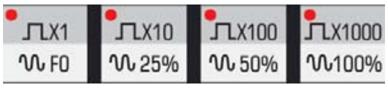
to complete the operation, and the system automatically adds 0.010mm to the previous tool compensation value.

Chapter 5 Step Operation

5.1 Step Feed

Press key  to switch the MPG mode and STEP mode. Pressing it every time can switch the two mode. In step feed mode, the machine moves by the selected step each time.

5.1.1 Selection of Moving Amount

Press any of keys  to select a moving increment, then the increment will be shown on the screen. A step of 0.100 is displayed in <POSITION> page(See Fig. 5-1-1-1):

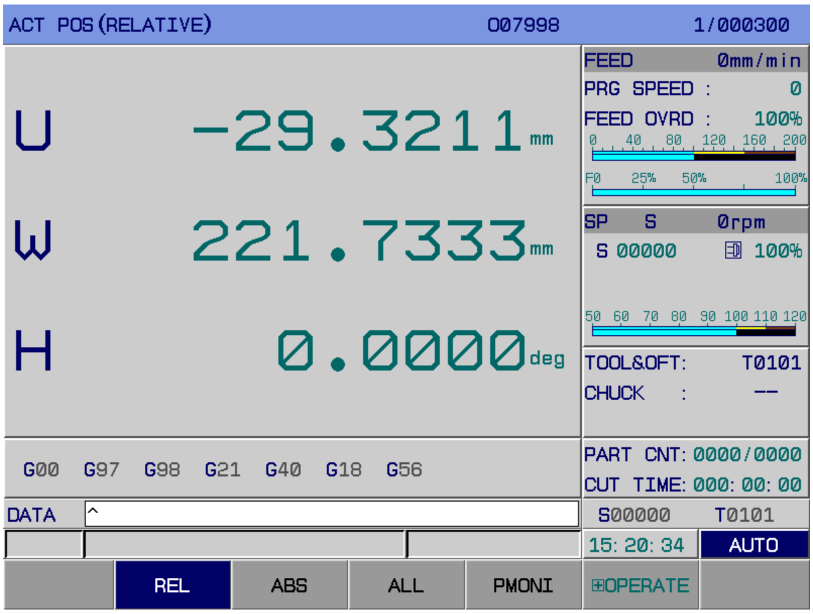
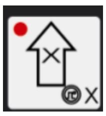
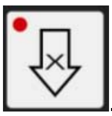


Fig. 5-1-1-1

By pressing the moving key each time, the corresponding axis on the machine is moved 0.1 mm.

5.1.2 Selection of Moving Axis and Direction

X axis may be moved in the positive or negative direction by pressing axis and direction key  or . Press the key once, the corresponding axis will be moved for a step distance defined by system. The operation for Z axis is identical with that of X axis. Simultaneous manual moving for 2 axes is unavailable in this system, but simultaneous zero return for 2 axes is available.

5.1.3 Step Feed Explanation

The step feed max. clamp speed is set by data parameter P155.
The step feedrate is beyond the control of the feedrate and rapid override.

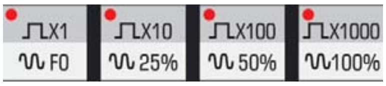
5.2 Auxiliary Control in Step Mode

It is the same as that of Manual mode. See Sections 4.2 and 4.3 in this manual for details.

Chapter 6 MPG Operation

Press key  to switch the MPG mode and STEP mode. Pressing it every time can switch the two mode. In step feed mode, the machine moves by the selected step each time.

6.1 MPG Feed

Press  to select the movement increment. The movement increment is displayed in the page. Press the key 25% and the system displays the MPG increment: 0.100 in <POS> page.（See **6-1-1**）:

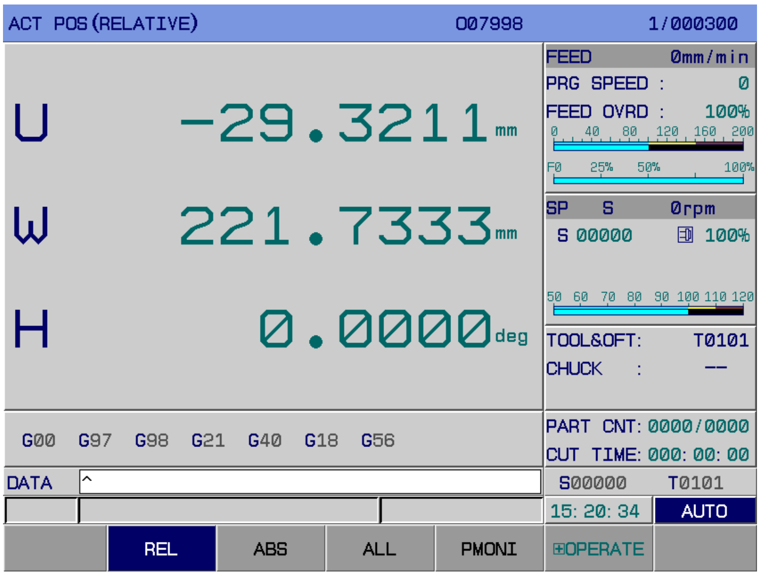
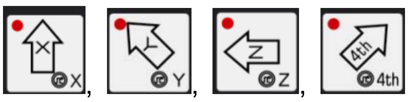
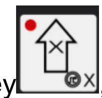


Fig. 6-1-1

6.1.1 Selection of Moving Axis and Direction

In MPG mode, select the moving axis to be controlled by the MPG, and press the corresponding key （）, then you can move the axis by the MPG. In MPG mode, if X axis is to be controlled by the MPG, press key , then you can move the X axis by rotating the MPG. Generally speaking, the MPG CW rotation is the feed's positive direction and the MPG CCW rotation is the feed's negative direction.

The feed direction is controlled by MPG rotation direction. See the manual provided by the machine tool builder for details. In general, MPG CW rotation indicates the positive feed, while CCW rotation indicates the negative feed.

6.1.2 MPG Feed Explanation

1. The relationship between MPG scale and machine moving amount is as follows:

Table 6-1-2-1

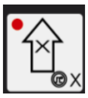
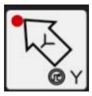
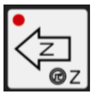
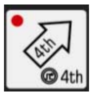
	Moving amount per MPG scale		
MPG increment （mm）	0.001	0.01	0.1
Machine moving amount (mm)	0.001	0.01	0.1
MPG increment （inch）	0.001	0.01	0.1
Machine moving amount (inch)	0.0001	0.001	0.01

2. The values in the table above vary with the mechanical transmission. See the manual provided by the machine tool builder for details.
3. No.17#5 is set to 1, the MPG moving amount selects complete run. The rotation speed of the MPG cannot exceed 5r/s, otherwise, the scale and the moving amount may be inconsistent.

6.2 Control in MPG Interruption

6.2.1 MPG Interruption Operation

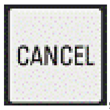
In auto, MDI mode, press【ALL】soft key in【POS】page to enter the all page, press {【OPERATION】and then press【INTERRUPT ON】. Selecting the panel MPG axis number can enter the MPG interruption state to execute the MPG interruption operation. Press【INTERRUPT SWITCH】one time, the system exits the MPG.

When the system enters the MPG interruption state, , , ,  is selected to execute the axis selection. When some axis' MPG interruption axis selection signal is "1", rotating the MPG can perform the axis' MPG interruption, gear amount in the MPG interruption state is the same the MPG feed equivalent.

The speed in the MPG interruption state is a sum of the speed in automatic run and the traverse speed in the MPG interruption state. But the added speed is controlled within the range of being than or equal to the upper speed of the axis' cutting feed. For example, when the MPG's upper limit is 2000mm/min, and the current feedrate is 2000mm/min, the MPG interruption speed range is 4000～0mm/min (the negative means the MPG interruption direction is opposite to the feed direction)

（The coordinate system for MPG interruption is shown in Fig.6-2-1-2.）

Fig. 6-2-1-2

Steps to clear MPG interruption coordinate system: Press , move the cursor upward and downward till the MPG interruption coordinate X flickers, and press key , then the coordinate system is cleared. The operations for Y and Z axes are the same as above; when the zero return operation is performed, the coordinate system is cleared automatically too.

Note: When the MPG interruption function is used to adjust the coordinate system, if an alarm or resetting occurs, the function is cancelled.

6.2.2 Relationship between MPG Interruption and Other Functions

When the axis moves by MPG interruption, the actual position of movement axis changes, the machine coordinates update and the absolute coordinates do not update. So after the MPG interruption, the slide moves and the machine coordinate system does not change, but the workpiece coordinate system does not offset.

Table **6-2-2-1**

Display	Relationship
Machine lock	After machine lock is effective, the machine movement by using MPG interruption is ineffective.
Absolute coordinate value	MPG interruption does not change the absolute coordinate values.
Relative coordinate value	MPG interruption does not change the relative coordinate values.
Machine coordinate value	The change amount of the machine coordinate value is the displacement amount caused by MPG rotation.

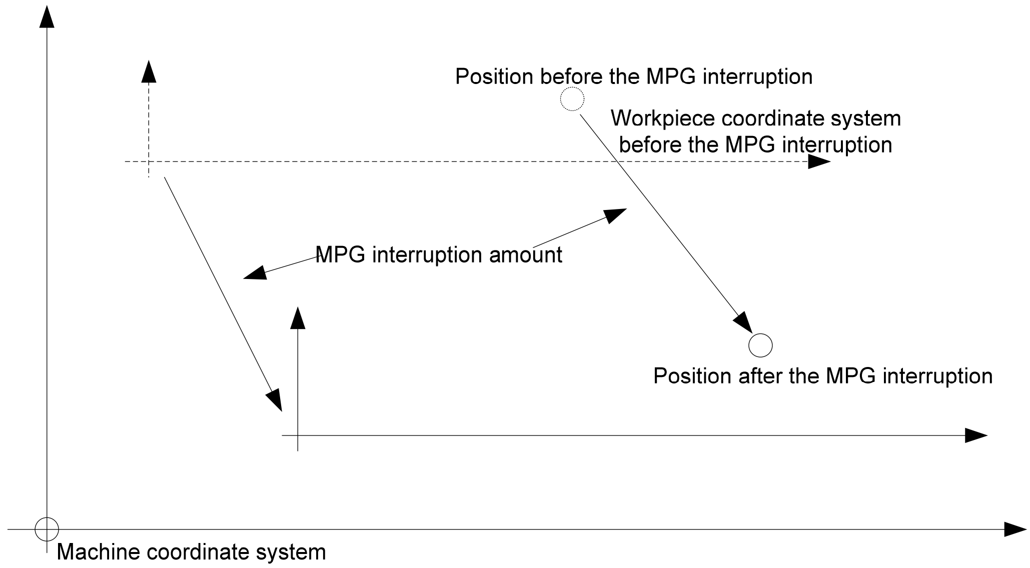
Note: The moving amount of MPG interruption is cleared when the manual reference point return is performed for each axis.

6.2.3 MPG Interruption Amount Cancel

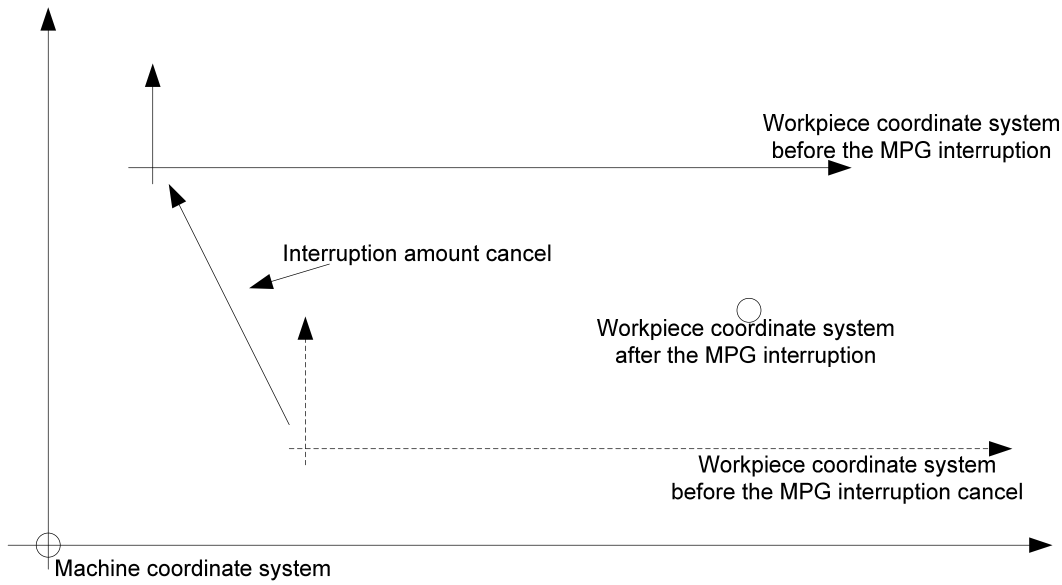
The MPG interruption makes the machine coordinate system offset's workpiece coordinate system return to the workpiece coordinate system which is prior to the offset, which is called the interruption amount cancel.

When the interruption amount cancel operation is performed, the workpiece coordinate system offsets only the MPG interruption amount's amount, and then the MPG interruption amount is reflected to the absolute coordinate value.

The workpiece coordinate offset created by MPG interruption is shown below:



After the MPG interruption is cancelled, the workpiece coordinate system recovers to the state which is prior to the interruption, but the actual position does not change, which is shown below:



The interruption amount can be cancelled in the followings:

- When the manual reference point is returned
- G28 is specified without establishing a reference point
- Press [INTERRUPT ZERO] to execute the interruption amount clear

6.3 Auxiliary Control in MPG Mode

The auxiliary operation in MPG mode is identical with that in JOG mode. See Sections 4.2 and 4.3 for details.

6.4 MPG Trial Cut Function

Manually rotating the MPG controls the part program run, and the user can operate the machine along the machining program commanding the tool path. The function is mainly used to the workpiece trail-cut and check machining programs.

Operation method:

In Auto mode, open the MPG trial cut, press 

to make each axis not to move, at the moment, rotating the MPG controls the part program run. The faster the MPG rotates, the faster the program execution speed is, and vice versa. Each pulse's movement amount can be adjusted by the rapid override.

When the system is in MPG trial cut mode, 

is pressed, the system returns to Auto mode. All operations in MPG trail cut mode are the same as those in Auto mode.

Speed when the system runs the interpolation command in MPG trial cut state:

G0：current rapid override X MPG override X MPG speed

G1：cutting override X MPG override X MPG speed

Note 1：After the MPG trial cut function is started, it is valid.

Note 2：Executing the single block pause is valid in single block mode.

Chapter 7 Auto Operation

7.1 Selection of the Auto Run Programs

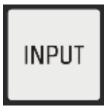
1. Program loading in auto mode

(a) Press key 

to enter the Auto mode;

(b) Press key 

to enter the 【DIR】 page, move the cursor to find the target program;

(c) Press key 

for confirmation.

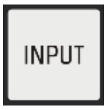
2. Program loading in Edit mode

(a) Press key 

to enter the Edit mode;

(b) Press key 

to enter the【DIR】page, move the cursor to find the target program;

(c) Press key 

for confirmation.

(d) Press key 

to enter the Auto mode;

7.2 Auto Run Start

After selecting the program using the two methods in section 7.1 above, press key 

to execute the program automatically. The execution of the program can be viewed by switching to <POSITION>, <MONI>, <GRAPH> etc. pages.

The program execution is started from the line where the cursor is located, so it is recommended to check whether the cursor is located at the program to be executed and whether the modal values are correct before pressing key 

. If the cursor is not located at the start line from which the program is started, press key 

, and then key 

to run the program automatically

II Operation

from the start line.

Note: The workpiece coordinate system and reference offset values cannot be modified during program execution in Auto mode.

7.3 Auto Run Stop

In Auto run, to stop the program being automatically executed, the system provides five methods:

1. Program stop (M00)

After the block containing M00 is executed, the auto running pauses and the modal message is saved. After key 

is pressed, the program execution continues.

2. Program optional stop (M01)

If key 

is pressed before the program execution, the automatic running pauses and the modal message is saved when the block containing M01 is executed in the program. After key 

is pressed, the program execution is continued.

3. Pressing key 

If key 

is pressed during the auto running, the machine states are as follows:

1) Machine feeding slows down and stops;

2) Dwell continues if Dwell (G04 code) is executed;

3) The other modal message is saved;

4）The program execution continues after key 

is pressed.

4. Pressing key 

See Section 2.3.1 in this manual.

5. Pressing Emergency Stop button

See Section 2.3.2 in this manual.

In addition, if the control is switched to other mode from Auto mode, DNC mode or MDI page of MDI mode in which the program is being executed, the machine can also be stopped.

The steps are as follows:

1）If the control is switched to Edit, MDI, DNC mode, the machine stops after the current block is executed.

2）If the control is switched to MANUAL, MPG, Step mode, the machine interruption stops immediately.

3）If the control is switched to Machine zero interface, the machine slows down to stop.

7.4 Auto Running from any Block

This system allows the auto run to start from any block of the current program. The steps are shown as follows:

1. Press key 

to enter Manual mode, start spindle and other miscellaneous functions;
2. Execute the modal values of the program in MDI mode, and ensure the modal values are correct;
3. Press key 

to enter Edit mode, and press key 

to enter program page, then find the program to be machined in【DIR】.
4. Open the program, and move the cursor to the block to be executed;
5. Press key 

to enter Auto mode;
6. Press key 

to execute the program automatically.

Note 1: Before execution, confirm the current coordinate point is the end position of the last block (confirmation for the current coordinate point is unnecessary if the block to be executed is absolute programming and contains G00/G01);

Note 2: If the block to be executed is for tool change operation, etc, ensure no interference and collision occur between the current position and workpiece in a bid to prevent machine damage and personnel hurt.

7.5 Dry Run

Before the machining by a program, use “Dry Run” (usually in combination with “M.S.T. Lock” or “Machine Lock”) to check the program.

Press key 

to enter Auto mode, and press key 

(that the indicator on the key lights up means Dry Run state is entered).

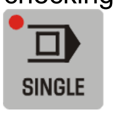
In rapid feed, the program speed equals to Dry Run speed × rapid feed override.

In cutting feed, the program thread’s actual cutting speed=F*S（S is a programmed value）F/S is the current mode.

Note 1: The Dry Run speed is set by data parameter P86;
Note 2: In cutting feed, whether the Dry Run is effective is set by bit parameter NO:12#6;
Note 3: In rapid positioning, whether the Dry Run is effective is set by bit parameter NO: NO:12#7.

7.6 Single Block Execution

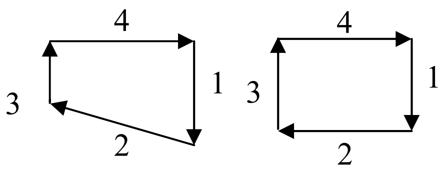
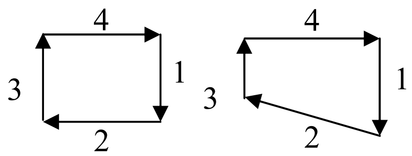
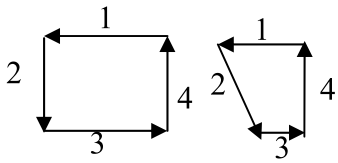
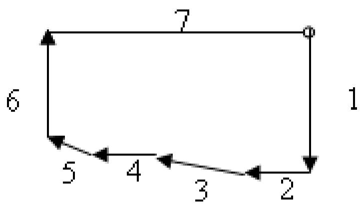
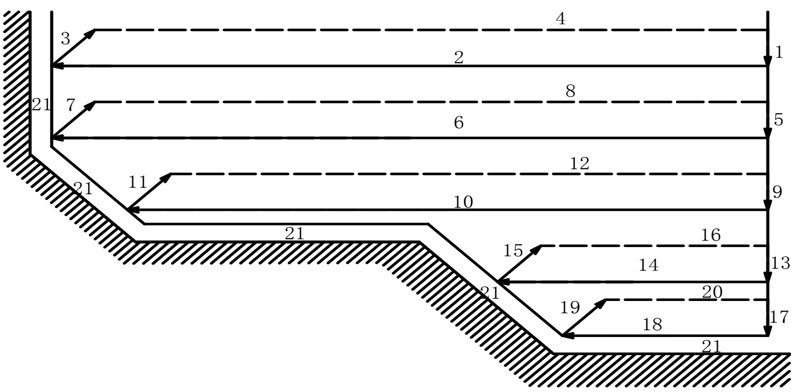
"Single Block" can be selected for checking the execution of a block.

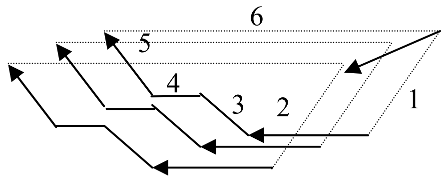
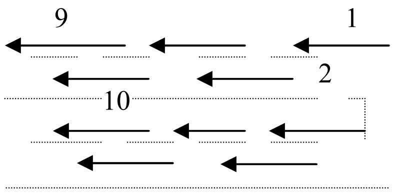
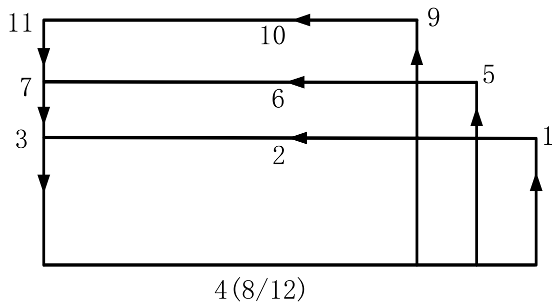
In Auto, or MDI mode, press key  (that the indicator on the key lights up means single block execution state is entered). In single block execution, the system stops after the execution of a single block. Press key  to execute the next block, and perform the operation like this repeatedly till the whole program is executed.

Note 1: In G28 mode, the single block stop can be performed at an intermediate point.

Note 2: In single block ON, execute G90, G92, G70—G75.

Table 7-6-1

G code	Tool path	Explanation
G90		Operation 1～4 as one cycle. Stop after operation 4 ends
G92		Operation 1～4 as one cycle. Stop after operation 4 ends
G94		Operation 1～4 as one cycle. Stop after operation 4 ends
G70		Operation 1～7 as one cycle. Stop after operation 7 ends
G71, G72	 **Note: the drawing is for G71, which is the same as G72.**	Operation 1～4, 5～8, 9～12, 13～16, 17～20 as one cycle. Stop after the cycle ends.

G73		Operation 1～6 as one cycle. Stop after the cycle ends.
G74，G75		Operation 1～10 as one cycle. Stop after the cycle ends.
G76		Operation 1～4，5～8，9～12 as one cycle. Stop after the cycle ends.

7.7　Machine Lock Run

In <AUTO> mode, press key  (that the indicator on the key lights up means the current Machine lock state is entered). In this mode, the axes on the machine do not move, but the position along each axis changes on the display as if the tool were moving. In addition, M, S and T functions can be executed. This function is for checking a program.

7.11 MST Lock Run

In <AUTO> mode, press key  (that the indicator on the panel lights up means MST lock state is entered). In this state, M, S and T codes are not executed. This function is used together with Machine Lock to check a program.

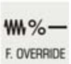
Note: M00, M01, M02, M30, M98, M99 are executed even in MST lock state.

7.11 Feedrate and Rapid Speed Override in Auto Run

In <AUTO> mode, the feedrate and rapid traverse speed can be overridden by the system.

In auto run, press keys  to select the rapid traverse speed with gears Fo, 25％, 50％ and 100％.

In auto run, the feedrate override, which is divided into 21 gears, can be selected by pressing keys 

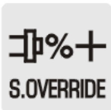


.

Note 1: Value specified by F in feedrate override program
The actual feedrate = Value specified by F X feedrate override
Note 2: The rapid traverse speed overridden by data parameter P88, P89, P90 and rapid override is calculated as follows:
X actual rapid traverse speed = Value specified by P88 X rapid override
The calculation methods for Z axis are the same as that X.

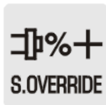
7.11 Spindle Speed Override in Auto Run

In auto run, the spindle speed can be overridden if it is controlled by analog quantity.

The spindle override, which is classified into 8 gears from 50％～120％, can be adjusted by pressing spindle override keys 

, 

in auto mode.

The spindle speed override increases by one gear (10%) till 120％ by pressing key 

each time.

The spindle speed decreases by one gear (10%) by pressing key 

once. When it decreases to 50%, the spindle stops.

7.11 Background Edit in Auto Run

The background edit function during processing is supported in this system.

During the program execution in Auto mode, press key <PROGRAM> to enter the program page, then press soft key【◆PRG】to enter the background edit page, as is shown in Fig.7-11-1:

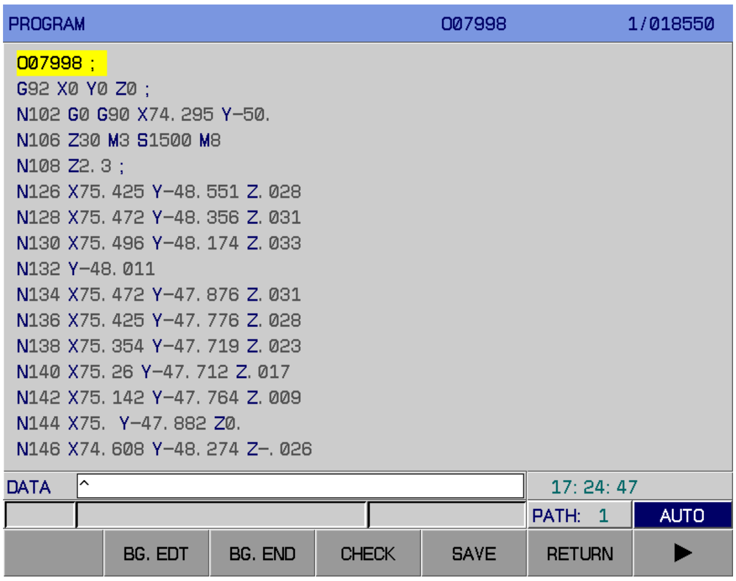


Fig. 7-11-1

Press soft key【B.EDT】to enter the program background edit page. The program editing operation is the same as that in Edit mode (Refer to Chapter 10 Programming Edit Operation). Press soft key【B.END】to save the edited program and exit this page.

In Auto mode, press【CHECK】softy key to check programs.

Note1: It is suggested that the file size in background edit be not more than 3000 lines, otherwise the processing effect will be affected.

Note2: Foreground program can be opened but can not be edited or deleted in background edit.

Note3: No running foreground program can be edited in background edit.

Chapter 8　MDI Operation

Besides the input and modification for parameters and offsets, the MDI operation function is also provided in MDI mode. The codes can be input directly using this function. The data input, parameter and offset modification etc. are described in “CHAPTER 3 PAGE DISPLAY AND DATA MODIFICATION AND SETTING”. This chapter will describe the MDI operation function in MDI mode.

8.1　MDI Block Input

Input in MDI mode：

1. The system can continuously input many block;

The input in 【MDI】type is identical with the program input in Edit mode. See “CHAPTER 10 PROGRAM EDIT” in this manual for details.

As shown in Fig. **8-1-1**：

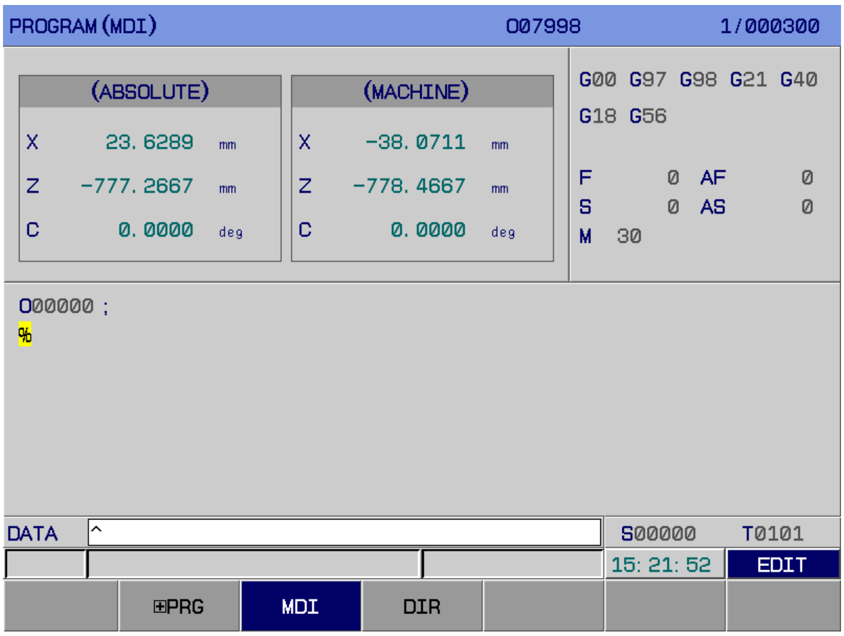


Fig.　8-1-1

8.2　MDI Block Execution and Stop

After the codes are input according to the steps in Section 8.1, press key 

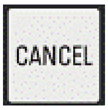
to execute them in MDI mode. During the execution, the code execution can be stopped by pressing key 

.

Note 1: MDI execution must be performed in MDI mode.

Note 2: MDI program editing up to 60 blocks.

8.3 MDI Word Value Modification and Deletion

If a mistake occurs during the input, press key  to cancel it; if a mistake is detected after the input, re-input the contents to replace the wrong ones or press key  to delete all the contents and then input them again.

8.4 Operation Modes Conversion

In Auto, MDI mode, when the control is converted to MDI, DNC, Auto or Edit mode during the program execution, the system stops the execution of the program after the current block is executed.

When the control is switched to Step mode by a dwell during the program execution in Auto, MDI mode, the step interruption is executed (See section 6.2 Step interruption). If the control is switched to MANUAL mode by a dwell, the manual intervention is executed (See Section 4.1.4 Manual Interruption).

When the control is directly switched to Step, MPG, MANUAL or Zero Return mode during the program execution in Auto, MDI, DNC mode, the system executes deceleration and stop.

Chapter 9 Zero Return Operation

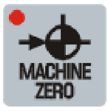
9.1 Concept of Machine Zero (Mechanical Zero)

The machine coordinate system is the inherent coordinate system of the machine. The origin of the machine coordinate system is called machine zero (mechanical zero), which is also called **reference point** in this manual. It is usually fixed at the maximum stroke point of X axis, Y axis and Z axis. This origin is determined as a fixed point after the design, manufacture and adjustment of the machine. As the machine zero is unknown at power-on, the auto or manual machine zero return is usually performed.

9.2 Program Zero Return

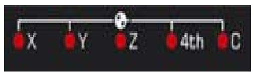
To be convenient programming, the programming personnel uses a workpiece coordinate system(called a part coordinate system) in programming, selects some point as a program zero (the point is specified by G50, called a program zero) to create a workpiece coordinate system.

9.2.1 Steps for Machine Zero Return

1. Press 

to enter Machine Zero Return mode, then "machine zero return" is displayed at the lower right corner of the LCD screen.

2. Select axis X, or Z for machine zero return.

3. The worktable moves along the program zero direction. When it returns to the program zero, the coordinate axis stops move, the zero return indicator 

is OFF.

Note 1：G50 must be executed after the system is turned on, otherwise, #0097 alarm occurs.

Note 2：The program zero return is executed when the machine lock is valid, the absolute coordinates return to the position set by G50 but the machine coordinates do not change.

Note 3：After NO：6#5 is set to 0，a program commands G28 zero return because the check stroke block is equivalent to the manual machine zero return.

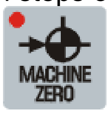
9.3 Bus Servo Zero Return Function Setting

There are two zero return methods when the system is allocated with a bus servo: common zero return, absolute setting zero. The two setting methods are introduced below:

9.3.1 Common Zero Return

No：0#0=1，the system executes a common zero return, select the zero return mode of signal per rotaton or the zero return without signal per rotation. The zero return mode is used to the system matched with GE2000 series incremental mode. Each axis' zero return is valid in zero return mode.

Operation steps of bus servo machine zero return:

1. Press 

to enter Machine Zero Return mode, then "machine zero return" is displayed at the lower right corner of the LCD screen;
2. Select axis X, or Z for machine zero return, the direction of which is set by bit parameter No.:7#0～N0:7#4;
3. When it moves towards the machine zero, the machine traverses rapidly (traverse speed set by data parameter No.100～No.104) before the deceleration point is reached. After the deceleration switch is touched, P342~P346 sets each axis' zero return speed. After it is away from the block, it moves to the machine zero point (i.e. reference point) at a speed of FL(set by data parameter P099). As the machine zero is reached, the coordinate axis movement stops and the Machine Zero indicator lights up.

9.3.2 Absolute Zero Setting and Zero Return

Set No：0#0=1, No：20#7=1, No：20#6=1: the system is allocated with GE2000 series version. The user manually moves each axis to its machine zero. Then in MDI mode, No：21#0=1, set the 1st axis' zero point position; No：21#1=1: set the 2nd axis zero point position; No：21#2=1: set the 3rd axis' zero point' position; No：21#3=1: the 4th axis' zero point' position; No：21#4=1: the 5th axis' zero point' position. In zero return mode, each axis' zero return indicator is ON, and the machine zero setting succeeds.

Note：The method to set zero is complicated. Please set zero in the bus allocation page.

The zero return mode can be directly set in **【 BUS ALLOCATION】**. See Section 3.3.5 Bus Sero Parameter Display, Modification, Setting.

Example:

For the absolute encoder sets the zero, the user can set the zero point position by the motor feedback's absolute position. Set #20.7=1， #20.6=1. See Fig. 9-3-2-1.

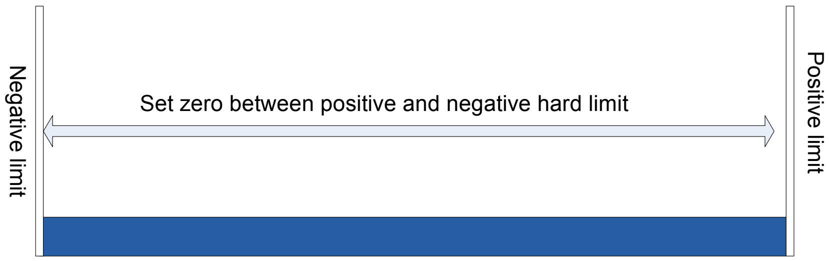


Fig. 9-3-2-1 absolute encoder zero setting

Note 1：The user must not execute the machine zero return when the machine is not installed with a zero return deceleration switch or a machine zero.

Note 2：The corresponding axis' indicator is ON when the machine zero return ends.

Note 3：The zero return indicator is OFF when the correspond axis is not on the machine zero.

Note 4：Refer to the machine manufacturer's user manual about the machine zero (reference point) direction.

Note 5：The user must not modify each axis' zero return direction, feed axis direction and gear ratio after the machine zero is set.

Note 6：Parameters relevant with the machine zero and all machine zero return modes are referred to Section 4.8. .

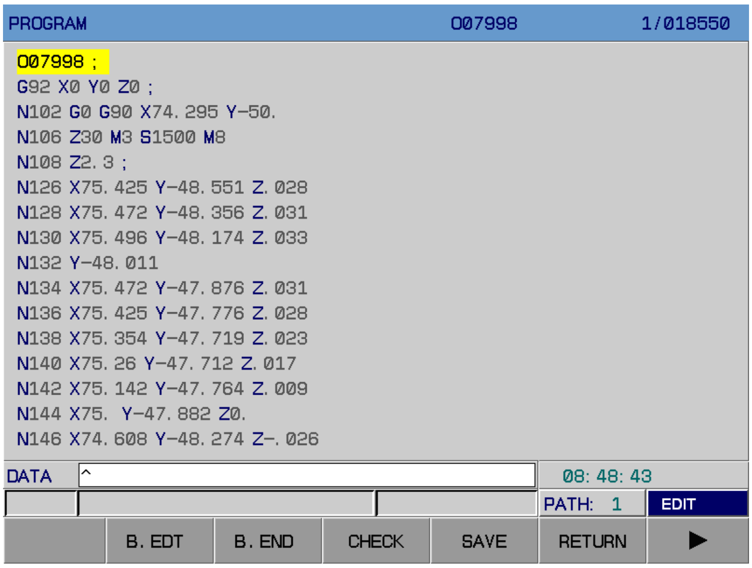
Chapter 10　Edit Operation

10.1　Program Edit

The edit for part programs should be operated in Edit mode. Press key 

to enter Edit mode; Press key 

to enter program page, and press soft key 【⊞ PROGRAM】to enter the program editting and modification page (see Fig. **10-1-1**):



Press 【▶】to enter the next page 

Press 【▶】to enter the next page 

Press 【◀】to return to the last page 

Fig.　10-1-1

The replacement, cut, copy, paste, reset operations, etc. can be done by pressing the corresponding soft keys.

The program switch must be turned on before program editing. See Section 3.3.1 Parameter and program switch page in this manual for its operation.

Note 1: A program contains no more than 100,000 lines.

Note 2: As is shown in fig. 10-1-1, if there is more than 1 sign "/" ahead of a block, the system will skip the block even if the block skip function is not turned on.

Note 3: It is forbidden to switch the control to other mode when the Check function is performed in Auto mode, or unexpected results will occur.

During Check in Auto mode, if there is a sign "/" ahead of a block, the Check function is performed for this block regardless of whether the skip function is ON.

10.1.1　Program Creation

10.1.1.1　Sequence Number Automatic Creation

Set the “AUTO SEQ” to 1 according to the method described in Section **3.3.1.** See fig. 10-1-1-1-1.

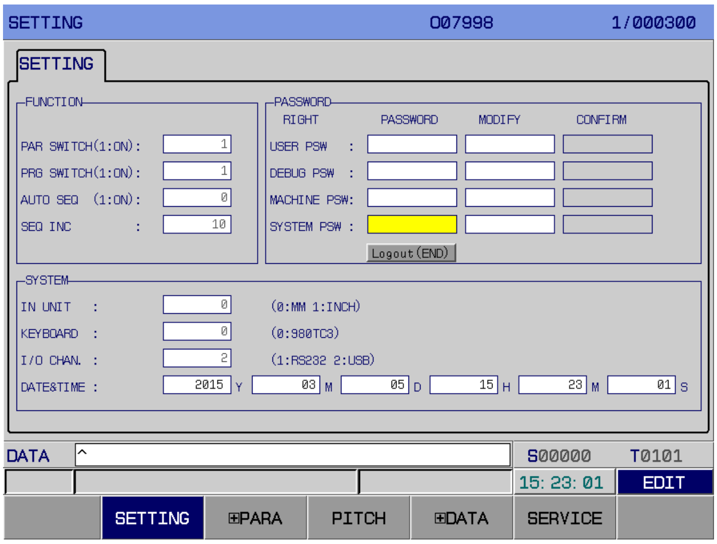


Fig. 10-1-1-1-1

In this way, the sequence number will be automatically inserted into the blocks during program editing. The incremental amount of the sequence number is set by its corresponding parameter.

10.1.1.2　Program Content Input

1. Press key  to enter Edit mode;

2. Press key  to enter program page. See fig. **10-1-1-2-1**:

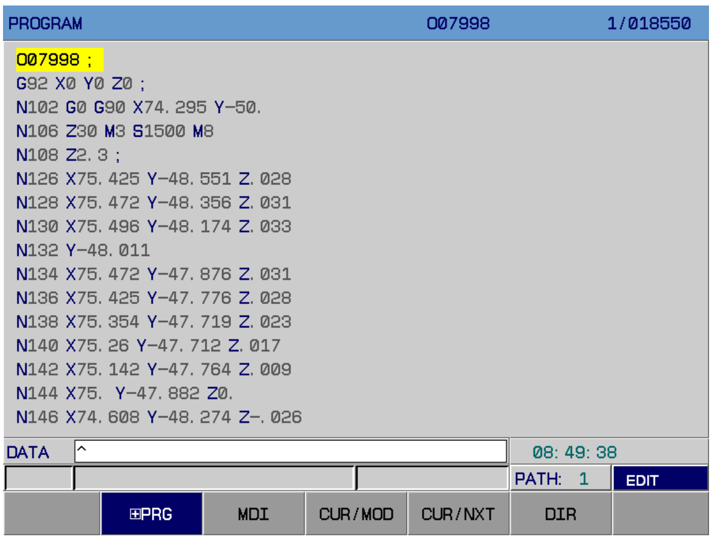


Fig.　10-1-1-2-1

3. Press address key  , and key in numerical keys , , ,  and  in sequence (an example of setting up a program name of O00002 here), then O00002 is displayed behind the DATA column (See Fig. 10-1-1-2-2):

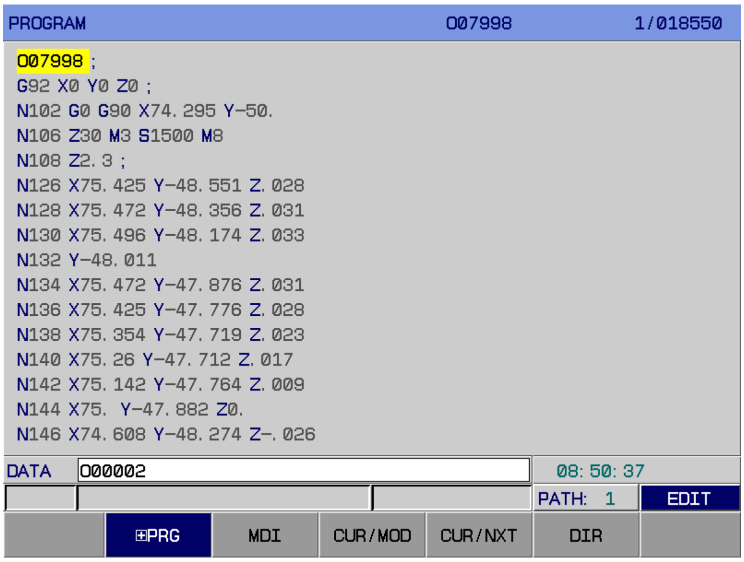


Fig. 10-1-1-2-2

4. Press key  to set up the new program name, as is shown in Fig.10-1-1-2-3:

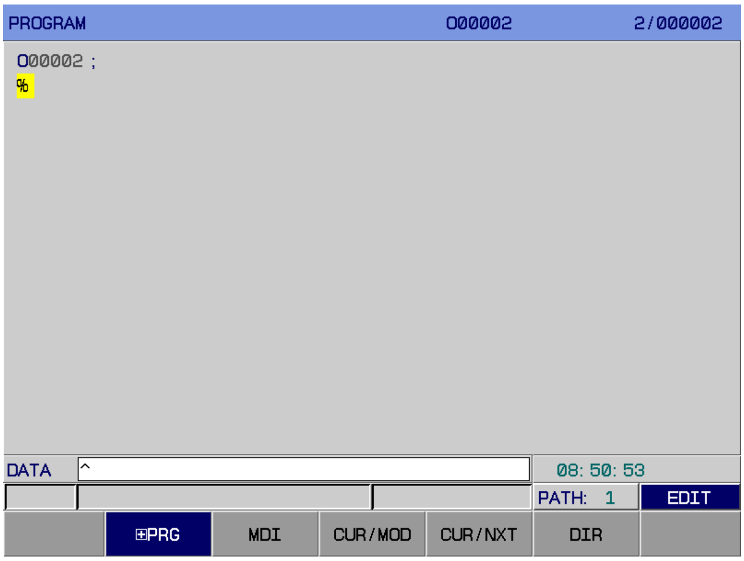
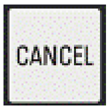


Fig. 10-1-1-2-3

5. Input the written program word by word. After the input, the program will be saved automatically when the control is switched to other operation modes. However, if the control needs to be switched to other pages (e.g.  page), first press key  to save the program and then finish the input of the program.

Note 1: Pure numerical value input is unavailable in Edit mode.

Note 2: If a wrong code word is detected during program inputting, press key 

to cancel the code.

Note 3: No more than 65 characters can be input in one block each time.

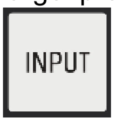
10.1.1.3 Search of Sequence Number, Word and Line Number

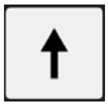
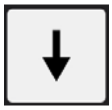
The sequence number search operation is used to search for a sequence number from which the program execution and edit are usually started. Those blocks skipped because of the search have no effect on the CNC state (This means that the data in the skipped blocks such as coordinates, M, S, T and G codes does not affect the CNC coordinates and modal values).

If the execution is started from a block searched in a program, it is required to check the machine and CNC states. The execution can only be performed when both the states are consistent with its corresponding M, S, T codes and coordinate system setting, etc (set in MDI mode).

The word search operation is used to search a specific address word or number, and it is usually used for editing a program.

Steps for the search of sequence number, word and line number in a program:

1. Select mode: <Edit > or <Auto>
2. Look up the target program in 【DIR】page;
3. Press key 

to enter the target program;
4. Key in the word or sequence number to be searched and press key  or  to search for it.
5. When needing to search a line number in a program, press key 

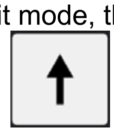
.

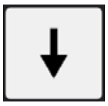
Note 1: The search function is automatically cancelled when the search for sequence number and word is performed to the end of a program.
Note 2: The searching for sequence number, word and line number can be performed in either【AUTO】or【EDIT】mode, but in【AUTO】mode, it can only be performed in the background edit page.

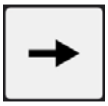
10.1.1.4 Location of the Cursor

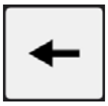
Select Edit mode, then press key 

to display the program.

a) Press key  to move the cursor upward a line, if the column where the cursor is located exceeds the end column of the last line, the cursor moves to the end of the last line.

b) Press key  to move the cursor downward a line. If the column where the cursor is located exceeds the end column of the next line, the cursor moves to the end of the next line.

c) Press key  to move the cursor one column to the right. If it is located at the end of the line, the cursor moves to the beginning of the next line.

d) Press key  to move the cursor one column to the left. If the cursor is at the beginning of the line, it moves to the end of the last line.

e) Press key  to scroll screen upward to move the cursor to the last screen.

f) Press key  to move the screen downward to move the cursor to the next screen.

g) Press key 

to move the cursor to the beginning of the line where it is located.

h) Press keys 

\+ 

to return the cursor to the beginning of the program.

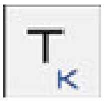
i) Press key 

to move the cursor to the end of the line where it is located.

j) Press keys 

\+ 

to move the cursor to the end of the program.

k) Double click 

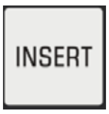
to switch the two letters.

10.1.1.5　Insertion, Deletion and Modification of a Word

Select <EDIT> mode, press key 

to display the program, then locate the cursor to the position to be edited.

1. Word insertion

After inputting the data, press key 

to insert the data to the left of the cursor.

2．Word deletion

Locate the cursor to the word to be deleted, press key DELETE to delete the word where the cursor is located.

3. Word modification

Move the cursor to the place to be modified, input the new contents, then press key ALTER to replace the old contents by the new ones.

10.1.1.6 Single Block Deletion

Select <EDIT> mode, then press key PROGRAM to display the program. Locate the cursor to the beginning of the block to be deleted. Press keys N + DELETE to delete the block where the cursor is located.

Note: Regardless of whether there is a sequence number in the block, the user can press key N to delete it (The cursor should be located at the beginning of the line).

10.1.1.7 Deletion of Many Blocks

Blocks deletion from the current displayed word to the block of which the sequence number is specified.

<u>N100</u> X100.0 M03 S2000; ..... N2233 S02 ; N 2300 M30 ;

Cursor current position

Area to be deleted

Fig.10-1-1-7-1

Select <EDIT> mode, press key PROGRAM to display the program. Locate the cursor to the beginning of the target position to be deleted (as the position of word N100 in the figure above), then key in the last word of the many blocks to be deleted, e.g. **S02** (See Fig.10-1-1-7-1), finally press key DELETE, "the system prompts: press DELETE again to execute deletion". Press DELETE on the panel to delete the blocks from the current cursor location to the address specified.

Note 1: Up to 100,000 lines of blocks can be deleted.
Note 2: If the last word to be deleted occurs many times in a program, the system will delete the blocks till the word nearest to the cursor location.
Note 3: When use N+ sequence numbers to delete program blocks, make sure that the initial start places of

II Operation

the N+ sequence numbers locate before the first lines of the blocks.

10.1.1.8 Deletion of Words

Words deletion from the current displayed word to the one specified.

N100 X100.0 M03 S2000 ; G01 X50.0 Y100.0 N2233 S02 ;

Cursor current positon

Area to be deleted

Fig. 10-1-1-8-1

Select <EDIT> mode, press key PROGRAM to display the program. Locate the cursor to the beginning of the target position to be deleted (as the position of word N100 in the figure above),then key in the last word of the words to be deleted, e.g. **Y100.0** (See Fig.**10-1-1-8-1**), finally press key DELETE, "the system prompts: press DELETE again to execute deletion". Press DELETE on the panel to delete the blocks from the current cursor location to the address specified.

10.1.2 Deletion of a Single Program

The steps for deleting a program in memory are as follows:

a) Select <EDIT> mode;

b) Enter program display page. There are two ways to delete a program:

1. Key in address key O P; key in the program name (e.g. for program O0002, key in number Keys 0 *, 0 *, 0 *, 2 #); press key DELETE, the corresponding program in memory will be deleted.

2. Select【DIR】subpage in program page, and select the program name to be deleted by moving the cursor, then press key DELETE. Here, "Delete the current file?" is prompted on the system state column, press key INPUT again, then "Deletion succeeded" is prompted and the program selected is deleted.

Note: If there is only one program file, by pressing key Delete, its name will be changed to O00001 first and then the contents be deleted in Edit (DIR) page regardless of whether it is O00001 or not; if there are multiple program files, the contents of program O00001 as well as its program name are deleted.

10.1.3 Deletion of all Programs

The steps for deleting all programs in memory are as follows:

a) Select <EDIT> mode;
b) Enter the program page;
c) Key in address O P;
d) Key in address keys − +, 9), 9), 9), 9), 9) in sequence;
e) Press key DELETE to delete all the programs saved in memory.

10.1.4 Copy of a Program

Steps for copying the current program and saving it with a new name:

a) Select <EDIT> mode;
b) Enter the program page; select the program to be copied using the cursor in【DIR】subpage, and press key INPUT to enter the program display page;
c) Press address key O P, and input the new program name;
d) Press soft key【COPY】to finish the file copying and enter the edit page for the new program;
e) Return to【DIR】can view the new copied program name.

The copy of a program can also be done in the program edit page (shown in Fig. 10-1-1):

1. Press address key O P and key in the new program number;
2. Press soft key【COPY】to finish the file copying and enter the edit page for the new program.
3. Return to 【DIR】page to view the new copied program name.

10.1.5 Copy and Paste of Blocks

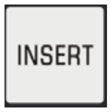
Steps for copying and pasting blocks:

a）Locate the cursor to the beginning of the blocks to be copied;

b）Key in the last character of the blocks to be copied;

c）Press keys SHIFT + SAVE, the blocks from the cursor to the character keyed in will be copied.

d）Locate the cursor to the position to be pasted, press keys 

 + 

 or soft key 【PASTE】 to complete the paste.

The copy and paste of the blocks can also be done in the program edit page (see fig. 10-1-1):

1. Locate the cursor to the beginning of the blocks to be copied;
2. Key in the last character of the blocks to be copied;
3. Press soft key【COPY】to finish copying the blocks from the cursor to the character keyed in.
4. Locate the cursor to the position to be pasted, press soft key 【PASTE】to complete the paste.

Note 1: If the last character keyed in occurs many times in the program, the system will copy the blocks till the word nearest to the cursor location.

Note 2: If the blocks are copied with method N+ sequence number, the blocks from the cursor to the N + sequence number are copied. The N+ sequence number must be at the beginning of the block, or the copy will fail.

Note 3: 10,000 lines of blocks can be copied at most.

10.1.6　Cut and Paste of Blocks

Steps for cutting blocks are as follows:

a） Enter the program edit page (as Fig.10-1-1);

b） Locate the cursor to the beginning of the block to be cut;

c） Key in the last character of the block to be cut;

d） Press soft key【CUT】to cut the block into clipboard.

e) Locate the cursor to the position to be pasted, and press soft key【PASTE】to finish block pasting.

Note 1: If the last character keyed in occurs many times in the program, the system will cut the blocks from the cursor to the word nearest to the cursor.

Note 2: If the blocks are cut with method N+ sequence number, the blocks from the cursor to the N sequence number are cut.

Note 3: in the EDIT, if the program name is in the same block with program content, the system can copy the words after the program name but can not cut them.

10.1.7　Block Replacement

Steps for replacing a block are as follows:

a）Enter the program edit page(Fig.10-1-1);

b）Locate the cursor to the character to be replaced;

c）Key in the new character;

d）Press soft key【REPLACE】to replace the character where the cursor is located as well as other identical characters in the block by the new one.

Note: This replacement operation is only for characters, but not for an entire block.

10.1.8　Rename of a Program

Step for renaming the current program to another one:

a) Select <EDIT> mode;

b) Enter the program page（the cursor specifies a program name with the cursor）;

c) Press address key 

 to key in the new name;

d) Press key 

to complete the renaming.

10.2 Program management

10.2.1 Program Directory Search

Press key 

, then press soft key【DIR】to enter the program directory page（See Fig.10-2-1-1）:

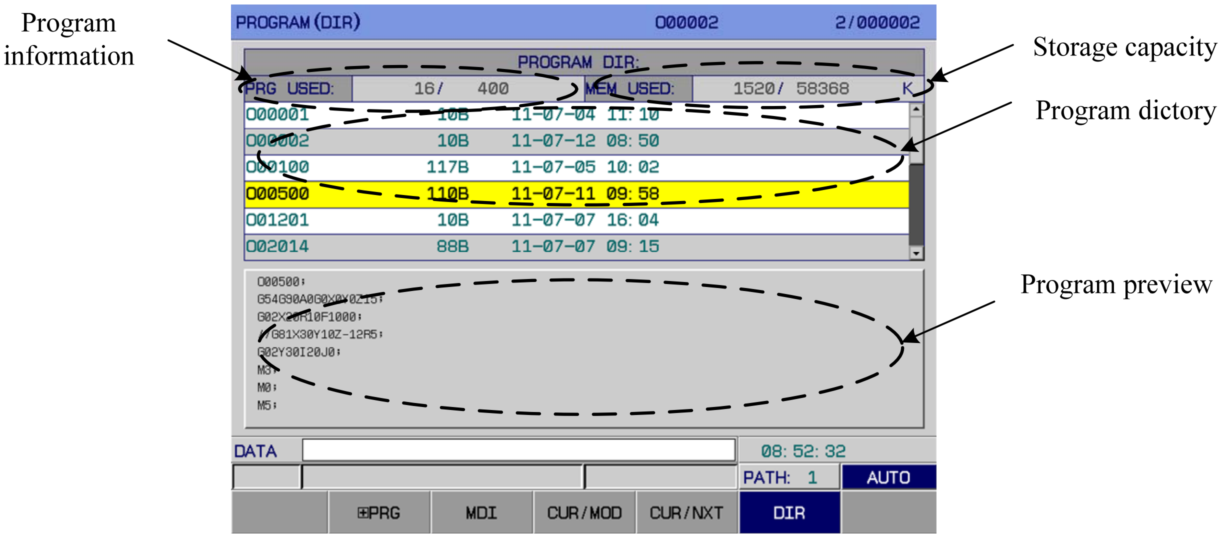


Fig. 10-2-1-1

1) Open a program

Open a specified program: O+ sequence number+ key ENTER (or key EOB), or sequence number + key ENTER (or key EOB)

In Edit mode, if the sequence number input does not exist, a new program will be created.

2) Deletion of a program:

1. In Edit mode, press key DEL to delete the program where cursor is located.
2. In Edit mode, press O+ sequence number + DEL, or sequence number + DEL

10.2.2 Number of Stored Programs

Not more than 400 programs can be stored in this system. The number of the stored programs can be viewed in the program directory page (program information) in Fig. **10.2.1.**

10.2.3 Storage Capacity

The storage capacity can be viewed in the program directory page in Fig. **10.2.1.**

10.2.4 Viewing of Program List

One program directory page can display 6 CNC program names at most. If there are more than 6 names, it is unavailable to display them all in one page. Here, you can press the PAGE key to display the remaining names on the next page. If the Page key is pressed repeatedly, all the CNC program names are displayed circularly on LCD.

10.2.5 Program Lock

The program switch is provided in this system to prevent the user programs from being modified by unauthorized personnel. After the program editing, turn off the program switch to lock the program, thus disabling the program edit. See Section 3.3.1 for details.

Chapter 11 System Communication

This system supports RS232, USB communication interface, separately communicates with PC or U disk to realize data transmission.

11.1 GSKComm Introduction

GSKComm is a communication management software specially for users, supporting a serial port connection mode. Its function includes file load between PC and the CNC, and file edit, with characteristics of easy operation, high communication efficiency and reliability. The software is for Windows interface, running in Win98, WinMe, WinXP and Win2000.

Run program GSK980TC3Comm.exe directly. The page is as follows:

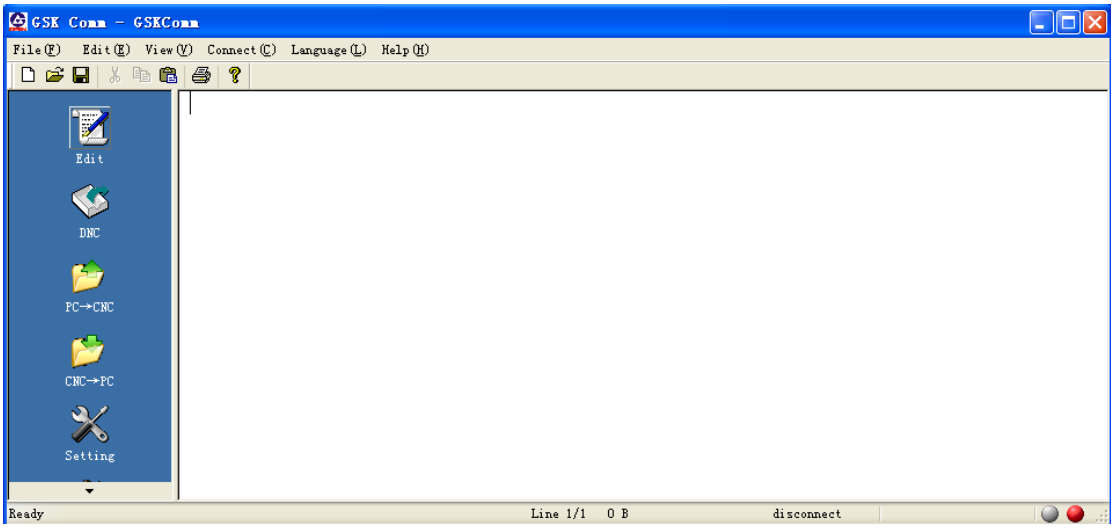


Fig. 11-1-1

11.1.1 Functions

1. File menu
   The file menu involves functions of New, Open, Save, Print and Print setting and the latest file list etc.
2. Edit menu
   The edit menu involves functions such as Cut, Copy, Paste, Undo, Find and Replace.
3. View menu
   Display the tool bar and status bar.
4. Connect menus
   It is mainly used for opening and setting the serial port.
5. Main menu
   Include file edit, DNC transmission function, PC—CNC transmission function, CNC—PC transmission function, software and serial port setting, user management and sum. View theses menu contents by pressing the main menu's black small triangle.
6. Help menu
   It is used to view the software version.

Note: DNC transmission function, user management, statistics are not provided.

11.1.2 Edit Operation

Click the main menu 

to enter the file edit interface. The edit operation makes the user create a new part program file or open the existed part program file to execute edit.

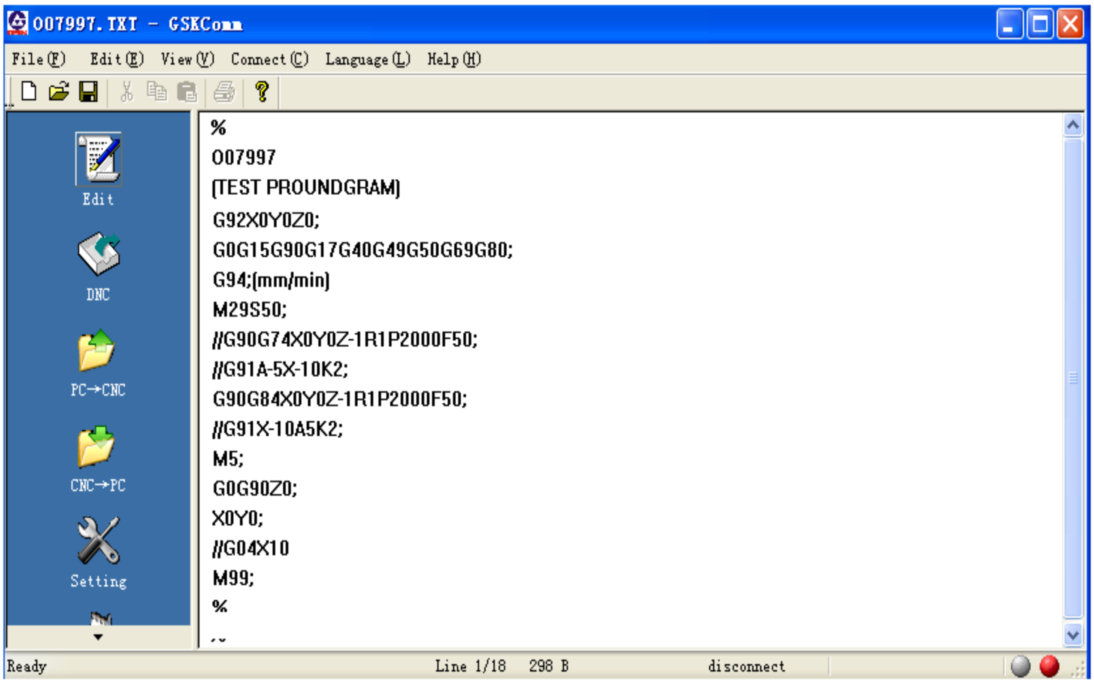
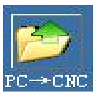


Fig.11-1-2-1

11.1.3 Sending a File（PC—CNC）

Click the main menu 

to enter the sending file interface. The user clicks the right mouse button on the shortcut menu at the right side of “Sending file” interface or the file display column in the middle of the interface, selects a corresponding operation or directly selects it to send it after an environmental menu pops up.

The【Additive…】function can add more files. The user can add one or more files one time. When the additive file name does not meet rules or the file exceeds 4M, the file’s list item is displayed to the red, and the 2nd column is displayed to “×”. When the file name and file size is up to specifications, the 2nd column is displayed to “√”（as shown in Fig. 11-1-3-1）.

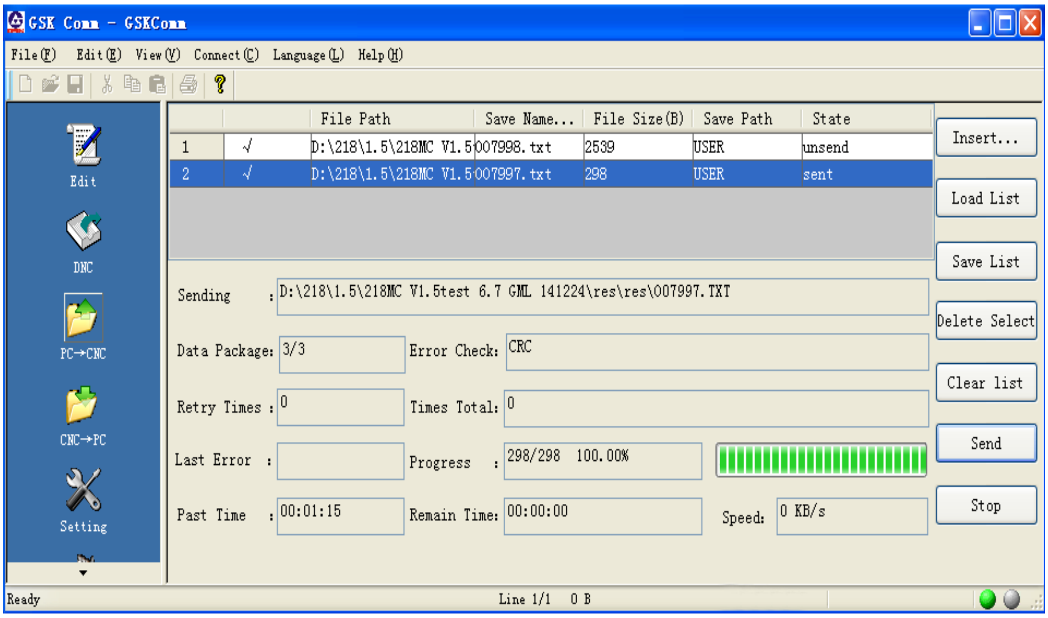


Fig. 11-1-3-1

The user can add the saved file by**【Load file list】**; save the current file list by**【Save list】**; select to delete one or more files one time, and delete the selected list item from PC—CNC listed files; clear the whole PC—CNC file list by **【Clear list】**; send the selected file to the CNC by **【Send】**; terminate the transmission being executed by **【Stop】**. The user sorts the sent file list by clicking the sending filelist's list head. After sorting, a black small triangle symbol appears on the list head, the upper triangular symbol means ascending sequence and the lower triangular symbol means decending sequence (as shown in Fig. 11-1-3-2）.

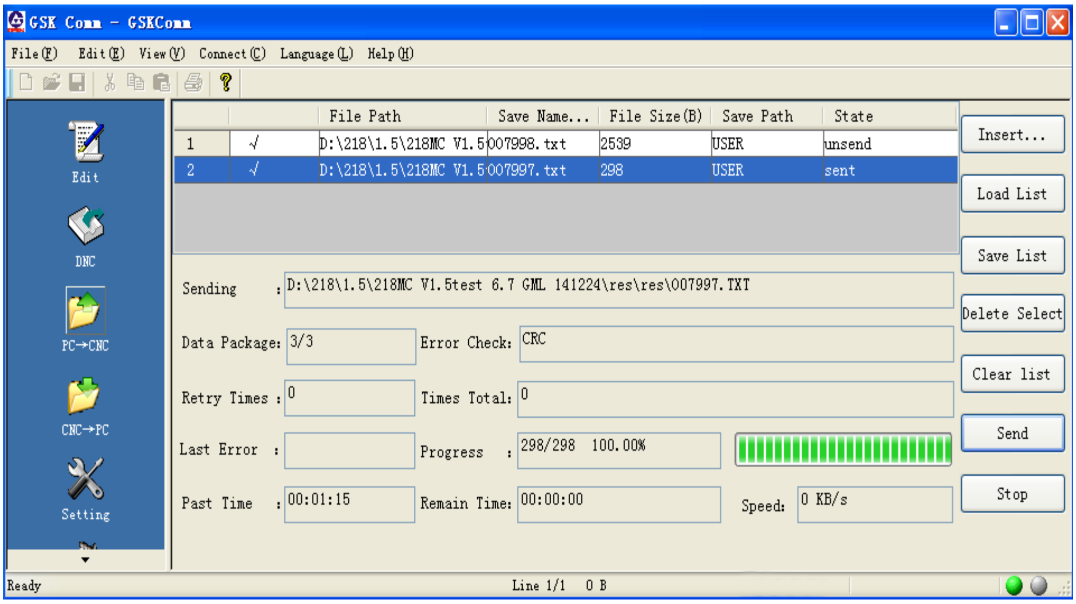


Fig. 11-1-3-2

Double click the required list item and a dialog box as shown in Fig. 11-1-3-3 pops up to modify a file path, a file name stored in the CNC, or a storage area.。

Appendix

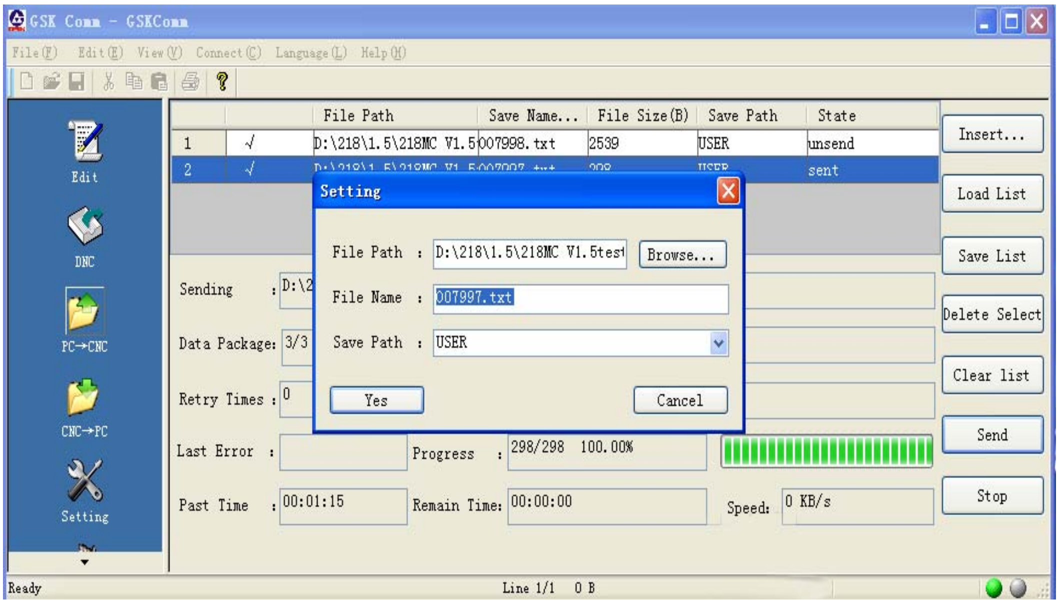


Fig. 11-1-3-3

When the file name which file is to be sent by the user is the same as that of some in the CNC system, a dialog box as shown in Fig11-1-3-4 pops up in the course of sending, the user can press “Yes” to directly shade it or press “No” to rename the file, or press “Cancel” to jump over the file.

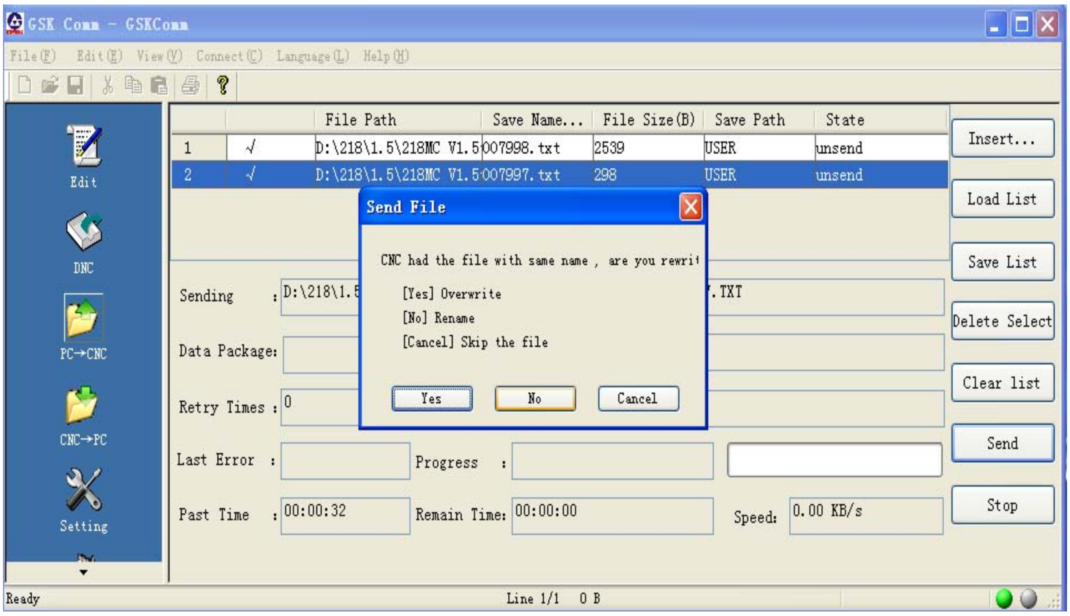


Fig. 11-1-3-4

11.1.4 Receiving Files（CNC—PC）

Receive the file list from the CNC by【**Receiving CNC list**】; delete the selected list item from the received file list by【**Delete form only the list**】; delete the selected file from the file list and simultaneously delete from the CNC; a dialog box (as shown in Fig. 11-1-4-1) pops up to select a position to store the received files by pressing【**Receive**】; stop the file which is being transmitted by【**Stop**】.

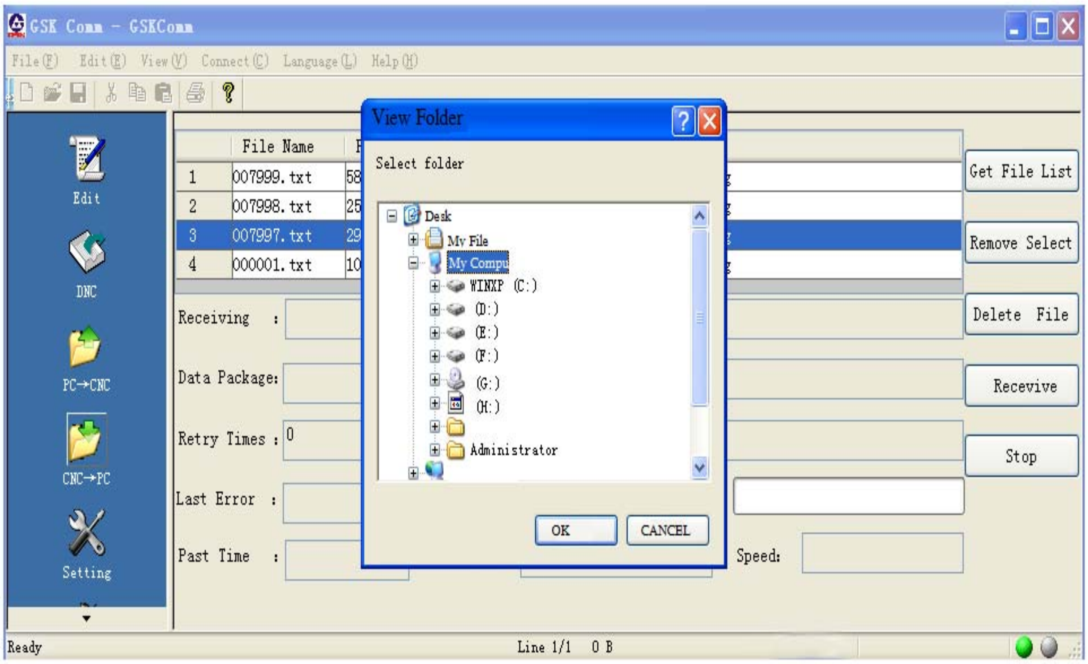


Fig.11-1-4-1

11.1.5 Software and Serial Port Setting

The user can set some softwares and serial ports and the setting page is shown in Fig. 11-1-5-1.

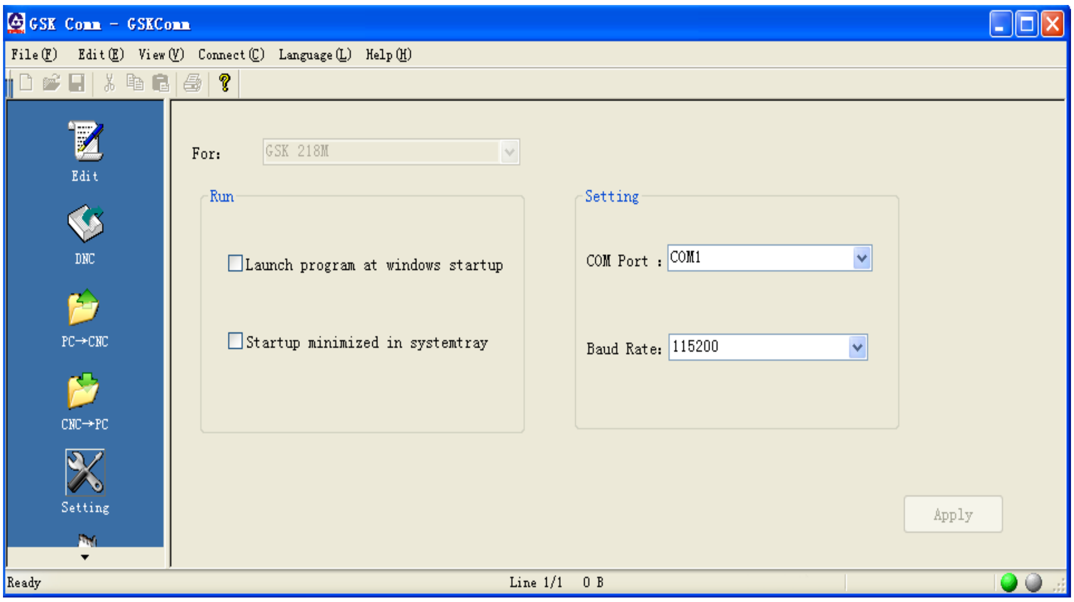


Fig.11-1-5-1

For the program start setting, the user can set whether the system automatically runs the software when the system is turned on and whether the software automatically minimizes at the lower right corner of the screen; for the communication setting, the user can select a serial port to set its baud rate. (after setting, press "Application", otherwise, the setting does not take effect.)

Note: when the program runs, automatically starting the server is not provided.

11.2 Serial communication

11.2.1 Preparations for Serial Port Communication

1. The PC serial port (COM port) is connected with the CNC RS232 interface by a serial port line.
2. Open the PC's GSK Comm software.
3、GSK Comm software setting:

（1） Baud rate setting:

Click the software "Setting" to enter the setting page, and then execute the serial port communication setting;

Port selection: select a port for communication by the drop-down menu "Serial port number" (the selected PC port is automatically identified by the software);

Baud rate setting: select a baud rate by the drop-down menu "Baud rate" to make the baud rate of the PC be the same as that of the CNC communication. The delivery standard setting: the baud rate selects 115200 when the data transmits.

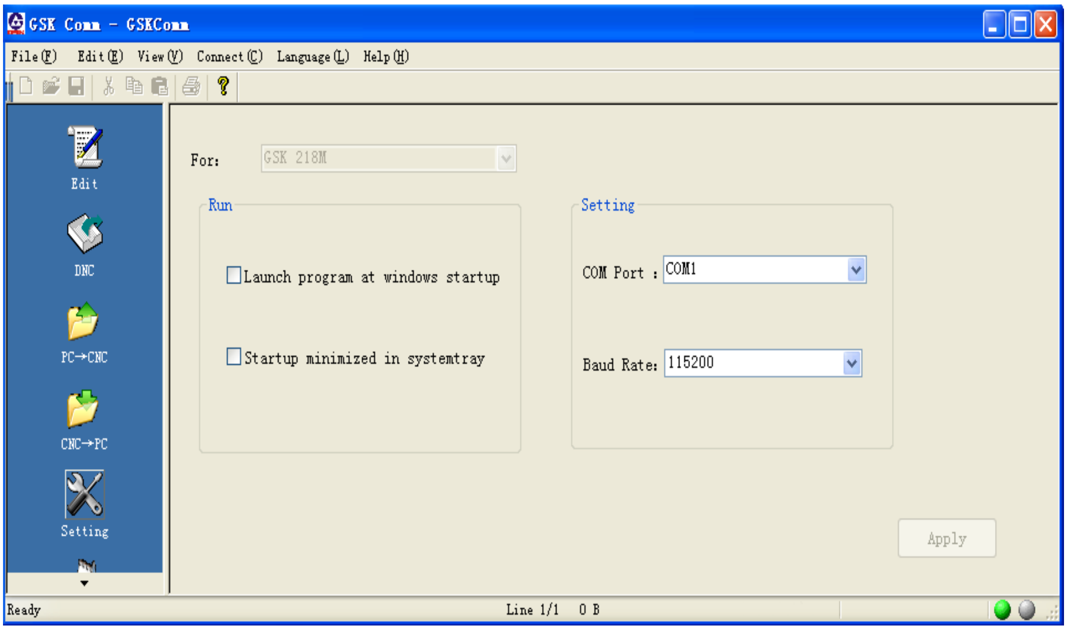


Fig. 11-2-1-1

（2） Click "Connection" menu，select "Via serial port". When the serial port is successfully opened, the status bar displays "Serial port open", and the small icon at lower right corner becomes "Green" and "Grey". But such means the local serial port has been opened, and does not means it is connected with the CNC system.

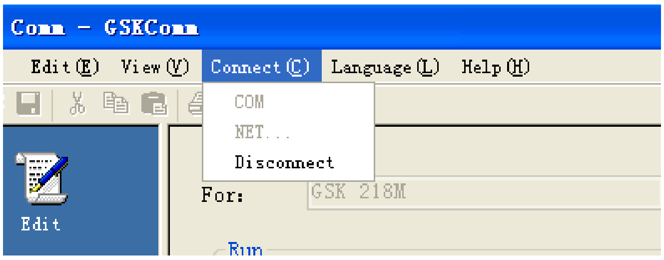


Fig. 11-2-1-2

Appendix

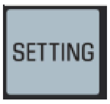
（3）　Click “Connection” menu，select “Disconnect” to disconnect the CNC system.

Note: the communication software is connected with the CNC by a serial port.

11.2.2　Serial Port Data Transmission

Steps are shown below:

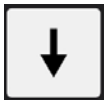
1) Select <MDI> mode;

2) Press key 

to enter setting page, set the I/O channel to 1.

3）Baud rate selects 115200 during data transmission.

4）Press soft key【CNC SET】to input corresponding password authority. For detail please see to Section 3.3.1.1 Setting and Modification for Password Authority.

5)Press key【DATA】to enter SETTING (DATA DEAL) page, then press key  or  to move the cursor to <**CNC Part Program**>.

A.　Data output（CNC→PC）

1. Press system soft key 【OUTPUT】, then the system prompts “transfer waiting”

2. Click button 

on GSK Com communication software to enter the “Receive Files” page.

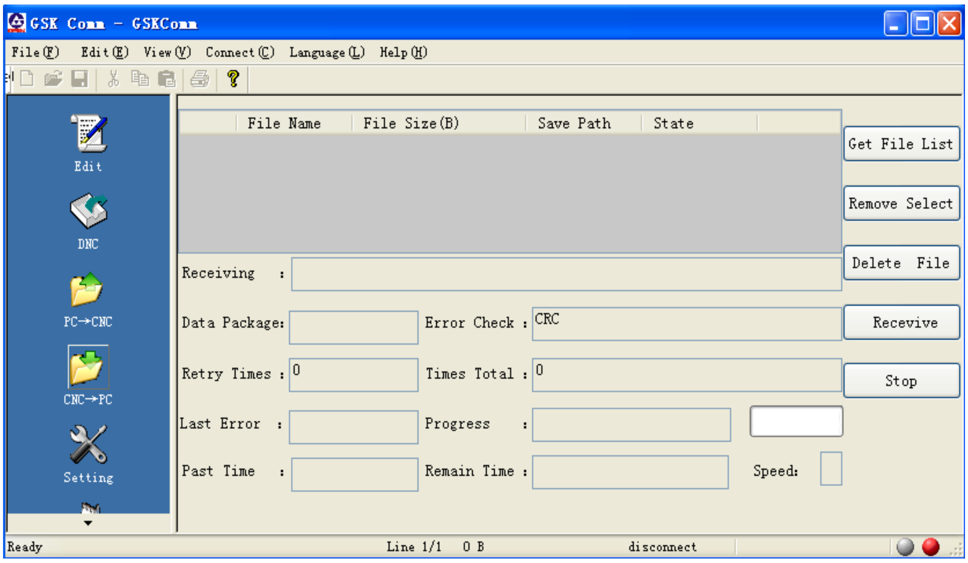


Fig. 11-2-2-1

3. Click button 

to obtain the CNC file list, which is shown in Fig. 11-2-2-2:

	File Name	File Size(B)	Save Path	State
1	007999.txt	581364	user	Not receiving
2	007998.txt	2539	user	Not receiving
3	007997.txt	298	user	Received
4	000001.txt	10	user	Not receiving

Fig. 11-2-2-2

4. Select the file (or many files) to be received, then press button  to start the file receiving, which is shown is Fig. 11-2-2-3.

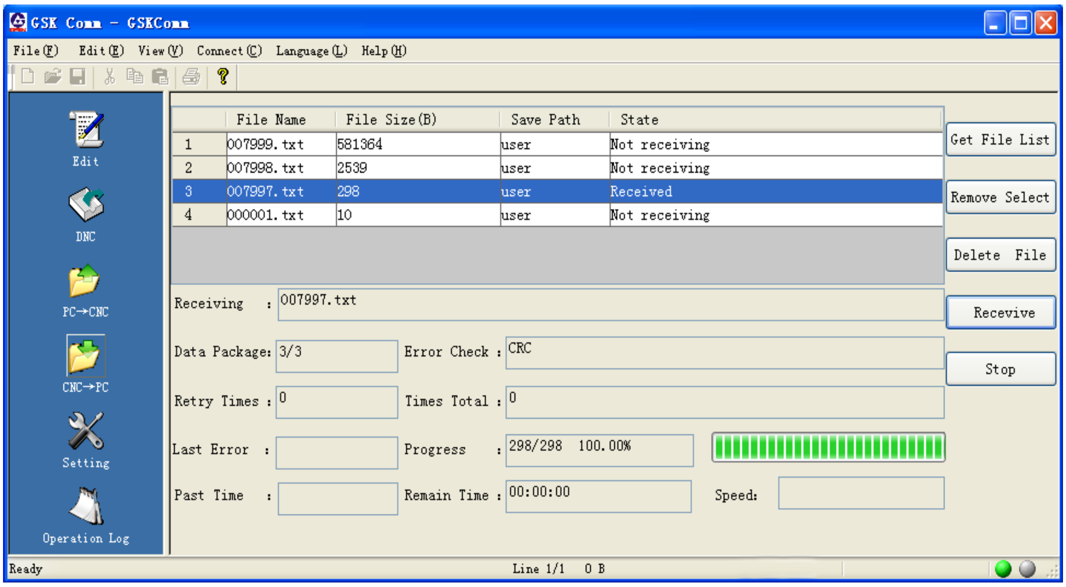


Fig. 11-2-2-3

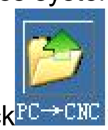
5. After the file receiving, the status bar of the dialog displays "Received", which is shown in Fig. 11-2-2-4.

	File Name	File Size(B)	Save Path	State
1	007999.txt	581364	user	Not receiving
2	007998.txt	2539	user	Not receiving
3	007997.txt	298	user	Received
4	000001.txt	10	user	Not receiving

Fig. 11-2-2-4

A. Data input（PC→CNC）

1. Press system soft key【INPUT】, then the system prompts "input waiting".

2. Click  in GSK Comm software to enter the file sending page.

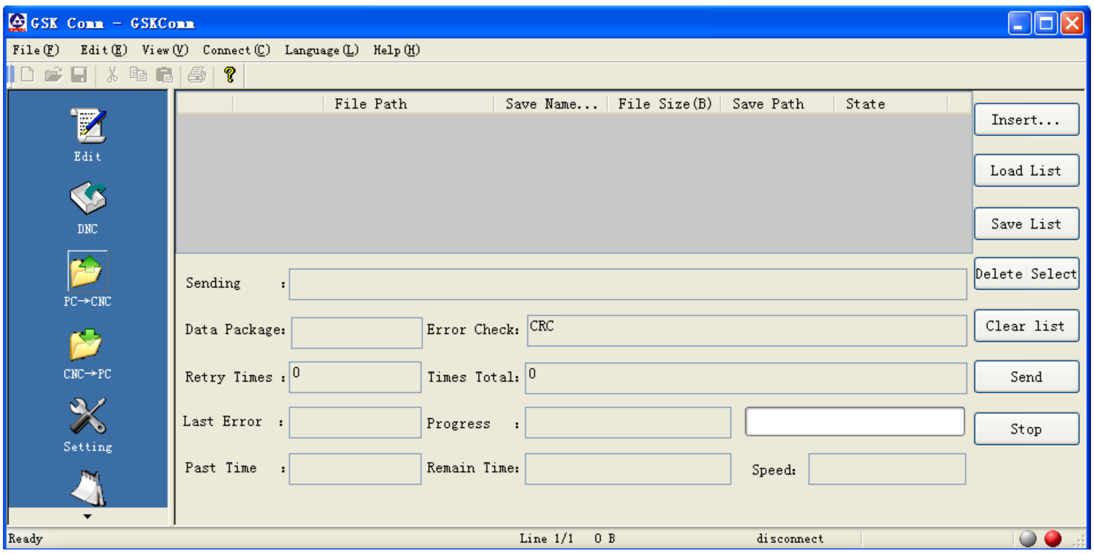


Fig. 11-2-2-5

Appendix

3. Click button Add File , and add the required file to send to the CNC, which is shown in Fig. 11-2-2-6.

		File Path	Save Name...	File ...	Save Path	State
1	√	D:\218\1.5\218MC V1.5	081013.txt	72	USER	unsend
2	√	D:\218\1.5\218MC V1.5	081006.txt	159	USER	unsend
3	√	D:\218\1.5\218MC V1.5	007997.txt	298	USER	unsend
4	√	D:\218\1.5\218MC V1.5	081011.txt	634	USER	unsend
5	√	D:\218\1.5\218MC V1.5	081012.txt	959	USER	unsend
6	√	D:\218\1.5\218MC V1.5	081019.txt	968	USER	unsend

Fig. 11-2-2-6

4. Double click the sent file list item, modify the file path, the CNC file name stored in the CNC or storage area.

Select the user partition when sending CNC part programs and custom macro programs; select the system partition when sending files such ladder (PLC), parameters (PLC), system parameter values, tool offset values, pitch offset values and system macro variables.

5. After selecting the partition, select the file (or multiple files) to be sent, and click button Send to start the file sending, which is shown in Fig. 11-2-2-7.

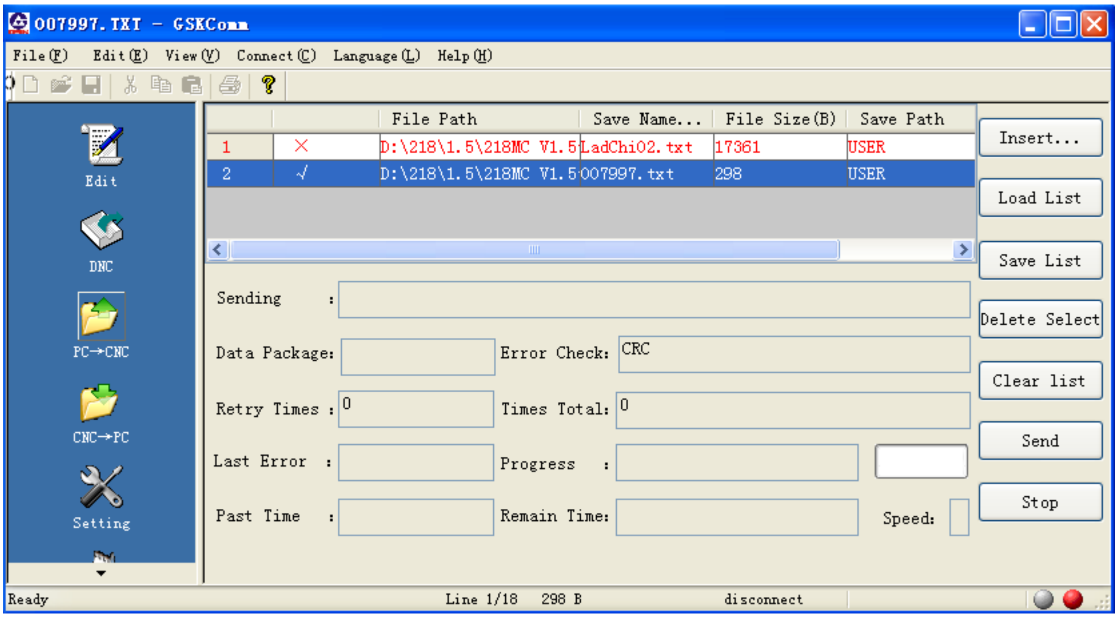


Fig. 11-2-2-7

6. After sending the file/files, “Sent” is displayed in the dialog, which is shown in Fig. 11-2-2-8.

		File Path	Save Name...	File Size(B)	Save Path	State
1	√	D:\218\1.5\218MC V1.5	007998.txt	2539	USER	unsend
2	√	D:\218\1.5\218MC V1.5	007997.txt	298	USER	sent

Fig. 11-2-2-8

Note 1: The other function’s use instructions in the sending file page are referred to Section 11.1.4 PC—CNC sending files, and The other function’s use instructions in the receiving file page are referred to Section 11.1.4 CNC—PC receiving files.

Note 2：Make sure the baudrate is correctly set and the serial line is reliably connected before data transmission.

Note 3：It is forbidden to switch operation modes or pages during data transmission, or critical errors will

Appendix

occur.

Note 4：File LADCHI.TXT is ineffective when transferred to the system unless the power is turned off.**

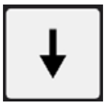
11.3 USB Communication

11.3.1 Overview and Precautions

Precautions:

1. Set I/O channel to 2 in <SETTING> page.
2. The CNC programs should be stored in the root directory of the U disk with file extension .txt, .nc or .CNC, or they cannot be read by the system.
3. Do not pull out USB when it is communicating with system to avoid product damage and unexpected results.
4. After the USB communication is finished, pull out the U disk when its indicator does not flicker (or after a moment is waited for) to ensure the completion of the data transmission.

11.3.2 Operations Steps for USB Part Programs

In <MDI> mode, enter the SETTING (DATA DEAL) page, press direction key  or  to move the cursor to "PART PRGR". Press soft key 【OUTPUT】or【INPUT】to enter the page shown as follows (Fig. 11-3-2-1):

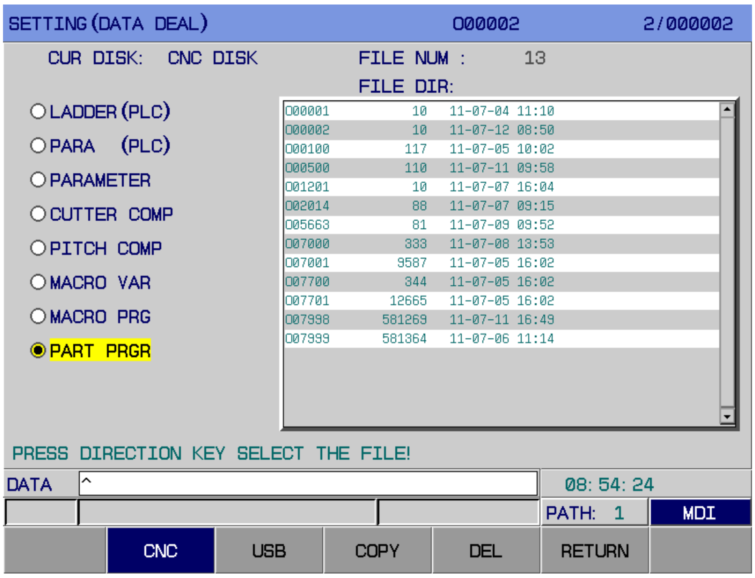
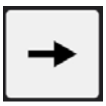
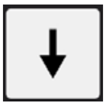


Fig. 11-3-2-1

1. To copy CNC program files to U disk from the system disk:

   a. Press key  to switch the cursor to the file directory.

b. Press key  or  to move the cursor to select the CNC program files to be copied in the system disk.

c. Press soft key 【COPY】, then the systems prompts “COPY TO USB DISC? New Name”, as is shown in fig. 11-3-2-2.

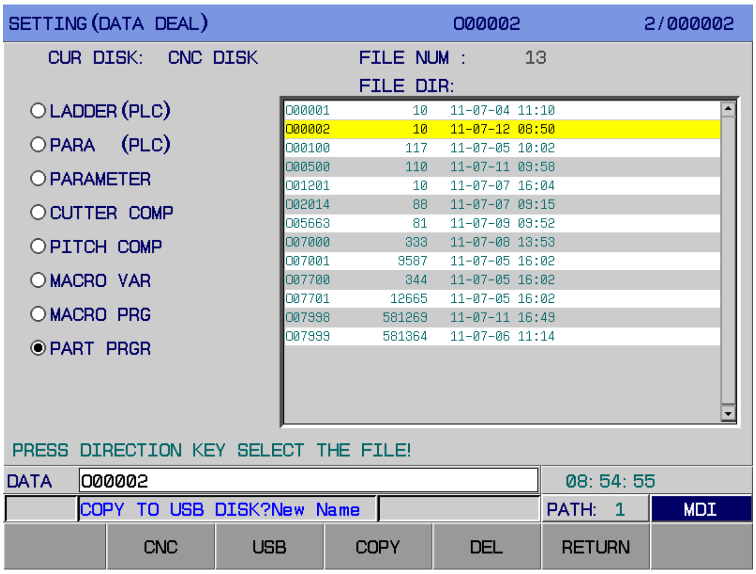


Fig.11-3-2-2

d. If renaming for CNC program files is not required, press key <INPUT> to copy the CNC program files directly.

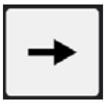
Renaming required, press key <CANCEL> to input the new program number (e.g. O10 or O100), and then press key <INPUT> to copy the CNC program files.

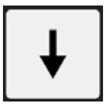
When the program name already exists in the U disk, the system prompts “Please rename the file”. Here, input the new program number (e.g. O10 or O100) .

Then press key <INPUT> to copy the CNC program files.

1. To copy CNC program files to system disk from U disk:

a. Press soft key【USB】to switch to USB file directory page;

b. Press key  to switch the cursor to the file directory.

c. Press key  or  to move the cursor to select the CNC program files to be copied in the U disk.

Press soft key 【COPY】, then the system prompts “COPY TO CNC DISC? New Name”, which is shown in fig. 11-3-2-3:)

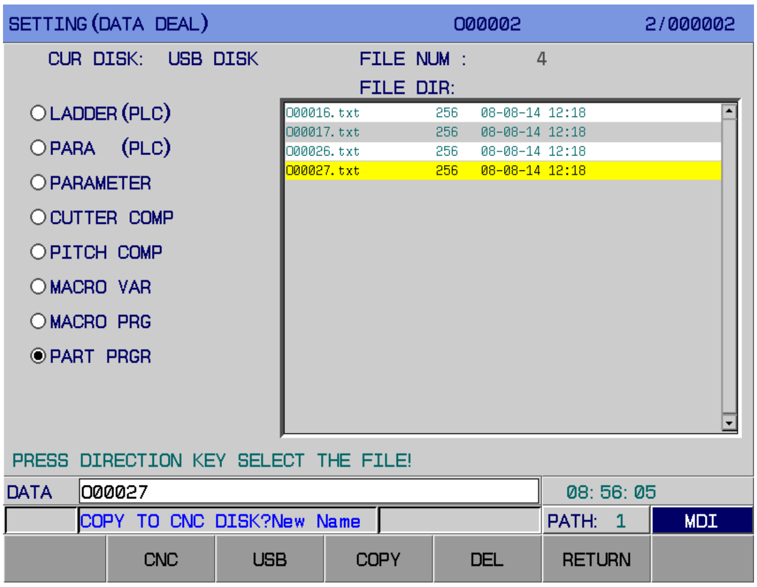


Fig.11-3-2-3

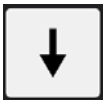
d. If renaming for CNC program files is not required, press key <INPUT> to copy the CNC program files directly.

Renaming required, press key <CANCEL> to input the new program number (e.g. O10 or O100), and then press key <INPUT> to copy the program files.

If the same program name already exists in the system disk, the system prompts "Please rename the file" . Here, input the new program number (e.g. O10 or O100) and then press key <INPUT> to copy the CNC program files.

Note: File LADCHI.TXT is ineffective after transmitted to the system unless the power is turned off.**

2. To delete files from system disk/U disk。

a. Press key  or  to move the cursor to select the CNC program files to be deleted in the system disk/U disk.

b. Press soft key【DEL】, then "DELETE CURRENT FILE?" is prompted at the bottom of the page. Press key <CANCEL> to cancel the file deletion; press key <ENTER> to delete the file.

Appendix

11.3.3 Exiting U Disk Page

1. Pull out the U disk as its indicator does not blink.
2. Press soft key 【RETURN】to return to【DATA】subpage in <SETTING> (DATA DEAL) page.

Appendix

Appendix 1　GSK980TC3 Serial Parameter List

Parameter Explanation:

The parameters are classified into following patterns according to the data type:

- 2 data types and data value range

Data type	Effective data range	Remark
Bit	0 or 1	The default value is given by the CNC, and user can modify the setting by requirement.
Data	Specified according to the parameter range	The default value is given by the CNC, and user can modify the setting by requirement.

1. For bit and axis parameters, the data are comprised by 8 bits with each bit having different meaning.
2. The data value range in above table is common effective range. The specific parameter value range actually differs. See the parameter explanation for details.

Example

（1）Meaning of the bit parameters

Data number	BIT7	BIT6	BIT5	BIT4	BIT3	BIT2	BIT1	BIT0

（（2）Meaning of the data parameters

0	**2**	**1**	
Data number			Data

Note 1: The blank bits in the parameter explanation and the parameter numbers that are displayed on screen but not in parameter list are reserved for further expansion. They must be set to 0.
Note 2: If 0 or 1 of the parameter is not specified with a meaning. It is assumed that: 1 for enable; 0 for disable.
Note 3: If INI is set to 0, in metric input, the parameter setting unit for linear axis is mm, mm/min; that for rotary axis is deg, deg/min.
If INI is set to 1, in inch input, the parameter setting unit for linear axis is inch, inch/min; that for rotary axis is deg, deg/min.

1.　Bit Parameter

System parameter number

0	**0**	**0**		**SVCD**	**SEQ**	**MSP**		**INI**	**INM**	**PBUS**

PBUS　=1: Driver transmission mode is bus type.
　=0: Driver transmission mode is pulse type.

INM　=1: Least increment of linear axis is inch type
　=0: Least increment of linear axis is metric type

If **INI** is set to 0, in metric input, the basic unit for linear axis is mm, mm/min; that for rotary axis is deg, deg/min.

Appendix

If **INI** is set to 1, in inch input, the basic unit for linear axis is inch, inch/min; that for rotary axis is deg, deg/min.

INI =1: inch input.
=0: metric input.

If **INI** is set to 0, in metric input, the basic unit for linear axis is mm, mm/min; that for rotary axis is deg, deg/min.

If **INI** is set to 1, in inch input, the basic unit for linear axis is inch, inch/min; that for rotary axis is deg, deg/min.

MSP =1: Double-spindle control is used.
=0: Double-spindle control is not used.

SEQ =1: Automatic insertion of sequence numbers is performed.
=0: Automatic insertion of sequence numbers is not performed.

SVCD =1: Use the bus servo card.
=0: Do not use the bus servo card.

Standard setting: 0 0 0 0 0 0 0 1

System parameter number

0	0	1				SPPT	SPEP	SPOM	SPT	SBUS	RASA

RASA =1: use an absolute grating ruler.
=0: do not use an absolute grating ruler.

SBUS =1: the spindle driver is in the bus control mode
=0: the spindle driver is in not the bus control mode

SPT =1: I/O control.
=0: frequency conversion or others.

SPOM =1: the spindle speed control signal output selects a pulse train
=0: the spindle speed control signal output selects an analog voltage

SPEP =1: when the bus spindle is used, the spindle encoder feedback interface selects: XS32 encoder interface
=0: when the bus spindle is used, the spindle encoder feedback interface selects: the interface is the same as that of output

SPPT =1: the spindle pulse output mode selects: AB-phase output
=0: the spindle pulse output mode selects: pulse+direction

Standard setting：0 0 0 0 0 0 0 0

System parameter number

0	0	2					DEC5	DEC4	DEC3	DEC2	DEC1

DEC1 =1: Decelerate when the 1st axis reference point returns and the deceleration signal is 1.
=0: Decelerate when the 1st axis reference point returns and the deceleration signal is 0.

DEC2 =1: Decelerate when the 2nd axis reference point returns and the deceleration signal is 1.
=0: Decelerate when the 1st axis reference point returns and the deceleration signal is 0.

DEC3 =1: Decelerate when the 3rd axis reference point returns and the deceleration signal is 1.
=0: Decelerate when the 3rd axis reference point returns and the deceleration

Appendix

signal is 0.

DEC4 =1: Decelerate when the 4th axis reference point returns and the deceleration signal is 1.

=0: Decelerate when the 4th axis reference point returns and the deceleration signal is 0.

DEC5 =1: Decelerate when the 5th axis reference point returns and the deceleration signal is 1.

=0: Decelerate when the 5th axis reference point returns and the deceleration signal is 0.

Standard setting：0 0 0 0 0 0 0 0

System parameter number

0	0	3					DIR5	DIR4	DIR3	DIR2	DIR1

DIR1 =1: the 1st axis feeding direction reverse.
=0: the 1st axis feeding direction.

DIR2 =1: the 2nd axis feeding direction reverse.
=0: the 2nd axis feeding direction.

DIR3 =1: the 3rd axis feeding direction reverse.
=0: the 3rd axis feeding direction.

DIR4 =1: the 4th axis feeding direction reverse.
=0: the 4th axis feeding direction

DIR5 =1: the 5th axis feeding direction reverse
=0: the 5th axis feeding direction.

Standard setting：0 0 0 0 0 0 0 0

System parameter number

0	0	4		SK0							

SK0 =1: it is taken as a signal to input when the skip signal SKIP is 0
=0: it is taken as a signal to input when the skip signal SKIP is 1

Standard setting：0 0 0 0 0 0 0 0

System parameter number

0	0	5		DOUS						ISC	

ISC =1: least code increment 0.0001mm&deg,0.00001inch.
=0: least code increment 0.001mm&deg,0.0001inch.

DOUS =1: the double-drive device uses a grating position.
=0: the double-drive device uses a grating position.

Standard setting：0 0 0 0 0 0 1 0

System parameter number

0	0	6		MAOB	ZPLS	SIOD	SJZ	AZR	JAX	ZMOD	ZRN

ZRN =1: when a reference point is not established, and other codes except form G28 in Auto run mode, an alarm occurs.

=0: when a reference point is not established, and other codes except form G28 in Auto run mode, no alarms occurs.

ZMOD =1: the zero return mode selection: before block.

Appendix

	=0:	the zero return mode selection: after block.
JAX	=1:	execute the manual reference point return and simultaneously control axis: single-axis.
	=0:	execute the manual reference point return and simultaneously control axis: multiple axes.
AZR	=1:	when the reference point is not established, G28: alarm.
	=0：	when the reference point is not established, G28: use the block
SJZ	=1:	the reference point is memorized
	=0：	the reference point is not memorized
SIOD	=1:	the machine zero deceleration signal executes the PLC logic operation.
	=0：	the machine zero deceleration signal is directly read.
ZPLS	=1:	Zero return mode selection: one-revolution signal
	=0:	Zero return mode selection: non-one-revolution signal
MAOB	=1:	Zero return mode selection for non-one-revolution signal: B
	=0:	Zero return mode selection for non-one-revolution signal: A

Standard setting：1 1 1 0 0 0 0 1

System parameter number

0	0	7					**ZMI5**	**ZMI4**	**ZMI3**	**ZMI2**	**ZMI1**

ZMI1 =1: Set the 1st axis direction of reference point return: negative
=0: Set the 1st axis direction of reference point return: positive
ZMI2 =1: Set the 2nd axis direction of reference point return: negative
=0: Set the 2nd axis direction of reference point return: positive
ZMI3 =1: Set the 3rd axis direction of reference point return: negative
=0: Set the 3rd axis direction of reference point return: positive
ZMI4 =1: Set the 4th axis direction of reference point return: negative
=0: Set the 4th axis direction of reference point return: positive
ZMI5 =1: Set the 5th axis direction of reference point return: negative
=0: Set the 5th axis direction of reference point return: positive

Standard setting：0 0 0 0 0 0 0 0

System parameter number

0	0	8					**AXS5**	**AXS4**	**AXS3**	**AXS2**	**AXS1**

AXS1 =1: the 1st axis is set to a rotary axis.
=0: the 1st axis is set to a linear axis.
AXS2 =1: the 2nd axis is set to a rotary axis.
=0: the 2nd axis is set to a linear axis.
AXS3 =1: the 3rd axis is set to a rotary axis.
=0: the 3rd axis is set to a linear axis.
AXS4 =1: the 4th axis is set to a rotary axis.。
=0: the 4th axis is set to a linear axis.
AXS5 =1: the 5th axis is set to a rotary axis.
=0：the 5th axis is set to a linear axis.

Standard setting：0 0 0 1 1 0 0 0

System parameter number

0	0	9								**A4TP**	**RAB**

RAB =1: each axis as a rotary axis rotates nearby.
=0: each axis as a rotary axis does not rotate nearby.

A4TP =1: 4-axis link system.
=0: non 4-axis link system.

Standard setting：0 0 0 0　0 0 0 1

System parameter number

0	1	0	RCUR	MSL	WCZS		RLC	ZCL	SCBM	

SCBM =1: execute the stroke check before movement.
=0: do not execute the stroke check before movement.

ZCL =1: clear the relative coordinate of the reference point return.
=0: do not clear the relative coordinate of the reference point return.

RLC =1: the relative coordinate system is cancelled after reset.
=0: the relative coordinate system is not cancelled after reset.

WCZS =1：the input value of workpiece coordinate system's zero subtracts the machine coordinate value.
=0：the input value of workpiece coordinate system's zero adds the machine coordinate value.

MSL =1：when multi-block in MDI mode is started circularly, the initial operation is done at the line where the cursor is.
=0: when multi-block in MDI mode is started circularly, the initial operation is done at the first line.

RCUR =1：the reset cursor returns to the program's initial position in Edit mode.
=0: the reset cursor does not return to the program's initial position in Edit mode.

Standard setting：0 0 1 0　0 0 1 0

System parameter number

0	1	1	BFA	LZR						OUT2

OUT2 =1: forbid entering into the 2nd stroke limit's outside area.
=0: forbid entering into the 2nd stroke limit's inside area.

LZR =1: perform the stroke check after the power supply is turned on and before the manual reference point return is performed.
=0: do not perform the stroke check after the power supply is turned on and before the manual reference point return is performed.

BFA =1: when an overtravel code is sent, an alarm occurs after overtravel.
=0: when an overtravel code is sent, an alarm occurs before overtravel. (the system alarm range is 5MM in the front of the set's forbidden area's each board).

Standard setting：0 0 0 0　0 0 0 1

System parameter number

0	1	2	RDR	FDR	TDR	RFO			LRP	RPD

RPD =1: manual rapid effective before reference point return after power-on

Appendix

	=0:	manual rapid ineffective before reference point return after power-on
LRP	=1:	the positioning (G00) interpolation type is linear
	=0:	the positioning (G00) interpolation type is non-linear
RFO	=1:	rapid feed stop when override is F0
	=0:	rapid feed does not stop when override is F0
TDR	=1:	dry run effective during tapping
	=0:	dry run ineffective during tapping
FDR	=1:	dry run effective during cutting feeding
	=0:	dy run ineffective during cutting feeding
RDR	=1:	dry run effective during rapid positioning
	=0:	dry run ineffective during rapid positioning

Standard setting：0 0 0 0　0 0 1 0

System parameter number

0	1	3							**HPC**	**NPC**

NPC =1: feed per revolution effective with no position encoder
=0: feed per revolution ineffective with no position encoder

HPC =1: position encoder installed
=0: position encoder not installed

Standard setting：0 0 0 0　0 0 1 0

System parameter number

0	1	4	**ROVT**						**DLF**	

DLF =1: reference point return by manual feed after reference point is setup and memorized
=0: reference point return by rapid traverse after reference point is setup and memorized

ROVT =1: the rapid override uses 6-gear
=0: the rapid override uses 4-gear

Standard setting：0 0 0 0　0 0 0 0

System parameter number

0	1	5			**PIIS**		**PPCK**	**ASL**	**PLAC**	**STL**

STL =1: To select prereading working type
=0: To select non-prereading working type

PLAC =1: Acceleration/deceleration type after forecasting interpolation: exponential
=0: Acceleration/deceleration type after forecasting interpolation: linear

ASL =1: Auto corner deceleration function of forecasting: speed difference control
=0: Auto corner deceleration function of forecasting: angular control

PPCK =1: To perform in-position check by forecasting
=0: Not perform in-position check by forecasting

PIIS =1: Overlapping interpolation effective in acceleration/deceleration blocks before forecasting
=0: Overlapping interpolation ineffective in acceleration/deceleration blocks before forecasting

Standard setting：0 0 0 0　0 0 0 1

System parameter number

0	1	6		ALS					FLLS	FBLS	FBOL

FBOL =1: Rapid traverse type: post acceleration/deceleration
=0: Rapid traverse type: pre- acceleration/deceleration
FBLS =1: Pre-acceleration/deceleration type of rapid traverse: S
=0: Pre-acceleration/deceleration type of rapid traverse: linear
FLLS =1: Post-acceleration/deceleration type of rapid traverse: exponential
=0: Post-acceleration/deceleration type of rapid traverse: linear
ALS =1: Auto corner feed effective
=0: Auto corner feed ineffective

Standard setting：0 0 0 0 0 010

System parameter number

0	1	7		CPCT	CALT	WLOE		HLOE	CLLE	CBLS	CBOL

CBOL =1: Cutting feed type: post-acceleration/deceleration
=0: Cutting feed type: pre-acceleration/deceleration
CBLS =1: Pre-acceleration/deceleration type of cutting feed: S
=0: Pre-acceleration/deceleration type of cutting feed: lineat
CLLE =1: Post-acceleration/deceleration type of cutting feed: exponential
=0: Post-acceleration/deceleration type of cutting feed: linear
HLOE =1: JOG running type: exponential
=0: JOG running type: linear
WLOE =1: MPG running type: exponential
=0: MPG running type: linear
CALT =1: Cutting feed acceleration clamping
=0: Cutting feed acceleration not clamping
CPCT =1: To control the in-position precision in cutting feed
=0: Not control the in-position precision in cutting feed

Standard setting：1 0 1 0 0 0 0 1

System parameter number

0	1	8		RVCS	RBK						RVIT

RVIT =1: To execute next block after compensation as backlash is over value allowable
=0: To execute next block during compensation as backlash is over value allowable
RBK =1 Backlash compensation when cutting/rapid move is performing
=0 No backlash compensation when cutting/rapid move is performing
RVCS =1: Backlash compensation type: ascending or decending
=0: Backlash compensation type: fixed frequency

Standard setting：0 0 0 0 0 0 0 0

System parameter number

0	1	9			ALS2	ALS1	ALM5	ALM4	ALM3	ALM2	ALM1

ALM1 =1: an alarm occurs when the 1st axis driver's alarm signal is 1.
=0: an alarm occurs when the 1st axis driver's alarm signal is 0.

Appendix

ALM2 =1: an alarm occurs when the 2nd axis river's alarm signal is 1.
=0: an alarm occurs when the 2nd axis river's alarm signal is 0.
ALM3 =1: an alarm occurs when the 3rd axis driver's alarm signal is 1.
=0: an alarm occurs when the 3rd axis driver's alarm signal is 0.
ALM4 =1: an alarm occurs when the 4th axis driver's alarm signal is 1.
=0: an alarm occurs when the 4th axis driver's alarm signal is 0.
ALM5 =1: an alarm occurs when the 5th axis driver's alarm signal is 1.
=0: an alarm occurs when the 5th axis driver's alarm signal is 0.
ALS1 =1: an alarm occurs when the 1st spindle driver's alarm signal is 1.
=0: an alarm occurs when the 1st spindle driver's alarm signal is 0.
ALS2 =1: an alarm occurs when the 2nd spindle driver's alarm signal is 1.
=0: an alarm occurs when the 2nd spindle driver's alarm signal is 0.

Standard setting：0 0 1 0 0 0 0 0

System parameter number

0	2	0			APC		USNO				ITL

ITL =1: All axes interlock signal effective
=0: All axes interlock signal ineffective
HVR =1: use the HVR function.
=0: do not use the HVR function.
USNO =1：the bus servo's older version.
=0：the bus servo's new version.
APC =1：use an absolute encoder.
=0：do not use an absolute encoder.

Standard setting：0 1 0 0 0 0 0 0

System parameter number

0	2	1						APC5	APC4	APC3	APC2	APC1

APC1 =1: the 1st axis servo encoder type selection: absolute.
=0: the 1st axis servo encoder type selection: incremental.
APC2 =1: the 2nd axis servo encoder type selection: absolute.
=0: the 2nd axis servo encoder type selection: incremental.
APC3 =1：the 3rd axis servo encoder type selection: absolute.
=0：the 3rd axis servo encoder type selection: incremental.
APC4 =1：the 4th axis servo encoder type selection: absolute.
=0：the 4th axis servo encoder type selection: incremental.
APC5 =1：the 5th axis servo encoder type selection: absolute.
=0：the 4th axis servo encoder type selection: incremental.

Standard setting：0 1 0 0 0 0 0 0

System parameter number

0	2	2		PRC	DAL						

DAL =1: the absolute position display considers the tool length compensation.
=0: the absolute position display does not consider the tool length compensation.

PRC =1: in tool offset compensation amount, PRC signal is used in the workpiece coordinate system's offset input.

=0: in tool offset compensation amount, PRC signal is not used in the workpiece coordinate system's offset input.

Standard setting：0 0 0 0 0 0 0 0

System parameter number

0	2	3			POSM						

POSM =1: Mode is displayed on program monitoring page

=0: Mode is not displayed on program monitoring page

Standard setting：0 1 0 0 0 0 0 0

System parameter number

0	2	4			NPA						

NPA =1: To switch to alarm page when alarm occurs

=0: Not switch to alarm page when alarm occurs

Standard setting：0 0 0 0 0 0 0

System parameter number

0	2	5		ALM	DGN	GRA	SET		SYS	PRG	POS

POS =1: To switch over page by pressing POSITION key in position page

=0: Not switch over page by pressing POSITION key in position page

PRG =1: To switch over page by repressing POSITION key in program page

=0: Not switch over page by repressing POSITION key in program page

SYS =1: To switch over page by repressing PARAMETER key in program page

=0: Not switch over page by repressing PARAMETER key in program page

SET =1: To switch over page by repressing SET key in set page

=0: Not switch over page by repressing SET key in set page

GRA =1: To switch over page by repressing GRAPHIC key in graphic page

=0: Not switch over page by repressing GRAPHIC key in graphic page

DGN =1: To switch over page by repressing DIAGNOSE key in diagnosis page

=0: Not switchover page by repressing DIAGNOSE key in diagnosis page

ALM =1: To switch over page by repressing ALARM key in alarm page

=0: Not switch over page by repressing ALARM key in alarm page

Standard setting：1 1 1 1 0 1 1 1

System parameter number

0	2	6		HELP	PLC		STOF		SMDI		PETP

PETP =1: To switch to program page by pressing panel Edit key

=0: Not to switch to program page by pressing panel Edit key

SMDI =1: Press <Program> to automatically skip to the MDI page in MDI mode.

=0: Press <Program> not to automatically skip to the MDI page in MDI mode.

STOF =1：directly switch to the offset page when the system switches to the setting page.

=0：do not directly switch to the offset page when the system switches to the setting page.

PLC =1: To switch over page by repressing PLC key in PLC page
=0: Not switch over page by repressing PLC key in PLC page
HELP =1: To switch over page by repressing HELP key in help page
=0: Not switch over page by repressing HELP key in help page

Standard setting：1 1 0 1 0 1 0 1

System parameter number

0 2 7				NE9				NE8

NE8 =1: Editting of subprogram with 80000 – 89999 unallowed
=0: Editting of subprogram with 80000 – 89999 allowed
NE9 =1: Editting of subprogram with 90000 - 99999 unallowed
=0: Editting of subprogram with 90000 - 99999 allowed

Standard setting：0 0 0 1 0 0 0 1

System parameter number

0 2 8	MCL			MKP				

MKP =1: To clear the program edited when M02，M30 or % is executed in MDI mode
=0: Not clear the program edited when M02，M30 or % is executed in MDI mode
MCL =1: To delete the program edited when pressing RESET key in MDI mode
=0: Not delete the program edited when pressing RESET key in MDI mode

Standard setting：0 0 0 1 0 0 0 0

System parameter number

0 2 9				IWZ	WZO	MCV	GOF	WOF

WOF =1: Tool wear offset input by MDI disabled
=0: Tool wear offset input by MDI enabled
GOF =1: Geometric tool offset input by MDI disabled
=0: Geometric tool offset input by MDI enabled
MCV =1: Macro variables input by MDI disabled
=0: Macro variables input by MDI enabled
WZO =1: Workpiece origin offset input by MDI disabled
=0: Workpiece origin offset input by MDI enabled
IWZ =1: Workpiece origin offset input by MDI during dwell disabled
=0: Workpiece origin offset input by MDI during dwell enabled

Standard setting：0 0 0 0 0 0 0 0

System parameter number

0 3 0		SFA						

SFA =1: output SF signal even if no gear shift.
=0: output SF signal with gear shift.

Standard setting：0 0 0 0 0 0 0 0

System parameter number

0 3 1						G19	G18	G01

G01 =1: G01 mode at power-on or clearing
=0: G00 mode at power-on or clearing
G18 =1: G18 plane at power-on or clearing
=0: is not G18 plane at power-on or clearing
G19 =1: it is G19 mode. When G19=1, please set G18 to 0.
=0: it is determined by No：31#1.

G19	G18	G17, G18, G19 mode
0	0	G17 mode（X-Y plane）
0	1	G18 mode（Z-X plane）
1	0	G19 mode（Y-Z plane）

Standard setting：0 0 0 0　0 0 1 0

System parameter number

0	3	2			AD2						

AD2 =1: an alarm occurs if two or more same addresses are specified in a block
=0: no alarm if two or more same addresses are specified in a block

Standard setting：0 1 0 0　0 0 0 0

System parameter number

0	3	3		M3B			M30		M02		

M02 =1: To return to block beginning when M02 is to be executed
=0: Not to return to block beginning when M02 is to be executed
M30 =1: To return to block beginning when M30 is to be executed
=0: Not to return to block beginning when M30 is to be executed
M3B =1: At most three M codes allowable in a section of program
=0: Only one M code allowable in a section of program

Standard setting：0 0 0 1　0 0 0 0

System parameter number

0	3	4		CFH							DWL

DWL =1: G04 for dwell per revolution in per revolution feed mode
=0: G04 not for dwell per revolution in per revolution feed mode
CFH =1: To clear F, H, D codes at reset or emergency stop
=0: To reserve F, H, D codes at reset or emergency stop

Standard setting：0 0 0 0　0 0 0 0

System parameter number

0	3	5		C07		C05	C04	C03	C02	C01	

C01 =1: To clear G codes of 01 group at reset or emergency stop
=0: To reserve G codes of 01 group at reset or emergency stop
C02 =1: To clear G codes of 02 group at reset or emergency stop
=0: To reserve G codes of 02 group at reset or emergency stop
C03 =1: To clear G codes of 03 group at reset or emergency stop

Appendix

	=0:	To reserve G codes of 03 group at reset or emergency stop
C04	=1:	To clear G codes of 04 group at reset or emergency stop
	=0:	To reserve G codes of 04 group at reset or emergency stop
C05	=1:	To clear G codes of 05 group at reset or emergency stop
	=0:	To reserve G codes of 05 group at reset or emergency stop
C06	=1:	To clear G codes of 06 group at reset or emergency stop
	=0:	To reserve G codes of 06 group at reset or emergency stop
C07	=1:	To clear G codes of 07 group at reset or emergency stop
	=0:	To reserve G codes of 07 group at reset or emergency stop

Standard setting：1 0 0 0 0 0 0 0

System parameter number

0	3	6		C15	C14	C13	C12	C11	C10	C09	C08

C08	=1:	To clear G codes of 08 group at reset or emergency stop
	=0:	To reserve G codes of 08 group at reset or emergency stop
C09	=1:	To clear G codes of 09 group at reset or emergency stop
	=0:	To reserve G codes of 09 group at reset or emergency stop
C10	=1:	To clear G codes of 10 group at reset or emergency stop
	=0:	To reserve G codes of 10 group at reset or emergency stop
C11	=1:	To clear G codes of 11 group at reset or emergency stop
	=0:	To reserve G codes of 11 group at reset or emergency stop
C12	=1:	To clear G codes of 12 group at reset or emergency stop
	=0:	To reserve G codes of 12 group at reset or emergency stop
C13	=1:	To clear G codes of 13 group at reset or emergency stop
	=0:	To reserve G codes of 13 group at reset or emergency stop
C14	=1:	To clear G codes of 14 group at reset or emergency stop
	=0:	To reserve G codes of 14 group at reset or emergency stop
C15	=1:	To clear G codes of 15 group at reset or emergency stop
	=0:	To reserve G codes of 15 group at reset or emergency stop

Standard setting：0 0 0 0 0 0 0 1

System parameter number

0	3	7		SCS	CSS	CSZR		SOC	RSC	BDP	SCRW

SCRW =1: To perform pitch compensation

=0: Not perform pitch compensation

BDP =1: use the two-way pitch error compensation.

=0: do not use the two-way pitch error compensation.

RSC =1: To calculate G96 spindle speed according to current coordinate during G0 rapid positioning

=0: To calculate G96 spindle speed according to end point coordinate during G0 rapid positioning

SOC =1: G96 spindle speed clamped behind spindle override

=0: G96 spindle speed clamped before spindle override

CSZR =1：CS contour controlling reference point return function is valid.

=0：CS contour controlling reference point return function is invalid.

CSS =1：execute Cs contour control in each spindle.

=0： do not execute Cs contour control in each spindle.

Appendix

SCS =1： use the Cs contour control function.

=0： do not use the Cs contour control function.

Standard setting：0 0 0 0 0 0 0 0

System parameter number

0	3	8		SGB	GTT	FLRE	FLR	EOV	MPP	SAR

SAR =1: detect the spindle speed in-position signal

=0: do not detect the spindle speed in-position signal

MPP =1: the program codes P to execute the spindle selection in multi-spindle control.

=0: the program codes P to execute the spindle selection in multi-spindle control.

EOV =1: use each spindle override signal.

=0: not use each spindle override signal.

FLR =1: unit of permissive rate (q) and change rate® in the spindle speed wave check is 0.1%。

=0: unit of permissive rate (q) and change rate® in the spindle speed wave check is 1%.

FLRE =1: the spindle speed wave check is enabled.

=0: the spindle speed wave check is disabled.

GTT =1： the spindle gear selects T type mode.

=0： the spindle gear selects M type mode.

SGB =0： M type the gear switch mode selects each gear's max. speed(mode A).

=1： M type the gear switch mode selects each gear's switch point's speed （mode B）

Standard setting：0 0 1 0 0 0 0 0

System parameter number

0	3	9	THSY	THAT	THST			DIA		TLC

TLC =1: tool length compensation type: B

=0: tool length compensation type: A

DIA =1： use the diameter programming.

=0： use the radius programming.

THST =1： the spindle pulse samples the even value in thread cutting.

=0： the spindle pulse samples the original in thread cutting.

THAT =1： the acceleration/deceleration in thread cutting is exponential type.

=0： the acceleration/deceleration in thread cutting is linear type.

THSY =1： the synchronous mode B in thread cutting.

=0： the synchronous mode A in thread cutting.

Standard setting：1 0 0 0 0 1 0 0

System parameter number

0	4	0	ODI		THST	TPRT		CCN		SUP

SUP =1: Start-up type in tool radius compensation: B

=0: Start-up type in tool radius compensation: A

CCN =1: To move to the intermediate point by G28 and cancel compensation in tool radius compensation

=0: To move to the intermediate point by G28 and reserve compensation in tool radius compensation

TPRT =0: the thread cutting's linear acceleration/deceleration uses the time constant.

=1: the thread cutting's linear acceleration/deceleration uses the acceleration constant.

THST =0: the thread head signal's read mode uses each read.

=1: the thread head signal's read mode is read only once.

ODI =1: Tool radius compensation value set by diameter

=0: Tool radius compensation value set by radius

Standard setting：0 0 0 1 0 1 0 0

System parameter number

0	4	1			CNI	G39		PUIT			

PUIT =1: Data parameter input and display are decided by 0#2 INI.

=0: Data parameter input and display are metric units

G39 =1: Corner rounding effective in radius compensation

=0: Corner rounding ineffective in radius compensation

CNI =1: Interference check enabled in radius compensation

=0: Interference check disabled in radius compensation

Standard setting：0 1 1 0 0 0 0 0

System parameter number

0	4	2				RD2	RD1				

RD1 =1: To set the retraction direction of G76，G87: negative

=0: To set the retraction direction of G76，G87: positive

RD2 =1: To set the retraction axis of G76，G87: the 2nd axis

=0: To set the retraction axis of G76，G87: the 1st axis

Standard setting：0 0 0 0 0 0 0 0

System parameter number

0	4	3							PLS		PDI

PDI =1: the plane's 2nd aixs specified by diameter is used in polar coordinate interpolation mode.

=0: the plane's 2nd aixs specified by radius is used in polar coordinate interpolation mode.

RD2 =1: use the polar coordinate interpolation offset function.

=0: do not use the polar coordinate interpolation offset function.

Standard setting：0 0 0 0 0 0 1 0

System parameter number

0	4	4				PCP	DOV				

DOV =1: Override effective during rigid tapping retraction

=0: Override ineffective during rigid tapping retraction

Appendix

PCP =1: High-speed peck drilling cycle for flexible tapping
=0: Standard peck drilling cycle for flexible tapping

Standard setting：0 0 0 0 0 0 0 0

System parameter number

0	4	5					OVS	OVU	TDR		NIZ

NIZ =1: To perform rigid tapping smoothing
=0: Not perform rigid tapping smoothing

TDR =1: To use the same constant during the rigid tapping advance and retraction
=0: Not use the same constant during the rigid tapping advance and retraction

OVU =1: 10% retraction override for rigid tapping
=0: 1% retraction override for rigid tapping

OVS =1: In rigid tapping, selection and cancel signal for feedrate override enable
=0: In rigid tapping, selection and cancel signal for feedrate override disable

Standard setting：0 0 0 0 0 0 0 0

System parameter number

0	4	6				ORI				SSOG	

SSOG =1: For servo spindle control at the beginning of rigid tapping
=0: For following spindle control at the beginning of rigid tapping

ORI =1: To perform spindle dwell when rigid tapping starts
=0: Not perform spindle dwell when rigid tapping starts

Standard setting：0 0 0 0 0 0 1 0

System parameter number

0	4	9									RPST

RPST =1: movement in G01 mode in the course of the program restart.
=0: movement at dry run speed in G00 mode in the course of the program restart.

Standard setting：0 0 0 0 0 0 0 0

System parameter number

0	5	0			SIM					REL	

REL =1: Relative position display setting of indexing table: within 360°
=0: Relative position display setting of indexing table: beyond 360°

SIM =1: alarm if indexing code and other axes codes are in the same block
=0: Do not make alarm if indexing code and other axes codes are in the same block

Standard setting：0 1 0 0 0 0 0 0

System parameter number

0	5	1		MDLY		SBM					

SBM =1: Single block allowed in macro statement
=0: Single block unallowed in macro statement

MDLY =1: Delay is allowed in macro statement

Appendix

=0: Delay is unallowed in macro statement

Standard setting：0 0 1 0 0 0 0 0

System parameter number

0	5	2		CLV	CCV						

CCV =1: Macro common variables #100 - #199 clearing after reset

=0: Macro common variables #100 - #199 not clearing after reset

CLV =1: Macro local variables #1 - #50 clearing after reset

=0: Macro local variables #1 - #50 not clearing after reset

Standard setting：0 0 0 0 0 0 0 0

System parameter number

0	5	3		PLCV				LAD3	LDA2	LAD1	LAD0

LAD0~LAD3 They are binary combination parameters. If they are 0, it uses No. 0 ladder, if they are 1~15, it uses 0~15 ladder diagram.

PLCV =1: Read and display PLC software version number.

=0: Do not read but display PLC software version number

Standard setting：1 0 0 0 0 0 1 1

System parameter number

0	5	4		OPRG	PRGS						

PRGS =1： initial state of program switch is ON.

=0： initial state of program switch is OFF.

OPRG =1: Debugging and above authorithies, one key input/output is effective for workpiece program

=0: Debugging and above authorities, one key input/output is ineffective for workpiece program

Standard setting：0 1 0 0 0 0 0 0

System parameter number

0	5	5									CANT

CANT =1: Automatic clearing for single piece

=0: Not automatic clearing for single piece

Standard setting：0 0 0 0 0 0 0 0

System parameter number

0	5	6		HNGD							HPF

HPF =1: To select full running for MPG moving

=0: Not select full running for MPG moving

HNGD =1: Axes moving direction are identical with MPG rotation direction

=0: Axes moving direction are not identical with MPG rotation direction

Standard setting：1 0 0 0 0 0 0 1

System parameter number

0	5	7					PLW5	PLW4	PLW3	PLW2	PLW1

PLW1 =1：the 1st axis pulse width changes along with speed.

=0：the 1st axis pulse width is fixed to 1usec.

PLW2 =1：the 2nd axis pulse width changes along with speed.

=0：the 2nd axis pulse width is fixed to 1usec.

PLW3 =1：the 3rd axis pulse width changes along with speed.

=0：the 3rd axis pulse width is fixed to 1usec.

PLW4 =1：the 4th axis pulse width changes along with speed.

=0：the 4th axis pulse width is fixed to 1usec.

PLW5 =1：the 5th axis pulse width changes along with speed.

=0：the 5th axis pulse width is fixed to 1usec.

Standard setting：0 0 0 0 0 0 0 0

System parameter number

0	**5**	**8**				**NEG 5**	**NEG 4**	**NEG 3**	**NEG 2**	**NEG1**

PLW1 =1：the 1st axis is ignored

=0：the 1st axis is not ignored

PLW2 =1：the 2nd axis is ignored

=0：the 2nd axis is not ignored

PLW3 =1：the 3rd axis is ignored

=0：the 3rd axis is not ignored

PLW4 =1：the 4th axis is ignored

=0：the 4th axis is not ignored

PLW5 =1：the 5th axis is ignored

=0：the 5th axis is not ignored

Standard setting：0 0 0 0 0 0 1 0

System parameter number

0	**5**	**9**		**LEDT**	**LOPT**					

LOPT =1: Use external operator panel lock

=0: Not use external operator panel lock

LEDT =1: Use external editing lock

=0: Not use external editing lock

Standard setting：0 0 0 0 0 0 0 0

System parameter number

0	**6**	**0**	**PMCA**	**PMCP**				**PMCS**	**EPW**	

EPW =1: maximum number of position switch is 16PCS.

=0: maximum number of position switch is 10PCS.

PMCS =1： PMC axis' axis selection is specified by G signal.

=0： PMC axis' axis selection is not specified by G signal.

PMCP =1： PMC axis' zero return mode with one-rotation signal.

=0： PMC axis' zero return mode without one-rotation signal.

PMCA =1： alarm when PMC axis does not execute the reference point return, codes

Appendix

the machine coordinate selection mode.

=0： no alarm when PMC axis does not execute the reference point return, codes the machine coordinate selection mode.

Standard setting：0 0 0 0 0 0 0 0

System parameter number

0	6	1		FALM	LALM	EALM	SALM	AALM			SSC

SSC =1: To use constant surface speed control
=0: Not use constant surface speed control
AALM =1: External user alarm ignored
=0: External user alarm not ignored
SALM =1: Spindle driver alarm ignored
=0: Spindle driver alarm not ignored
EALM =1: Emergency stop alarm ignored
=0: Emergency stop alarm not ignored
LALM =1: Limit alarm ignored
=0: Limit alarm not ignored
FALM =1: Feed axis driver alarm ignored
=0: Feed axis driver alarm not ignored

Standard setting：0 0 0 0 0 0 0 0

2． Data Parameters

No.	definition	Default
0000	I/O channel, input and output device selection (1:RS232 2:USB)	2

Setting range：1～2
It is set to 0 or 1 for communication between CNC and PC via RS232 interface, and set to 2 when CNC is connected with U disk.

0005	Axes controlled by the CNC	3

Setting range：3～5

0006	CNC language selection（0：Chinese 1：English）	0

Setting range：0～2 .

0008	Ethernet bus's slave station MDT data package bytes	16

Setting range：0～20

0009	Max. retransmission times of Ethernet bus	10

Setting range：0～30

0010	The 1st axis offset amount of external workpiece origin	0

Setting range：-19999.9998～19999.9998（mm)

0011	The 2nd axis offset amount of external workpiece origin	0.0000

Setting range：-9999.9999～9999.9999（mm)

Appendix

0012	The 3^{rd} axis offset amount of external workpiece origin	0.0000

Setting range：-9999.9999～9999.9999（mm)

0013	The 4^{th} axis offset amount of external workpiece origin	0.0000

Setting range：-9999.9999～9999.9999（mm)

0014	The 5^{th} axis offset amount of external workpiece origin	0.0000

Setting range：-9999.9999～9999.9999（mm)

0015	Offset amount of the 1^{st} axis' workpiece origin in G54	0.0000

Setting range：-19999.9998～19999.9998（mm）

0016	Offset amount of the 2^{nd} axis' workpiece origin in G54	0.0000

Setting range：-9999.9999～9999.9999（mm）

0017	Offset amount of the 3^{rd} axis' workpiece origin in G54	0.0000

Setting range：-9999.9999～9999.9999（mm）

0018	Offset amount of the 4^{th} axis' workpiece origin in G54	0.0000

Setting range：-9999.9999～9999.9999（mm）

0019	Offset amount of the 5^{th} axis' workpiece origin in G54	0.0000

Setting range：-9999.9999～9999.9999（mm）

0020	Offset amount of the 1^{st} axis' workpiece origin in G55	0.0000

Setting range：-19999.9998～19999.9998（mm）

0021	Offset amount of the 2^{nd} axis' workpiece origin in G55	0.0000

Setting range：-19999.9999～19999.9999（mm）

0022	Offset amount of the 3^{rd} axis' workpiece origin in G55	0.0000

Setting range：-9999.9999～9999.9999（mm）

0023	Offset amount of the 4^{th} axis' workpiece origin in G55	0.0000

Setting range：-9999.9999～9999.9999（mm）

0024	Offset amount of the 5^{th} axis' workpiece origin in G55	0.0000

Setting range：-9999.9999～9999.9999（mm）

0025	Offset amount of the 1^{st} axis' workpiece origin in G56	0.0000

Setting range：-19999.9998～19999.9998（mm）

0026	Offset amount of the 2^{nd} axis' workpiece origin in G56	0.0000

Setting range：-9999.9999～9999.9999（mm）

0027	Offset amount of the 3rd axis' workpiece origin in G56	0.0000

Setting range：-9999.9999～9999.9999（mm）

0028	Offset amount of the 4th axis' workpiece origin in G56	0.0000

Setting range：-9999.9999～9999.9999（mm）

0029	Offset amount of the 5th axis' workpiece origin in G56	0.0000

Setting range：-9999.9999～9999.9999（mm）

0030	Offset amount of the 1st axis' workpiece origin in G57	0.0000

Setting range：-19999.9998～19999.9998（mm）

0031	Offset amount of the 2nd axis' workpiece origin in G57	0.0000

Setting range：-9999.9999～9999.9999（mm）

0032	Offset amount of the 3rd axis' workpiece origin in G57	0.0000

Setting range：-9999.9999～9999.9999（mm）

0033	Offset amount of the 4th axis' workpiece origin in G57	0.0000

Setting range：-9999.9999～9999.9999（mm）

0034	Offset amount of the 5th axis' workpiece origin in G57	0.0000

Setting range：-9999.9999～9999.9999（mm）

0035	Offset amount of the 1st axis' workpiece origin in G58	0.0000

Setting range：-19999.9998～19999.9998（mm）

0036	Offset amount of the 2nd axis' workpiece origin in G58	0.0000

Setting range：-9999.9999～9999.9999（mm）

0037	Offset amount of the 3rd axis' workpiece origin in G58	0.0000

Setting range：-9999.9999～9999.9999（mm）

0038	Offset amount of the 4th axis' workpiece origin in G58	0.0000

Setting range：-9999.9999～9999.9999（mm）

0039	Offset amount of the 5th axis' workpiece origin in G58	0.0000

Setting range：-9999.9999～9999.9999（mm）

0040	Offset amount of the 1st axis' workpiece origin in G59	0.0000

Setting range：-19999.9998～19999.9998（mm）

0041	Offset amount of the 2nd axis' workpiece origin in G59	0.0000

Setting range：-9999.9999～9999.9999（mm）

0042	Offset amount of the 3rd axis' workpiece origin in G59	0.0000

Setting range：-9999.9999～9999.9999（mm）

0043	Offset amount of the 4th axis' workpiece origin in G59	0.0000

Setting range：-9999.9999～9999.9999（mm）

0044	Offset amount of the 5th axis' workpiece origin in G59	0.0000

Setting range：-9999.9999～9999.9999（mm）

0045	The 1st reference point's 1st axis coordinate value on the machine coordinate system	0.0000

Setting range：-19999.9998～19999.9998（mm）

0046	The 1st reference point's 2nd axis coordinate value on the machine coordinate system	0.0000

Setting range：-9999.9999～9999.9999（mm）

0047	The 1st reference point's 3rd axis coordinate value on the machine coordinate system	0.0000

Setting range：-9999.9999～9999.9999（mm）

0048	The 1st reference point's 4th axis coordinate value on the machine coordinate system	0.0000

Setting range：-9999.9999～9999.9999（mm）

0049	The 1st reference point's 5th axis coordinate value on the machine coordinate system	0.0000

Setting range：-9999.9999～9999.9999（mm）

0050	The 2nd reference point's 1st axis coordinate value on the machine coordinate system	0.0000

Setting range：-19999.9998～19999.9998（mm）

0051	The 2nd reference point's 2nd axis coordinate value on the machine coordinate system	0.0000

Setting range：-9999.9999～9999.9999（mm）

0052	The 2nd reference point's 3rd axis coordinate value on the machine coordinate system	0.0000

Setting range：-9999.9999～9999.9999（mm）

0053	The 2nd reference point's 4th axis coordinate value on the machine coordinate system	0.0000

Setting range：-9999.9999～9999.9999（mm）

0054	The 2nd reference point's 5th axis coordinate value on the machine coordinate system	0.0000

Setting range：-9999.9999～9999.9999（mm）

0055	The 3rd reference point's 1st axis coordinate value on the machine coordinate system	0.0000

Setting range：-19999.9998～19999.9998（mm）

0056	The 3rd reference point's 2nd axis coordinate value on the machine coordinate system	0.0000

Setting range：-9999.9999～9999.9999（mm)

0057	The 3rd reference point's 3rd axis coordinate value on the machine coordinate system	0.0000

Setting range：-9999.9999～9999.9999（mm）

0058	The 3rd reference point's 4th axis coordinate value on the machine coordinate system	0.0000

Setting range：-9999.9999～9999.9999（mm）

0059	The 3rd reference point's 5th axis coordinate value on the machine coordinate system	0.0000

Setting range：-9999.9999～9999.9999（mm）

0060	The 4th reference point's 1st axis coordinate value on the machine coordinate system	0.0000

Setting range：-19999.9998～19999.9998（mm）

0061	The 4th reference point's 2nd axis coordinate value on the machine coordinate system	0.0000

Setting range：-9999.9999～9999.9999（mm)

0062	The 4th reference point's 3rd axis coordinate value on the machine coordinate system	0.0000

Setting range：-9999.9999～9999.9999（mm)

0063	The 4th reference point's 4th axis coordinate value on the machine coordinate system	0.0000

Setting range：-9999.9999～9999.9999（mm)

0064	The 4th reference point's 5th axis coordinate value on the machine coordinate system	0.0000

Setting range：-9999.9999～9999.9999（mm)

0066	Stored stroke check 1's 1st axis negative border's coordinate value	-19998

Setting range：-19999.9998～19999.9998（mm)

0067	Stored stroke check 1's 1st axis positive border's coordinate value	19998

Setting range：-19999.9998～19999.9998（mm)

0068	Stored stroke check 1's 2nd axis negative border's coordinate value	-9999

Setting range：-9999.9999～9999.9999（mm)

0069	Stored stroke check 1's 2nd axis positive border's coordinate value	9999

Setting range：-9999.9999～9999.9999（mm)

0070	Stored stroke check 1's 3rd axis negative border's coordinate value	-9999

Setting range：-9999.9999～9999.9999（mm)

0071	Stored stroke check 1's 3rd axis positive border's coordinate value	9999

Setting range：-9999.9999～9999.9999（mm)

0072	Stored stroke check 1's 4th axis negative border's coordinate value	-9999

Setting range：-9999.9999～9999.9999（mm)

0073	Stored stroke check 1's 4th axis positive border's coordinate value	9999

Setting range：-9999.9999～9999.9999（mm)

0074	Stored stroke check 1's 5th axis negative border's coordinate value	-9999

Setting range：-9999.9999～9999.9999（mm)

0075	Stored stroke check 1's 5th axis positive border's coordinate value	9999

Setting range：-9999.9999～9999.9999（mm)

0076	Stored stroke check 2's 1st axis negative border's coordinate value	-19998

Setting range：-19999.9989～19999.9998（mm)

0077	Stored stroke check 2's 1st axis positive border's coordinate value	19998

Setting range：-1999.9998～19999.9998（mm)

0078	Stored stroke check 2's 2nd axis negative border's coordinate value	-9999

Setting range：-9999.9999～9999.9999（mm)

0079	Stored stroke check 2's 2nd axis positive border's coordinate value	9999

Setting range：-9999.9999～9999.9999（mm)

0080	Stored stroke check 2's 3rd axis negative border's coordinate value	-9999

Setting range：-9999.9999～9999.9999（mm)

0081	Stored stroke check 2's 3rd axis positive border's coordinate value	9999

Setting range：-9999.9999～9999.9999（mm)

0082	Stored stroke check 2's 4th axis negative border's coordinate value	-9999

Setting range：-9999.9999～9999.9999（mm)

0083	Stored stroke check 2's 4th axis positive border's coordinate value	9999

Setting range：-9999.9999～9999.9999（mm)

0084	Stored stroke check 2's 5th axis negative border's coordinate value	-9999

Setting range：-9999.9999～9999.9999（mm)

0085	Stored stroke check 2's 5th axis positive border's coordinate value	9999

Setting range：-9999.9999～9999.9999（mm)

0086	Dry run speed	1000

Setting range：0～9999 (mm/min)

0087	Cutting feedrate at power-on	100

Setting range：0～9999 (mm/min)

0088	G0 rapid traverse speed of X axis	4000

Setting range：

Metric：1～30000 (mm/min)
Inch ：1～30000/ 25.4（inch/min）
Rotary axis: 1～30000（deg/min）

0089	G0 rapid traverse speed of the 2nd axis	8000

Setting range：

Metric ：0～30000 (mm/min)
Inch ：0～30000/ 25.4（inch/min）
Rotary axis：0～30000（deg/min）

0090	G0 rapid traverse speed of the 3rd axis	8000

Setting range：

Metric ：0～30000 (mm/min)
Inch ：0～30000/ 25.4（inch/min）
Rotary axis ：0～30000（deg/min）

0091	G0 rapid traverse speed of the 4th axis	8000

Setting range：

Metric ：0～30000 (mm/min)
Inch ：0～30000/ 25.4（inch/min）
Rotary axis：0～30000（deg/min）

0092	G0 rapid traverse speed of the 5th axis	8000

Setting range：

Metric ：0～30000 (mm/min)
Inch ：0～30000/ 25.4（inch/min）
Rotary axis：0～30000（deg/min）

0093	F0 rapid override of each axis (for all axes)	30

Setting range：1～1000 (mm/min)

Appendix

0094	Maximum control speed in rapid positioning (for all axes)	8000

Setting range：300～30000(mm/min)

0095	Minimum control speed in rapid positioning (for all axes)	0

Setting range：0～300 (mm/min)

0096	Maximum control speed in cutting feed (for all axes)	8000

Setting range：300～30000(mm/min)

0097	Minimum control speed in cutting feed (for all axes)	0

Setting range：0～300 (mm/min)

0098	Feedrate of manual continuous feed for axes (JOG)	1000

Setting range：0～30000 (mm/min)

0099	（FL）speed when receiving Z axis pulse signal (for all axes)	100

Setting range：1～200 (mm/min)

0100	The 1st axis' reference point return speed	2000

Setting range：0～9999 (mm/min)

0101	The 2nd axis' reference point return speed	4000

Setting range：0～9999 (mm/min)

0102	The 3rd axis' reference point return speed	4000

Setting range：0～9999 (mm/min)

0103	The 4th axis' reference point return speed	4000

Setting range：0～9999 (mm/min)

0104	The 5th axis' reference point return speed	4000

Setting range：0～9999 (mm/min)

0105	L type time constant of pre-acceleration/deceleration of rapid 1st axis	100

Setting range：3～400 （ms）

0106	L type time constant of pre-acceleration/deceleration of rapid 2nd axis	100

Setting range：3～400 （ms）

0107	L type time constant of pre-acceleration/deceleration of rapid 3rd axis	100

Setting range：3～400 （ms）

0108	L type time constant of pre-acceleration/deceleration of rapid 4th axis	100

Appendix

Setting range：3～400 （ms）

0109	L type time constant of pre-acceleration/deceleration of rapid 5th axis	100

Setting range：3～400 （ms）

0110	S type time constant of pre-acceleration/deceleration of rapid 1st axis	100

Setting range：3～400 （ms）

0111	S type time constant of pre-acceleration/deceleration of rapid 2nd axis	100

Setting range：3～400 （ms）

0112	S type time constant of pre-acceleration/deceleration of rapid 3rd axis	100

Setting range：3～400 （ms）

0113	S type time constant of pre-acceleration/deceleration of rapid 4th axis	100

Setting range：3～400 （ms）

0114	S type time constant of pre-acceleration/deceleration of rapid 5th axis	100

Setting range：3～400 （ms）

0115	L type time constant of post-acceleration/ deceleration of rapid 1st axis	80

Setting range：0～400 （ms）

0116	L type time constant of post-acceleration/ deceleration of rapid 2nd axis	80

Setting range：0～400 （ms）

0117	L type time constant of post-acceleration/ deceleration of rapid 3rd axis	80

Setting range：0～400 （ms）

0118	L type time constant of post-acceleration/ deceleration of rapid 4th axis	80

Setting range：0～400 （ms）

0119	L type time constant of post-acceleration/ deceleration of rapid 5th axis	80

Setting range：0～400 （ms）

0120	S type time constant of post-acceleration/ deceleration of rapid 1st axis	60

Setting range：0～400 （ms）

0121	S type time constant of post-acceleration/ deceleration of rapid 2nd axis	60

Setting range：0～400 （ms）

0122	S type time constant of post-acceleration/ deceleration of rapid 3rd axis	60

Setting range：0～400 （ms）

0123	S type time constant of post-acceleration/ deceleration of rapid 4th axis	60

Setting range：0～400 （ms）

0124	S type time constant of post-acceleration/ deceleration of rapid 5th axis	60

Setting range：0～400 （ms）

0125	L type time constant of cutting feed's front-acceleration/deceleration	100

Setting range：3～400 （ms）

0126	S type time constant of cutting feed's front-acceleration/deceleration	100

Setting range：3～400 （ms）

0127	L type time constant of cutting feed's post-acceleration/deceleration	80

Setting range：3～400 （ms）

0128	E type time constant of cutting feed's post-acceleration/deceleration	60

Setting range：3～400 （ms）

0129	Exponential type acceleration/deceleration FL speed	10

Setting range：0～9999 （mm/min）

0130	Maximum blocks merged in pre-interpolation	0

Setting range：0～10

0131	In-position precision of cutting feed	0.03

Setting range：0.001～0.5（mm）

0132	Control precision of circular interpolation	0.03

Setting range：0～0.5（mm）

0133	Contour control precision of pre-interpolation	0.01

Setting range：0.0010～0.5000 （mm）

0134	Acceleration of the fore linear acceleration/deceleration interpolated in forecasting control	250

Setting range：0～2000 （mm/s²）

0135	Forecasting control, S type pre-acceleration /deceleration time constant	100

Setting range：0～400 （ms）

Appendix

0136	Linear time constant of the post acceleration /deceleration in forecasting control	80

Setting range：0～400 （ms）

0137	Exponential time constant of the post acceleration/deceleration in forecasting control	60

Setting range：0～400 （ms）

0138	Exponential acceleration/deceleration FL speed of cutting feed in forecasting control	10

Setting range：0～400 （ms）

0139	Contour control precision in forecasting control	0.01

Setting range：0～0.5 (mm)

0140	Blocks merged in forecasting control	0

Setting range：0～10

0141	In-position precision in forecasting control	0.05

Setting range：0～0.5 (mm)

0142	Length condition of spline formation in forecasting	5

Setting range：0～30

0143	Angular condition of spline formation in forecasting	10

Setting range：0～30

0144	Critical angle of two blocks during automatic corner deceleration in forecasting control	5

Setting range：2～178 (degree)

0145	Minimum feedrate of automatic corner deceleration in forecasting control	120

Setting range：10～1000 (mm/min)

0146	Axis error allowable for speed difference deceleration in forecasting control	80

Setting range：60～1000

0147	Cutting precision grade in forecasting control	2

Setting range：0～8

0148	External acceleration limit of circular interpolation	1000

Setting range：100～5000 （mm/s^2）

0149	Lower limit of external acceleration clamp for circular interpolation	200

Setting range：0～2000 (mm/min)

0150	Acceleration clamp time constant of cutting feed	50

Appendix

Setting range：0～1000（ms）

0151	Maximum clamp speed of MPG incomplete running	2000

Setting range：0～3000 (mm/min)

0152	Linear acceleration /deceleration time constant of MPG	120

Setting range：0～400 （ms）

0153	Exponential acceleration/deceleration time constant of MPG	80

Setting range：0～400 （ms）

0154	Acceleration clamp time constant of MPG	100

Setting range：0～400 （ms）

0155	Maximum clamp speed of step feed	1000

Setting range：0～3000（mm/min）

0156	Linear acceleration/deceleration time constant of axes JOG feed	30

Setting range：0～400 （ms）

0157	Exponential acceleration/deceleration time constant of axes JOG feed	30

Setting range：0～400 （ms）

0158	Acceleration clamp time constant of MPG incomplete running	50

Setting range：0～1000 （ms）

0159	The 5th gear's override value in MPG feed(least code increment X's setting value)	1000

Setting range：1～10000

0160	Multiplication coefficient (CMR) of the 1st axis	1

Setting range：1～65536

0161	Multiplication coefficient (CMR) of the 2nd axis	1

Setting range：1～65536

0162	Multiplication coefficient (CMR) of the 3rd axis	1

Setting range：1～65536

0163	Multiplication coefficient (CMR) of the 4th axis	1

Setting range：1～65536

0164	Multiplication coefficient (CMR) of the 5th axis	1

Setting range：1～65536

Appendix

0165	Frequency division coefficient (CMR) of the 1st axis	1

Setting range：1～65536

0166	Frequency division coefficient (CMR) of the 2nd axis	1

Setting range：1～65536

0167	Frequency division coefficient (CMR) of the 3rd axis	1

Setting range：1～65536

0168	Frequency division coefficient (CMR) of the 4th axis	1

Setting range：1～65536

0169	Frequency division coefficient (CMR) of the 5th axis	1

Setting range：1～65536

0170	The 1st axis manual rapid positioning speed	4000

Setting range：0～30000

0171	The 2nd axis manual rapid positioning speed	8000

Setting range：0～30000

0172	The 3rd axis manual rapid positioning speed	8000

Setting range：0～30000

0173	The 4th axis manual rapid positioning speed	8000

Setting range：0～30000

0174	The 5th axis manual rapid positioning speed	8000

Setting range：0～30000

0175	The 1st axis' program name(0:X 1:Y 2:Z 3:A 4:B 5:C)	0

Setting range：0～5

0176	The 2nd axis' program name (0:X 1:Y 2:Z 3:A 4:B 5:C)	1

Setting range：0～5

0177	The 3rd axis' program name (0:X 1:Y 2:Z 3:A 4:B 5:C)	2

Setting range：0～5

0178	The 4th axis' program name (0:X 1:Y 2:Z 3:A 4:B 5:C)	4

Setting range：0～5

0179	The 5th axis' program name (0:X 1:Y 2:Z 3:A 4:B 5:C)	5

Setting range：0～5

Appendix

0180	The 1st axis grid/reference point offset amount	0

Setting range：-50～50

0181	The 2nd axis grid/reference point offset amount	0

Setting range：-50～50

0182	The 3rd axis grid/reference point offset amount	0

Setting range：-50～50

0183	The 4th axis grid/reference point offset amount	0

Setting range：-50～50

0184	The 5th axis grid/reference point offset amount	1

Setting range：-50～50

0185	Mode when machine's Z axis friction compensation（0：disabled，1: Up，2：Down，3：Up and down）	0

Setting range：0～3

0186	Machine's Z axis friction compensation amount（mm）	0.05

Setting range：0～0.5

0187	Z axis backlash compensation condition（default：1）	1

Setting range：0～50

0188	Z axis backlash compensation's accumulated distance（default：0.02）	0.02

Setting range：0～0.5

0189	Reverse precision by backlash compensation (X0.0001)	0.0100

Setting range：0.0001～1.0000 (mm)

Set $\alpha = p(189) \times 0.0001$, in reverse feeding, if the feeding of single servo period is more than α, the backlash compensation begins.

Therefore, in machining outer circle contour with a large radius, in order to make the offset position not to exceed the quadrant, it needs to set a smaller precision. While in machining a curve surface, in order to not to perform backlash compensation in a fixed point of the tool path to form a swollen ridge, it needs to set a larger precision to make the clearance compensation to be distributed in a certain width.

0190	The 1st axis' backlash compensation amount	0.0000

Setting range：

Metric：-0.5～0.5 (mm)
Inch：-0.5～0.5/25.4 (inch)
Rotary axis：-0.5～0.5000 (deg)

0191	The 2nd axis' backlash compensation amount	0.0000

Setting range：

Metric ：-0.5～0.5 (mm)
Inch ：-0.5～0.5/25.4 (inch)
Rotary axis ：-0.5～0.5 (deg)

0192	The 3rd axis' backlash compensation amount	0.0000

Setting range：
Metric：-0.5～0.5 (mm)
Inch：-0.5～0.5/25.4 (inch)
Rotary axis：-0.5～0.5 (deg)

0193	The 4th axis' backlash compensation amount	0.0000

Setting range：
Metric：-0.5～0.5 (mm)
Inch：-0.5～0.5/25.4(inch)
Rotary axis：-0.5～0.5(deg)

0194	The 5th axis' backlash compensation amount	0.0000

Setting range：
Metric ：-0.5～0.5(mm)
Inch ：-0.5～0.5/25.4(inch)
Rotary axis：-0.5～0.5 (deg)

0195	Compensation step width of the 1st axis clearance by fixed frequency	0.0030

Setting range：0～0.5 (mm)

0196	Compensation step width of the 2nd axis clearance by fixed frequency	0.0030

Setting range：0～0.5 (mm)

0197	Compensation step width of the 3rd axis clearance by fixed frequency	0.0030

Setting range：0～0.5 (mm)

0198	Compensation step width of the 4th axis clearance by fixed frequency	0.0030

Setting range：0～0.5 (mm)

0199	Compensation step width of the 5th axis clearance by fixed frequency	0.0030

Setting range：0～0.5 (mm)

0200	Time constant of backlash compensation by ascending and descending	20

Setting range：0～400 （ms）

0201	Backlash compensation mode 0：mode A，1：mode B，2：mode C	0

Setting range：0～2

0203	Output time of reset signal	200

Appendix

Setting range：50～400 (ms)

0204	Digits allowable for M codes	2

Setting range：1～2

0205	Digits allowable for s codes	5

Setting range：1～6

0206	Digits allowable for T codes	4

Setting range：1～4

0207	Delay time of strobe pulse signal MF/SF/TF（n*8）	64

Setting range：16～65535

0208	Acceptable width of M/S/T function completion signal （FIN）（n*8）	64

Setting range：16～65535

0209	Compensation conditions when the machine Z axis friction compensates（default：1.0）	0

Setting range：0～0

0210	Incremental amount for automatic sequence number insertion	10

Setting range：0～1000

0211	Tool offset heading number input by MDI disabled	1

Setting range：0～99

0212	Tool offset numbers input by MDI disabled	99

Setting range：0～99

0214	Error limit of arc radius (mm)	0.05

Setting range：0.0001～0.1000 （mm）

0216	Pitch error compensation number of the 1st axis reference point	0

Setting range：0～9999

0217	Pitch error compensation number of the 2nd axis reference point	0

Setting range：0～9999

0218	Pitch error compensation number of the 3rd axis reference point	0

Setting range：0～9999

0219	Pitch error compensation number of the 4th axis reference point	0

Setting range：0～9999

0220	Pitch error compensation number of the 5th axis reference point	0

Setting range：0～9999

0221	The 1st axis' pitch error compensation amount from the zero return's reverse direction to zero	0

Setting range：-0.9999～0.9999

0222	The 2nd axis' pitch error compensation amount from the zero return's reverse direction to zero	0

Setting range：-0.9999～0.9999

0223	The 3rd axis' pitch error compensation amount from the zero return's reverse direction to zero	0

Setting range：-0.9999～0.9999

0224	The 4th axis' pitch error compensation amount from the zero return's reverse direction to zero	0

Setting range：-0.9999～0.9999

0225	The 5th axis' pitch error compensation amount from the zero return's reverse direction to zero	0

Setting range：-0.9999～0.9999

0226	Pitch error compensation interval of the 1st axis	10

Setting range：0～19999.9998

0227	Pitch error compensation interval of the 2nd axis	0

Setting range：0～9999.9999

0228	Pitch error compensation interval of the 3rd axis	5

Setting range：0～9999.9999

0229	Pitch error compensation interval of the 4th axis	5

Setting range：0～9999.9999

0230	Pitch error compensation interval of the 5th axis	5

Setting range：0～9999.9999

0231	Blacklash compensation of the 1st axis at rapid move	0

Setting range：-0.5～0.5

0232	Blacklash compensation of the 2nd axis at rapid move	0

Setting range：-0.5～0.5

0233	Blacklash compensation of the 3rd axis at rapid move	0

Setting range：-0.5～0.5

0234	Blacklash compensation of the 4th axis at rapid move	0

Setting range：-0.5～0.5

0235	Blacklash compensation of the 5th axis at rapid move	0

Setting range：-0.5～0.5

0240	Gain adjustment data for spindle analog output	1

Setting range：0.98～1.02

0241	Compensation value of offset voltage for spindle analog output	0

Setting range：-0.2～0.2

0242	Spindle speed at spindle orientation, or motor speed at spindle gear shift	50

Setting range：0～9999 (r/min)

0246	Spindle maximum speed to gear 1	6000

Setting range：0～99999 (r/min)

0247	Spindle maximum speed to gear 2	6000

Setting range：0～99999 (r/min)

0248	Spindle maximum speed to gear 3	6000

Setting range：0～99999 (r/min)

0250	Spindle motor speed of gear shifting	50

Setting range：0～1000 (r/min)

0251	Maximum spindle motor speed of shifting	6000

Setting range：0～99999 (r/min)

0254	Axis as counting for surface speed control	0

Setting range：0～4

0255	Spindle minimum speed for constant surface speed control (G96)	100

Setting range：0～9999 (r/min)

0257	Spindle upper limit speed in tapping cycle	6000

Setting range：0～9999 (r/min)

0258	Spindle upper limit speed	6000

Setting range：0～99999 (r/min)

0261	Spindle encoder lines	1024

Appendix

Setting range：0～9999

0262	Spindle motor speed at switch point between gear 1 and gear 2	6000

Setting range：0～9999 (r/min)

0263	Spindle motor speed at switch point between gear 2 and gear 3	6000

Setting range：0～9999 (r/min)

0264	Spindle motor speed at switch point between gear 3 and gear 4	6000

Setting range：0～9999 (r/min)

0266	Limit with vector ignored when moving along outside corner in tool radius compensation C	0

Setting range：0～9999.9999

0267	Wear value's positive/negative limit value input every time in tool offset（mm）	1

Setting range：0～999.9999 (mm)

0268	Maximum error value of tool radius compensation C	0.0010

Setting range：0.0001~0.0100

0269	Helical infeed radius coefficient in groove cycle	1.5

Setting range：0.01~3

0270	Retraction amount of high-speed peck drilling cycle G73	2.0000

Setting range：0～999.9999 (mm)

0271	Reserved space amount of canned cycle G83	2.0000

Setting range：0～999.9999 (mm)

0272	Dwell time for returning to hole top	2

Setting range：0～100 (ms)

0281	Minimum dwell time at the hole bottom	50

Setting range：0～1000 (ms)

0282	Maximum dwell time at the hole bottom	9999

Setting range：1000～9999 (ms)

0283	Override for retraction in rigid tapping	100

Setting range：0～100

Note: N0：44#4=1, the override value is valid.

N0:45#3=1，the set data unit: 10%，max. override value is set to 1000%.

Appendix

0284	Retraction or spacing amount in peck tapping cycle	2

Setting range：0～100 (mm)

0286	Tooth number of spindle side gear (the 1^{st} gear)	1

Setting range：1～999

0287	Tooth number of spindle side gear (the 2^{nd} gear)	1

Setting range：1～999

0288	Tooth number of spindle side gear (the 3^{rd} gear)	1

Setting range：1～999

0289	Gear's teeth number at side of the spindle(the 3^{rd} gear)	1

Setting range：1～999

0290	Gear's teeth number at side of the position encoder (the 1^{st} gear)	1

Setting range：1～999

0291	Gear's teeth number at side of the position encoder (the 2^{nd} gear)	1

Setting range：1～999

0292	Gear's teeth number at side of the position encoder (the 3^{rd} gear)	1

Setting range：1～999

0293	Gear's teeth number at side of the position encoder (the 4^{th} gear)	1

Setting range：1～999

0294	Maximum spindle speed in rigid tapping (the 1^{st} gear)	6000

Setting range：0～9999

0295	Maximum spindle speed in rigid tapping (the 2^{nd} gear)	6000

Setting range：0～9999

0296	Maximum spindle speed in rigid tapping (the 3^{rd} gear)	6000

Setting range：0～9999

0297	Maximum spindle speed in rigid tapping (the 4^{th} gear)	6000

Setting range：0～9999

0298	Linear acceleration/deceleration time constants of spindle and tapping axis (the 1^{st} gear)	200

Setting range：0～9999

0299	Linear acceleration/deceleration time constants of spindle and tapping axis (the 2^{nd} gear)	200

Setting range：0～9999

0300	Linear acceleration/deceleration time constants of spindle and tapping axis (the 3rd gear)	200

Setting range：0～9999

0301	Linear acceleration/deceleration time constants of spindle and tapping axis (the 4th gear)	200

Setting range：0～9999

0302	Time constant of spindle and tapping axis in retraction (the 1st gear)	200

Setting range：0～9999

0303	Time constant of spindle and tapping axis in retraction (the 2nd gear)	200

Setting range：0～9999

0304	Time constant of spindle and tapping axis in retraction (the 3rd gear)	200

Setting range：0～9999

0305	Time constant of spindle and tapping axis in retraction (the 4th gear)	200

Setting range：0～9999

0310	The spindle arrival code speed's speed permissive rate（q）	5

Setting range： 0～1000

0311	Do not send the spindle speed change check alarm's spindle change rate (r)	5

Setting range： 0～1000

0312	Do not send the spindle speed change check alarm's spindle speed variation amount (i)	10

Setting range： 0～9999

0313	Time from coding the spindle change to starting the spindle speed change check (p)ms	1000

Setting range： 0～99999

0320	Spindle clearance in rigid tapping (the 1st gear)	0

Setting range：0～99.9999

0321	Spindle clearance in rigid tapping (the 2nd gear)	0

Setting range：0～99.9999

0322	Spindle clearance in rigid tapping (the 3rd gear)	0

Setting range：0～99.9999

0323	Spindle command multiplication coefficient (CMR) (the 1st	512

Appendix

	gear)	

Setting range：0～9999

0324	Spindle command multiplication coefficient (CMR) (the 2nd gear)	512

Setting range：0～9999

0325	Spindle command multiplication coefficient (CMR) (the 3rd gear)	512

Setting range：0～9999

0326	Spindle command frequency dividing coefficient (CMD) (the 1st gear)	125

Setting range：0～9999

0327	Spindle command frequency dividing coefficient (CMD) (the 2nd gear)	125

Setting range：0～9999

0328	Spindle command frequency dividing coefficient (CMD) (the 3rd gear)	125

Setting range：0～9999

0340	Coding pretreatment block quantity	0

Setting range：0～0

0341	ARM interpolation point buffer size	36

Setting range：0～99999

0342	The 1st axis zero return at low speed	200

Setting range：0～1000

0343	The 2nd axis is zero return at low speed	200

Setting range：0～1000

0344	The 3rd axis zero return at low speed	200

Setting range：0～1000

0345	The 4th axis zero return at low speed	200

Setting range：0～1000

0346	The 5th axis zero return at low speed	200

Setting range：0～1000

0352	Acceleration/deceleration time constant of zero return at high-speed	100

Setting range：3～400

0353	Acceleration/deceleration time constant of zero return at low-speed	30

Setting range：3～100

Appendix

0361	Acceleration/deceleration time constant 1 in thread cutting	20

Setting range：1～200（ms）

0362	Max. acceleration of linear acceleration/deceleration run in thread cutting (mm/s*s)	5000

Setting range：1～10000(ms)

0365	Short-axis acceleration/deceleration time constant in thread run-out	10

Setting range：1～200(ms)

0366	The spindle pulse sampling period in thread turning（ms）	16

Setting range：0～32

0367	Linear acceleration/deceleration time constant in rigid thread turning	40

Setting range：1～400

0368	Thread turning's initial speed	0

Setting range：1～8000

0370	Thread block end's chip time（ms）	0

Setting range：0～9

0371	The 1st axis' reverse positioning allowance	0.0150

Setting range：0～99.9999 (mm)

0372	The 2nd axis' reverse positioning allowance	0.0150

Setting range：0～99.9999 (mm)

0373	The 3rd axis' reverse positioning allowance	0.0150

Setting range：0～99.9999 (mm)

0374	The 4th axis' reverse positioning allowance	0.0150

Setting range：0～99.9999 (mm)

0375	The 5th axis' reverse positioning allowance	0.0150

Setting range：0～99.9999 (mm)

When the set backlash compensation value (P0190---P0193) of an axis is bigger than the reverse positioning allowable error (P0371---P0374) of this axis, the speed at the end point of a single block reduces to minimum speed before this backlash compensation begins. This will make the other axes move a small distance in the backlash compensation period, and that will ensure the resultant path deviates the real path least.

0376	Axes moving sequence to program beginning	12345

Setting range：0～99999

0380	Set the axis which is synchronous with the 4th axis 0:is not synchronous 1:X 2:Y 3:Z	0

Setting range：0～3

0381	Max. permissive error among synchronous axes	200

Setting range：0～10000

0382	Set the double-drive reference position's difference value	0.0000

Setting range：0.0000～2000.0000

0392	Move distance when the servo optimizes	50

Setting range： 0～100

0393	Motion rate when the servo optimizes	2000

Setting range： 0～5000

0394	The coordinate system's 1st axis backup	0

Setting range： -9999.9999～9999.9999

0395	The coordinate system's 2nd axis backup	0

Setting range： -9999.9999～9999.9999

0396	The coordinate system's 3rd axis backup	0

Setting range： -9999.9999～9999.9999

0397	The coordinate system's 4th axis backup	0

Setting range： -9999.9999～9999.9999

0398	The coordinate system's 5th axis backup	0

Setting range： -9999.9999～9999.9999

400	Gain regulation data of the spindle speed analog output	0

Setting range： -0.2～0.2

401	Compensation value of the spindle speed alaong outputting voltage	50

Setting range： 0～9999

402	The spindle speed of the 2nd spindle orientation or gear shifting	50

Setting range： 0～9999

403	Corresponding to the gear 1's spindle max. speed	6000

Setting range： 0～9999

404	Corresponding to the gear 2's spindle max. speed	6000

Setting range： 0～9999

Appendix

405	Corresponding to the gear 3's spindle max. speed	6000

Setting range： 0～9999

406	Corresponding to the gear 4's spindle max. speed	6000

Setting range： 0～9999

407	Motor speed when the 2nd spindle gear shifting	50

Setting range： 0～1000

408	Max. motor speed when the 2nd spindle gear shifting	6000

Setting range： 0～99999

409	The 2nd spindle's upper limit speed	6000

Setting range： 0～99999

410	The 2nd spindle encoder lines	1024

Setting range： 1～999

411	Gear's teeth number at side of the 2nd spindle（the 1st gear）	1

Setting range： 1～999

412	Gear's teeth number at side of the 2nd spindle（the 2nd gear）	1

Setting range： 1～999

413	Gear's teeth number at side of the 2nd spindle（the 3rd gear）	1

Setting range： 1～999

414	Gear's teeth number at side of the 2nd spindle（the 4th gear）	1

Setting range： 1～999

415	Gear's teeth number at side of the 2nd spindle encoder（the 1st gear）	1

Setting range： 1～999

416	Gear's teeth number at side of the 2nd spindle encoder（the 2nd gear）	1

Setting range： 1～999

417	Gear's teeth number at side of the 2nd spindle encoder（the 3rd gear）	1

Setting range： 1～999

418	Gear's teeth number at side of the 2nd spindle encoder（the 4th gear）	1

Setting range： 1～999

419	Spindle motor speed at the 2nd spindle's switch point between gear 1 and gear 2	6000

Setting range：　0～99999（r/min）

420	Spindle motor speed at the 2nd spindle's switch point between gear 2 and gear 3	6000

Setting range：　0～99999（r/min）

421	Spindle motor speed at the 2nd spindle's switch point between gear 3 and gear 4	6000

Setting range：　0～99999（r/min）

4440	M code for Cs contour controllable axis function start	14

Setting range：　0～9999

4441	M code for Cs contour controllable axis function close	15

Setting range：　0～9999

0445	Axis 1 allocating the grating precision	0.001

Setting range：0.0000～10.0000

0446	Axis 2 allocating the grating precision	0.001

Setting range：0.0000～10.0000

0447	Axis 3 allocating the grating precision	0.001

Setting range：0.0000～10.0000

0448	Axis 4 allocating the grating precision	0.001

Setting range：0.0000～10.0000

0449	Axis 5 allocating the grating precision	0.001

Setting range：0.0000～10.0000

0450	Position offset limit value when the 1st axis moving（mm）	10

Setting range：0.0000～99.9999

0451	Position offset limit value when the 2nd axis moving（mm）	10

Setting range：0.0000～99.9999

0452	Position offset limit value when the 3rd axis moving（mm）	10

Setting range：0.0000～99.9999

0453	Position offset limit value when the 4th axis moving（mm）	10

Setting range：0.0000～99.9999

0454	Position offset limit value when the 5th axis moving（mm）	10

Setting range：0.0000～99.9999

0455	Position offset limit value when the 1st axis stopping （mm）	0.5

Setting range：0.0000～9.9999

0456	Position offset limit value when the 2nd axis stopping （mm）	0.5

Setting range：0.0000～9.9999

0457	Position offset limit value when the 3rd axis stopping （mm）	0.5

Setting range：0.0000～9.9999

0458	Position offset limit value when the 4th axis stopping （mm）	0.5

Setting range：0.0000～9.9999

0459	Position offset limit value when the 5th axis stopping （mm）	0.5

Setting range：0.0000～9.9999

0460	Non-montone permissive value of the compound turning cycle（Z）	0.1

Setting range：0.0000～9999.9999

0461	Non-montone permissive value of the compound turning cycle（X）	0.1

Setting range：0.0000～9999.9999

0462	Feed amount of compound turning fixed cycle G71, G72	0

Setting range：-9999.0000～9999.9999

0463	Feed amount of compound turning fixed cycle G71, G72	0

Setting range：-9999.0000～9999.9999

0465	Tool retraction distance of compound turning fixed cycle G73（X）	0

Setting range：-9999.0000～9999.9999

0466	Tool retraction distance of compound turning fixed cycle G73（Z）	0

Setting range：-9999.0000～9999.9999

0467	Division times of compound turning fixed cycle G73	0

Setting range：-9999～9999

0468	Return amount of compound turning fixed cycle G74，G75	0

Setting range：-9999.0000～9999.9999

0469	Least tool infeed amount of compound turning fixed cycle G76	0

Setting range：-9999.0000～9999.9999

0470	Finishing allowance of compound turning fixed cycle G76	0

Setting range：-9999.0000～9999.9999

0471	Finishing allowance's repetitive times of G76	0

Setting range：-9999～9999

0472	Too nose angle of compound turning fixed cycle G76	0

Setting range：9999～9999

0473	Chamfering (setting value*0.1 lead) of turning thread cycle（G92,G76）	0

Setting range：0～127

0474	Cutting angle of turning thread cycle G76	0

Setting range：0～89

0475	Whether to execute the last roughing path when the compound turning fixed cycle G71，G72 is type II	0

Setting range：0～1

0476	Idle stroke amount of compound turning fixed cycle G71's cutting feed start position	1

Setting range：0～5

0477	Whether the slot bottom's tool retraction of compound turning fixedcycle G71 to execute 45 degree tool retraction	0

Setting range：0～1

0480	Execute the controlled axis of No.1 position switch function	0

Setting range：0～5

0481	Execute the controlled axis of No.2 position switch function	0

Setting range：0～5

0482	Execute the controlled axis of No.3 position switch function	0

Setting range：0～5

0483	Execute the controlled axis of No.4 position switch function	0

Setting range：0～5

0484	Execute the controlled axis of No.5 position switch function	0

Setting range：0～5

0485	Execute the controlled axis of No.6 position switch function	0.

Setting range：0～5

0486	Execute the controlled axis of No.7 position switch function	0

Setting range：0.～5

0487	Execute the controlled axis of No.8 position switch function	0

Setting range：0～5

0488	Execute the controlled axis of No.9 position switch function	0

Setting range：0～5

0489	Execute the controlled axis of No.10 position switch function	0

Setting range：0～5

0490	Max. value of No. 01 position switch operation range	0.0000

Setting range：-9999.9999～9999.9999

0491	Max. value of No. 02 position switch operation range	0.0000

Setting range：-9999.9999～9999.9999

0492	Max. value of No. 03 position switch operation range	0.0000

Setting range：-9999.9999～9999.9999

0493	Max. value of No. 04 position switch operation range	0.0000

Setting range：-9999.9999～9999.9999

0494	Max. value of No. 05 position switch operation range	0.0000

Setting range：-9999.9999～9999.9999

0495	Max. value of No. 06 position switch operation range	0.0000

Setting range：-9999.9999～9999.9999

0496	Max. value of No. 07 position switch operation range	0.0000

Setting range：-9999.9999～9999.9999

0497	Max. value of No. 08 position switch operation range	0.0000

Setting range：-9999.9999～9999.9999

0498	Max. value of No. 09 position switch operation range	0.0000

Setting range：-9999.9999～9999.9999

0499	Max. value of No.10 position switch operation range	0.0000

Setting range：-9999.9999～9999.9999

0500	Least value of No. 01 position switch operation range	0.0000

Setting range：-9999.9999～9999.9999

0501	Least value of No. 02 position switch operation range	0.0000

Setting range：-9999.9999～9999.9999

0502	Least value of No. 03 position switch operation range	0.0000

Setting range：-9999.9999～9999.9999

0503	Least value of No. 04 position switch operation range	0.0000

Setting range：-9999.9999～9999.9999

0504	Least value of No. 05 position switch operation range	0.0000

Setting range：-9999.9999～9999.9999

0505	Least value of No. 06 position switch operation range	0.0000

Setting range：-9999.9999～9999.9999

0506	Least value of No. 07 position switch operation range	0.0000

Setting range：-9999.9999～9999.9999

0507	Least value of No. 08 position switch operation range	0.0000

Setting range：-9999.9999～9999.9999

0508	Least value of No. 09 position switch operation range	0.0000

Setting range：-9999.9999～9999.9999

0509	Least value of No.10 position switch operation range	0.0000

Setting range：-9999.9999～9999.9999

0530	Linear axis' axis number in polar coordinate interpolation（0：none 1~5：the 1st axis~the 5th axis）	0.0000

Setting range：0～5

0531	Rotary axis' axis number in polar coordinate interpolation（0：none 1~5：the 1st axis~the 5th axis）	0.0000

Setting range：0～5

0532	Automatic rate's permissive rate % in polar coordinate interpolation（0~100）	90

Setting range：0～100

0533	Imaginary axis direction error's compensation amount in polar coordinate interpolation	0.0000

Setting range：-999.0000～999.0000

0540	System axis number controlled by PMC axis（0：none 1~5：the 1st axis~the 5th axis）	0

Setting range：0～5

0541	Least unit of data controlled by PMC axis（0.0001~360.0）	0.0010

Setting range：0.0001～360.0000

0542	System axis number controlled by the 2nd PMC axis（0：none 1~5：the 1st axis~the 5th axis）	0

Setting range：0～5

Appendix

0543	Least unit of data controlled by the 2 PMC axi （0.0001~360.0）	0.001

Setting range：0.0001～360.0000

Appendix 2 Alarm List

Alarm No.	Content	Remark
0000	Parameter for cutting off power once is modified	
0001	File open fail	
0002	Data input overflow	
0003	Program number already in use	
0004	There is no address but figure or character "-" at the beginning of the block. Modify the program	
0005	There is no appropriate data but another address or EOB code behind the address. Modify the program	
0006	Sign "-" input is wrong (One or more "-" signs are input behind the address where negative sign can not be used). Modify the program	
0007	Decimal point "." input is wrong (One or more "." signs are input in the address where the sign can not be used). Modify the program	
0008	The program file is too large. Please use CNC to transmit it	
0009	Illegal address input. Modify the program.	
0010	Have used the unusable G doe or have commanded the G code which has no the function. Modify the program	
0011	Feedrate is not specified or it is wrong in cutting feed. Modify the program	
0012	Disk space is not enough. Setup or add file is not allowed	
0013	The program files are up to the upper limit. New programs can not be setup	
0015	Exceed the number of simultaneously controlled axes	
0016	Current pitch compensation beyond range	
0017	No authority to modify	
0018	Dummy variable and local variable are not allowed to modify. G10 only to modify parameter of user grade	
0020	In circular interpolation (G02 or G03), the distance between the start point and the circle center is not equal to the distance between the end point and the circle center. The value beyond the one specified by parameter 214	
0021	In circular interpolation, axes which are not in the selected planes (G17, G18, G19) are specified. Modify the program.	
0022	In circular interpolation, R (radius), I, J and k (distance from the start point to the center) are not be specified	
0023	In circular interpolation, I, J, K and R are specified together	
0026	Unsupported file format. It is too large or with above 1024 bytes	
0028	In plane selection codes, two or more axes are specified at the same direction. Modify the program	
0031	When G10 sets an offset amount, a workpiece coordinate system and an external workpiece coordinate system, P value is too big or is not specified	
0032	When G10 sets the offset amount or the system variable writes an offset amount, the offset amount is too big or is not specified. Modify the program	
0033	The intersecting point of offset C or chamfer is not confirmed. Modify the program	
0034	Set-up or offset cancel are not allowed in circular command. Modify the program	

Appendix

0036	In tool compensation mode, have commanded the skip cutting （G31）。Modify the program	
0037	The plane selected by G17, G18 or G19 is changed in tool compensation C. Modify the program	
0038	In tool compensation C, overcut appears because the arc start point or end point is consistent with the arc center. Modify the program	
0039	Tool nose positioning error in tool compensation C	
0040	Cannot convert the workpiece coordinate system in tool compensation C. Cancel the tool compensation before changing the workpiece coordinate	
0041	Interference occurs in tool compensation C leads overcutting. Modify the program	
0042	Ten blocks with stop tool code are specified in tool compensation mode. Modify the program	
0043	No authority. Change it in password page	
0044	In canned cycle, one of codes in G27，G28，G29，G30 is specified. Modify the program	
0045	In canned cycle G73/G83, cutting depth (Q) is not specified or it is 0. Modify the program	
0046	In 2nd, 3rd, 4th reference return codes, codes besides P2, P3 and P4 is specified	
0050	Move it when changing canned cycle mode	
0051	Have specified a mistaken movement operation or movement distance in a block following the cornering or chamfering. Modify the program	
0053	Wrong command format for rounding or chamfering. Modify the program	
0055	Chamfering movement failed	
0056	M99 shall not in the same block with macro code G65. Modify the program	
0057	File input failed. Cut off the power and reset it	
0058	In block of rounding or chamfering, specified axis is not in the selected plane. Modify the program	
0059	Program number is not found in external program retrieving or it is edited in background. Check program number or external signal, or stop background editing	
0060	Specified sequence number is not found in retrieving. Check sequence number	
0061	The 1st axis is not on the reference point	
0062	The 2nd axis is not on the reference point	
0063	The 3rd axis is not on the reference point	
0064	The 4th axis is not on the reference point	
0065	The 5th axis is not on the reference point	
0066	Cancel canned cycle mode before inputting parameter (G10)	
0067	G10 does not support the set format	
0068	Parameter switch is not switched on	
0069	U-disk operation page should be closed when machining	
0070	Insufficient memory. Delete unneeded programs and try it again	
0071	Have not found the required searched address or have not found the specified program number’s program in the program search. Check the data	
0072	Stored programs exceed 400. Delete unnecessary programs	
0073	Program number already in use. Change the program number or delete	

	unneeded program	
0074	Program number beyond the range 1-99999). Change the program number	
0075	To register a protected program number	
0076	Address P (program name) is not specified in block M98. Modify the program	
0077	Program nesting exceed 5 layers	
0078	In blocks M98, G65, program name specified by address P is not found or macro program called by M06 does not exist	
0079	CNC expires the using date. Please contact the supplier	
0080	Input data is wrong, Max. speed is smaller than Min. speed or Min. speed is bigger than Max. speed	
0081	Subprogram can not be called. Modify the program	
0084	Overtime or short circuit occurs in key	
0085	Overflow occurs when data is transmitted to memory by series port. Baud rate setting or I/O equipment is wrong	
0086	Planes can not be shifted in canned cycle mode	
0087	Alarm NO.0087~0091 are for reference point return unfinished (starting point of reference return is too close to the reference point or the speed is too slow).	
0092	G27(check for reference return) code can not return to the reference point	
0093	Motor model error	
0098	After power-on or emergency stop, when the program with G28, program restarts without executing reference return	
0100	On parameter (setting) screen, PWE (parameter input is active) is set to 1. Restart CNC after setting it to 0.	
0101	Memory data disordered after power off, please ensure correct location	
0102	Driver motor does not match CNC	
0103	Bus communication error. Please check reliability of the cable	
0104	Machine zero point setting error	
0105	Time-out error while data is being fetched	
0106	Drive unit is not consistent with gear ratio of servo parameter	
0107	Drive unit parameter is not consistent with servo unit parameter	
0108	Please insert U-disk	
0110	Position data exceeds the allowed range. Please reset	
0111	Calculated result exceeds the allowed range (-10^{47} to -10^{-29}, 0 and 10^{-29} to 10^{47})	
0112	Zero (including tan900) is specified as a divisor	
0113	Unusable functional code is specified in user macro program. Modify the program	
0114	G39 format error. Modify the program	
0115	Variable value can not be specified. O, N can not be specified as variables in user macro program	
0116	A variable is on the left of the assignment statement, while value assignment to it is not allowed. Modify the program	
0117	G10 online modification is not supported by this parameter. Please modify the program	
0118	Nest exceeds the upper limit (5). Modify the program	

Appendix

0119	Codes M00,M01,M02,M30,M98,M99,M06 can not in a same block with other M codes	
0120	Part of setting is restored	
0121	Machine coordinates and encoder feedback values exceed setting value of error	
0122	Called nests of macro program exceed 5 layers. Modify the program	
0124	Program end illegally, without M30, M02, M99 or end sign. Modify the program	
0125	Macro program format error. Modify the program	
0126	Program cycle failure. Modify the program	
0127	NC coexists with user macro code statement. Modify the program	
0128	Sequence number in branch code is not at the range 0-99999, or the number is not found. Modify the program	
0129	The address of argument assignment. Modify the program	
0130	PLC axis control code is input to the axis controlled by CNC, or opposite. Modify the program	
0131	5 or more external alarm signals occur. Check the ladder diagram	
0132	The alarm of the external alarm signal does not exist. Check PLC	
0133	The system does not support axis code. Modify the program	
0135	The index angle of index table is not an ange unit's multiple. Modify the program	
0136	In index table's indexing, it with B axis commands another axis. Modify the program	
0137	Sequence number to be transferred by skip code is in loop body. Modify the program	
0138	Cycle statement is wrong or skip code enters loop body. Modify the program	
0139	PLC axis change disabled. Modify the program	
0140	Sequence number does not exist	
0142	The proportion scaling override exceeding 1-999999 is commanded. Correct the proportion scaling setting	
0145	G28 is commanded when a reference point is not established	
0148	The automatic corner deceleration speed exceeds the the judgement angle range's setting value. Modify the parameter	
0150	Mistaken modal command	
0151	Blocks specified by P and Q have commanded the unusable commands in the compound fixed cycle	
0160	Arc programming only by R in polar system	
0161	Reference point, plane selection or direction-related codes can not be executed in polar coordinate mode	
0165	Please specify revolution, scaling or G10 codes in a single block	
0166	No axis specified in reference return	
0167	Intermediate point coordinate too large	
0168	The min. dwell time at the hole bottom should be shorter than the max. dwell time	
0170	Tool radius compensation is not cancelled while entering or exiting subprogram	
0172	P is not an integer or less than 0 in a block calling subprogram	
0173	Subprogram call should be less than 9999	

0175	Canned cycle can only be executed in G17 plane	
0176	Spindle speed is not specified before rigid tapping	
0177	Spindle orientation is not supported by IO control in G76 code. Modify the program or parameter	
0178	Spindle speed is not specified before the canned cycle starts	
0181	Illegal M code	
0182	The spindle speed is too big or small	
0183	Illegal T code	
0184	The selected too exceed its range	
0189	U is too small, U should bigger than or equal to tool radius	
0190	V is too small or it is undefined. V should be bigger than 0	
0191	W is too small or it is undefined. W should be bigger than 0	
0192	Q is too small or it is undefined. Q should be bigger than 0	
0193	I is undefined or it is 0	
0194	J is undefined or it is 0	
0195	D is undefined or it is 0	
0198	In constant surface cutting speed control, specified axis error (see parameter No.254)	
0199	Macro code modification program is not defined	
0200	In rigid tapping, illegal S code	
0204	M29 should be specified in G80 mode. Modify the program	
0206	Plane shifting is specified in rigid tapping. Modify the program	
0208	This code can not be executed in G10 mode. Please cancel G10 mode first	
0210	The program restart file name is not consistent. Please select a correct file name	
0212	Chamfer or R is specified, or other axis is specified in plane. Modify the program	
0213	Tool changing macro program does not support G31 skip. Modify the program	
0214	Tool changing macro program does not support skip operation	
0215	Tool changing macro program does not support modifying coordinate system and tool compensation dynamically	
0219	M06 Tool magazine is not used (parameter is not opened). Tool changing code can not be used	
0220	Metric/inch switching is not supported by scaling, revolution and polar coordinate mode	
0221	Metric/inch switching is not supported by tool changing macro program	
0224	Reference return is not performed before auto run started	
0231	There are following errors in the parameter input's specified format error: 1) N or R is not input 2) Parameter number is not defined 3) Address P is not defined in bit parameter input L50 4) N，P，R exceed the range	
0232	3 or more axes are specified as helical interpolation axis	
0233	Device connected to RS-232-C is being used	

Appendix

0235	Specified record end sign (%)	
0236	Parameter setting of program restart is wrong	
0237	The command's decimal point which must be specified is not specified	
0238	The same address in one block exists many times or two or more G codes in the same group are in one block	
0239	An illegal G code is specified in pre-reading control mode. In pre-reading control mode, dividing spindle is specified, max. cutting feeding parameter is set to 0 and interpolation pro-acceleration/deceleration parameter is set to 0. Set the parameter correctly	
0241	MPG pulse is abnormal	
0242	Bus connection error	
0250	Axis name repeated, please modify parameter NO.175~179	
0251	Emergency stop alarm, perform zero return again after canceling the alarm	
0252	Program ends illegally (CNC transmission speed is low, please reduce feedrate)	
0261	Pulse code of DSP interpolation axis is too big. Perform zero return again after reset	
0262	DSP is not started. Please power on again	
0263	DSP parameter setting error	
0264	DSP alarm. Data is too big	
0265	DSP alarm. The bus can not be connected or bus initialization failure	
0266	Speed of DSP interpolation axis exceeds 200M/MIN. Perform zero return again after reset	
0267	DSP initial sign (5555) is abnormal. Perform zero return again after reset	
0268	DSP pulse output volume per revolution is too big. Perform zero return again after reset	
0269	DSP internal alarm. Perform zero return again after reset	
0270	Length of DSP equally distributed interpolation point is too small	
0271	DSP received interpolation data is too small. Perform zero return again after reset	
0272	DSP received undistinguishable G code	
0273	DSP hardware data interchange is abnormal (codes)	
0274	DSP hardware data interchange is abnormal (data)	
0275	In high-speed mode, interpolation multiple is 0	
0280	Perform axes zero return before using tool setting function	
0281	Switch to [SET] [Halving] interface before using tool setting function	
0282	Please check whether toolsetting gauge is installed or parameter 1.6 is set	
0283	Z axis exceeds safety position， please check toolsetting gauge or tool length setting	
0286	Automatic tool length measurement is wrong. Please measure it again	
0401	The drive alarm 01：the servo motor speed exceeds its setting value	
0402	The drive alarm 02：the main circuit's power supply voltage is too high	
0403	The drive alarm 03：the main circuit's power supply voltage is too low	
0404	The drive alarm 04：the numerical value of position deviation counter exceeds its setting value	

0405	The drive alarm 05：the motor's temperature is too high	
0406	The drive alarm 06：the speed regulator saturates for long time	
0407	The drive alarm 07：CCW, CW drive inhibiting input are OFF	
0408	The drive alarm 08：the numerical value's absolute value of position deviation counter exceeds 230	
0409	The drive alarm 09：the encoder signal is mistaken	
0410	The drive alarm 10：control power supply ±15V is low	
0411	The drive alarm 11：IPM faults	
0412	The drive alarm 12：the motor's current is too big	
0413	The drive alarm 13：the servo driver and motor are overload(instantaneous overheat)	
0414	The drive alarm 14：the braking circuit fault	
0415	The drive alarm 15：encoder counter fault	
0420	The drive alarm 20：EEPROM error	
0430	The drive alarm 30：encoder Z pulse error	
0431	The drive alarm 31：encoder UVW signal error or it does not match encoder	
0432	The drive alarm 32：UVW with all high level or with all low level	
0433	The drive alarm 33：communication interrupted	
0434	The drive alarm 34：encoder speed is abnormal	
0435	The drive alarm 35：encoder state is abnormal	
0436	The drive alarm 36：encoder counter is abnormal	
0437	The drive alarm 37：single circle number of encoder overflow	
0438	The drive alarm 38：encoder counter overflow	
0439	The drive alarm 39：encoder battery alarm	
0440	The drive alarm 40：encoder battery's power shortage	
0441	The drive alarm 41：motor model dismatched	
0442	The drive alarm 42：absolute position data abnormal alarm	
0443	The drive alarm 43：encoder EPPROM check alarm	
0449	Ethernet initialization failure. Please check hardware	
0450	Drive unit is disconnected. Please check whether connection of hardware is correct	
0451	The 1^{st} axis driver alarm	
0452	The 2^{nd} axis driver alarm	
0453	The 3^{rd} axis driver alarm	
0454	The 4^{th} axis driver alarm	
0455	The 5^{th} axis driver alarm	
0456	Spindle driver alarm	
0460	Exceed the machining cycle processed by G71, G72, G73, G76, G90, G92, G94	
0466	The path is non-monotonous：X direction	
0467	The path is non-monotonous：Z direction	

Appendix

0500	The 1st axis soft limit - direction overtravel:(manual or MPG +direction movement release).	
0501	The 1st axis soft limit - direction overtravel:(manual or MPG -direction movement release).	
0502	The 2nd axis soft limit - direction overtravel:(manual or MPG +direction movement release).	
0503	The 2nd axis soft limit - direction overtravel:(manual or MPG -direction movement release).	
0504	The 3rd axis soft limit - direction overtravel:(manual or MPG +direction movement release).	
0505	The 3rd axis soft limit - direction overtravel:(manual or MPG -direction movement release).	
0506	The 4th axis soft limit - direction overtravel:(manual or MPG +direction movement release).	
0507	The 4th axis soft limit - direction overtravel:(manual or MPG -direction movement release).	
0508	The 5th axis soft limit - direction overtravel:(manual or MPG +direction movement release).	
0509	The 5th axis soft limit - direction overtravel:(manual or MPG -direction movement release).	
0510	The 1st axis hard limit - direction overtravel:(manual or MPG +direction movement release).	
0511	The 1st axis hard limit - direction overtravel:(manual or MPG -direction movement release).	
0512	The 2nd axis hard limit - direction overtravel:(manual or MPG +direction movement release).	
0513	The 2nd axis hard limit - direction overtravel:(manual or MPG -direction movement release).	
0514	The 3rd axis hard limit - direction overtravel:(manual or MPG +direction movement release).	
0515	The 3rd axis hard limit - direction overtravel:(manual or MPG -direction movement release).	
0516	The 4th axis hard limit - direction overtravel:(manual or MPG +direction movement release).	
0517	The 4th axis hard limit - direction overtravel:(manual or MPG -direction movement release).	
0518	The 5th axis hard limit - direction overtravel:(manual or MPG +direction movement release).	
0519	The 5th axis hard limit-direction overtravel: (manual or MPG -direction movement release).	
0600	The operation keyboard is not connected. Please check its connection cable	
1001	Address of relay or coil is not set	
1002	Function code of input code does not exist	
1003	Function code COM is not used correctly. Corresponding relationship between COM and COME is wrong, or function code is used between COM and COME	
1004	User ladder beyond the maximum permissible linage or step number. Reduce NET number	
1005	Function commands END1，END2 do not exist；errors in END1，END2；or END1，END2’s sequence is not correct	
1006	Illegal output in the network. Please check the output format	
1007	PLC communication failure due to hardware failure or system interruption. Please contact with the system device manufacture	
1008	Functional code is not linked correctly	
1009	Network horizontal line is not linked	

Appendix

1010	Editing NET losses due to power-off in ladder editing	
1011	Address or data format is not the one specified by this function. Input it again	
1012	Address or data is wrongly input. Input it again	
1013	Illegal character or data defined	
1014	CTR address repeated. Reselect other unused TMR addresses	
1015	Functional code is wrongly used. Correspondence between JMP and LBL is wrong. JMP is used again between JMP and LBL	
1016	Incomplete network structure. Change the ladder diagram	
1017	Unsupported network exists. Change the ladder diagram	
1019	TMR address repeated. Reselect other unused TMR addresses	
1020	No parameter in functional code. Input the legal parameters	
1021	PLC stops automatically by CNC when the PLC executes overtime. Please check the ladder logic and resolve the endless loop or the overmuch repetitive calls	
1022	Names of functional code missing. Please input the name of functional codes	
1023	Address or constant of functional code parameter is out of range	
1024	Unnecessary relay or coil exists. Delete the unnecessary connection	
1025	Functional code output wrongly	
1026	NET link linage beyond the supported range. Change the ladder diagram	
1027	The same output address is used in another place. Reselect the unused output address	
1028	File format wrong	
1029	File losses from ladder diagram being used	
1030	False vertical line in network. Delete the vertical line	
1031	Message data area is full. Please reduce COD code data list capacity	
1032	The first level of ladder diagram is too large to complete execution on time. Reduce the first level of ladder diagram	
1033	SFT codes beyond the max. allowed number. Please reduce the usage amount	
1034	Functional code DIFU/DIFD address is repeated. Please reselect the address	
1039	Code or network beyond executable area. Please clear them	
1040	Functional code CALL or SP is wrongly used. Correspondence between CALL and SP or between SP and SPE is wrong. SP functional code is used again between SP and SPE or SP is set before using END2	
1041	Level conducting line in parallel with node network	
1042	PLC system parameter file is not loaded	

Appendix