This user manual describes all items concerning the operation of the system in detail as much as possible. However, it is impractical to give particular descriptions of all unnecessary and/or unavailable operations of the system due to the manual content limit, pstract specific operations and other causes. Therefore, the operations not specified herein shall be considered impossible or unallowable.

FOREWORD

Dear user,

We are really grateful for your patronage and purchase of this **C1000T CNC system** made by CNCmakers Limited.

The user manual describes the programming, operation, installation and connection of this **C1000T CNC system**. Please read it carefully before operation in order to get the safe and effective working.

Warning

⚠ This system can only be operated by authorized and qualified personnel as improper operations may cause accidents.

Please carefully read this user manual before use!

Note: The power supply installed on (in) the cabinet is exclusive to CNCmakers Limited'S CNC systems.

The power supply form is forbidden to be used for other purposes. Otherwise, there may be extreme danger!

This user manual shall be kept by final user.

Notes

■ Delivery and storage

1. Packing box over 6 layers in pile is unallowed.
2. Never climb the packing box, neither stand on it, nor place heavy objects on it.
3. Do not move or drag the product by the cables connected with it.
4. Forbid collision or scratch to the panel and displayer.
5. Packing box should be protected from damping, insolation and raining.

■ Open packing box to check

1. Ensure things in packing box are the required ones.
2. Ensure the product is not damaged in delivery.
3. Ensure the parts in packing box are in accordance to the order.
4. Contact us in time if the product type is inconsistent with the order, there is short of accessories, or product damage in delivery.

■ Connection

1. Only qualified persons can connect the system or check the connection.
2. The system must be earthed, its resistance must be less than 4 Ω and the ground wire cannot be replaced by zero wire.
3. Connection must be correct and firm to avoid the product to be damaged or other unexpected result.
4. Connect with surge diode in the specified direction to avoid the damage to the system.
5. Switch off power supply before pulling out plug or opening electric cabinet.

■ Troubleshooting

1. Switch off power supply before troubleshooting or changing components.
2. Troubleshoot and then startup the system when there is short circuit or overload.
3. Do not switch on or off it frequently and an interval is 1 minute at least after the system is powered on again.

Announcement！

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

Warning！

Please read this user manual and a manual from machine builder completely before installation, programming and operation; do operate the system and machine according to user manuals, otherwise it may damage the system, machine, workpiece and even injure the operator.

Cautions！

Functions, technical indexes described in this user manual are only for the system. Actual functions and technical performance of machine tool with this CNC system are determined by machine builder's design, so refer to its user manual.

The system is employed with integrated machine control panel and the keys on machine control panel are defined by PLC program. Functions of keys in this user manual are for standard PLC program. Please notice it!

Refer to user manual from machine manufacturer about functions and meanings of keys on machine control panel.

All specification and designs are subject to change without further notice.

Volume Ⅰ Programming

Technical Specification, Product Type, Command and Program Format

Volume Ⅱ Operation

CNCmakers Limited C1000T Operation Use

Volume Ⅲ Installation and Connection

C1000T CNC Installation, Connection and Setting

Appendix

CNC Ladder Function Allocation, Alarm Message Table

Safety Responsibility

Manufacturer's safety responsibility

——The manufacturer should be responsible for the cleared or the controlled safety in the design and the structure of the CNC system and the accessories.

——The manufacturer should be responsible for the CNC system and the accessories.

——The manufacturer should be responsible for the message and the suggestion for the user.

User's safety responsibility

——The user should study and train the system safety operation, master the safety operation content.

——The user should be responsible for the danger caused by increasing, changing or modifying the CNC system, the accessories by itself.

——The user should be responsible for the danger because of the mistaken operation, regulation, maintenance, installation and storage.

CONTENTS

Volume Ⅰ Programming

Volume Ⅱ Operation

Volume Ⅲ Connection

Appendix

Volume Ⅰ Programming

CHAPTER 1 PROGRAMMING

1.1 C1000T introduction

1.1.1 Product introduction

C1000T can control 5 feed axes(including C axis), 2 analog spindles, 1ms high-speed interpolation, 0.1μm control precision, which can obviously improve the machining efficiency, precision and surface quality.

CNCmakers Limited

X, Z, Y, 4th, 5th ; axis name and axis type of Y, 4th, 5th can be defined

1ms interpolation period, control precision 1μm, 0.1μm

Max. speed 60m/min (up to 24m/min in 0.1μm)

Adapting to the servo spindle to realize the spindle continuously positioning, rigid tapping, and the rigid thread machining

Built-in multi PLC programs, and the PLC program currently running can be selected

Statement macro command programming, macro program call with parameter

Metric/inch programming, automatic toolsetting, automatic chamfer, tool life management function

Chinese, English display can be selected by parameters.

USB interface, U disc file operation, system configuration and software

2-channel 0V ~ 10V analog voltage output, two-spindle control

1-channel MPG input, MPG function

36 input signals and 36 output signals

Appearance installation dimension, and command system are compatible with C1000T

1.1.2 Technical specification

Controllable axes

Controllable axes: 5 (X, Z, Y , 4th,5th)

Link axes：4

Feed axis function

Least input unit: 0.001mm (0.0001inch) and 0.0001mm (0.00001inch)

Least command unit：0.001mm (0.0001inch) and 0.0001mm (0.00001inch)

Position command range: ±99999999× least command unit

Rapid traverse speed : max. speed 60m/min in 0.001mm command unit

Rapid override: F0, 25%, 50%, 100%

Feedrate override: 0～150% 16 grades to tune

Interpolation mode: linear interpolation, arc interpolation(three-point arc interpolation), thread interpolation and rigid tapping

Automatic chamfer function

Thread function

General thread(following spindle)/rigid thread

Single/multi metric, inch straight thread, taper thread, end face thread, constant pitch thread and variable pitch thread

Thread run-out length, angle, speed characteristics can be set

Thread pitch: 0.01mm～500mm or 0.06 tooth/inch～2540 tooth/inch

Acceleration/deceleration function

Cutting feed: front acceleration/deceleration linear, front acceleration/deceleration S back acceleration/deceleration linear,back acceleration/deceleration exponent

Rapid traverse: linear,S type

Thread cutting: linear, exponential

Initial speed, termination speed, time of acceleration/deceleration can be set by parameters.

Spindle function

2-channel 0V～10V analog voltage output, two-spindle control

1-channel spindle encoder feedback, spindle encoder line can be set (100p/r～5000p/r)

Transmission ratio between encoder and spindle: (1～255) : (1～255)

Spindle speed: it is set by S or PLC, and speed range: 0r/min～9999r/min

Spindle override: 50%～120% 8 grades tune

Spindle constant surface speed control

Rigid tapping

Tool function

Tool length compensation

Tool nose radius compensation (C)

Tool wear compensation

Tool life management

Tool setting mode: fixed-point tool setting, trial-cut tool setting, reference point return tool setting, automatic tool setting

Tool offset execution mode: modifying coordinate mode, tool traverse mode

Precision compensation

Backlash compensation

Memory pitch error compensation

PLC function

Two-level PLC program，up to 5000 steps，the 1st program refresh period 8ms

PLC program communication download

PLC warning and PLC alarm

Many PLC programs（up to 20PCS）, the PLC program currently running can be selected

Basic I/O：18 input signals /18 output signals

Man-machine interface

8.0″ wide screen LCD，resolution: 800X600

Chinese, English display

Planar tool path display

Real-time clock

Operation management

Operation mode: edit, auto, MDI, machine zero return, MPG/single, manual, program zero return

Multi-level operation privilege management

Alarm record

Program edit

Program capacity: 56MB，400 programs（including subprograms and macro programs）

Edit function: program/block word search, modification, deletion,copying,pasting

Program format: ISO command, statement macro command programming, relative coordinate, absolute coordinate and compound coordinate programming

Program call: macro program call with parameter, 4-level program built-in „

Communication function

RS232：two-way transmitting part programs and parameters, PLC program, system software serial upgrade

USB：U file operation, U file directly machining, PLC program, system software U upgrade

Safety function

Emergency stop

Hardware travel limit

Software travel check

Data backup and recovery

G command table

Table 1-1

Command	Function	Command	Function	Command	Function
G00	Rapid traverse (positioning)	G40	Tool nose radius compensation cancle	G96	Constant surface speed control
G01	Linear interpolation	G41	Tool nose radius compensation left	G97	Constant surface speed control cancel
G02	CW arc interpolation	G42	Tool nose radius compensation right	G98	Feed per minute
G03	CCW arc interpolation	G50	Floate workpiece coordinate system	G99	Feed per revolution
G04	Dwell, exact stop	G54~G59	Workpiece coordinate system setting		
G17	Plane selection command	G65	Macro command non-modal call		
G18	Plane selection command)	G71	Axial roughing cycle		
G19	Plane selection command	G72	Radial roughing cycle		
G10	Data input mode ON	G73	Closed cutting cycle		
G11	Cancle data input mode	G74	Axial grooving cycle		
G20	Input in inch	G75	Radial grooving cycle		
G21	Input in metric	G76	Multiple thread cutting cycle		
G28	Automatic return machinel zero point	G80	Rigid tapping state cancel		
G30	2nd, 3rd, 4th reference point	G84	Axial rigid tapping		
G31	Skip function	G88	Radial rigid tapping		
G32	Constant pitch thread cutting	G90	Axial cutting cycle		
G33	Z tapping cycle	G92	Thread cutting cycle		
G34	Thread cutting with variable lead	G94	Radial cutting cycle		

1.1.3 Environment and conditions

C1000T storage delivery, working environment as follows:

Table 1-2

Item	Working conditions	Storage delivery conditions
Ambient temperature	0℃～45℃	-40℃～+70℃
Ambient humidity	≤90%(no freezing)	≤95%(40℃)
Atmosphere pressure	86 kPa～106 kPa	86 kPa～106 kPa
Altitude	≤1000m	≤1000m

1.1.4 Power supply

C1000T can normally run in the following AC input power supply.
Voltage: within(0.85～1.1)×rated AC input voltage (AC 220V); Frequency: 49Hz～51Hz continuously changing

1.1.5 Guard

C1000T guard level is not less than IP20.

1.2 CNC system of machine tools and CNC machine tools

CNC machine tool is an electro-mechanical integrated product, composed of Numerical Control Systems of Machine Tools, machines, electric control components, hydraulic components, pneumatic components, lubricant, cooling and other subsystems (components), and CNC systems of machine tools are control cores of CNC machine tools. CNC systems of machine tools are made up of computerized numerical control(CNC), servo (stepper) motor drive devices, servo (or stepper) motor etc.

Operational principles of CNC machine tools: according to requirements of machining technology, edit user programs and input them to CNC, then CNC outputs motion control commands to the servo (stepper) motor drive devices, and last the servo (or stepper) motor completes the cutting feed of machine tool by mechanical driving device; logic control commands in user programs to control spindle start/stop, tool selections, cooling ON/OFF, lubricant ON/OFF are output to electric control systems of machine tools from CNC, and then the electric control systems control output components including buttons, switches, indicators, relays, contactors and so on. Presently, the electric control systems are employed with Programmable Logic Controller (PLC) with characteristics of compact, convenience and high reliance. Thereof, the motion control systems and logic control systems are the main of CNC machine tools.

C1000T Turning Machine CNC system has simultaneously motion control and logic control function to control two axes of CNC machine tool to move, and has nested PLC function. Edit PLC programs (ladder diagram) according to requirements of input and output control of machine tool and then download them to C1000T Turning Machine CNC system, which realizes the required electric control requirements of machine tool, is convenient to electric design of machine tool and reduces cost of CNC machine tool.

Software used to control C1000T Turning Machine CNC system are divided into system software (NC for short) and PLC software (PLC for short). NC system is used to control the display, communication, edit, decoding, interpolation and acceleration/deceleration, and PLC system for controlling explanations, executions, inputs and outputs of ladder diagrams.

Standard PLC programs are loaded (except for the special order) when C1000T Turning Machine CNC System is delivered, concerned PLC control functions in following functions and operations are described according to control logics of standard PLC programs, marking with “Standard PLC functions” in C1000T Turning CNC System User Manual. Refer to Operation Manual of machine manufacturer about functions and operations of PLC control because the machine manufacturer may modify or edit PLC programs again.

Fig. 1-1

Programming is a course of workpiece contours, machining technologies, technology parameters and tool parameters being edit into part programs according to special CNC programming G codes. CNC machining is a course of CNC controlling a machine tool to complete machining of workpiece according requirements of part programs.

Technical flow of CNC machining is as following Fig. 1-2.

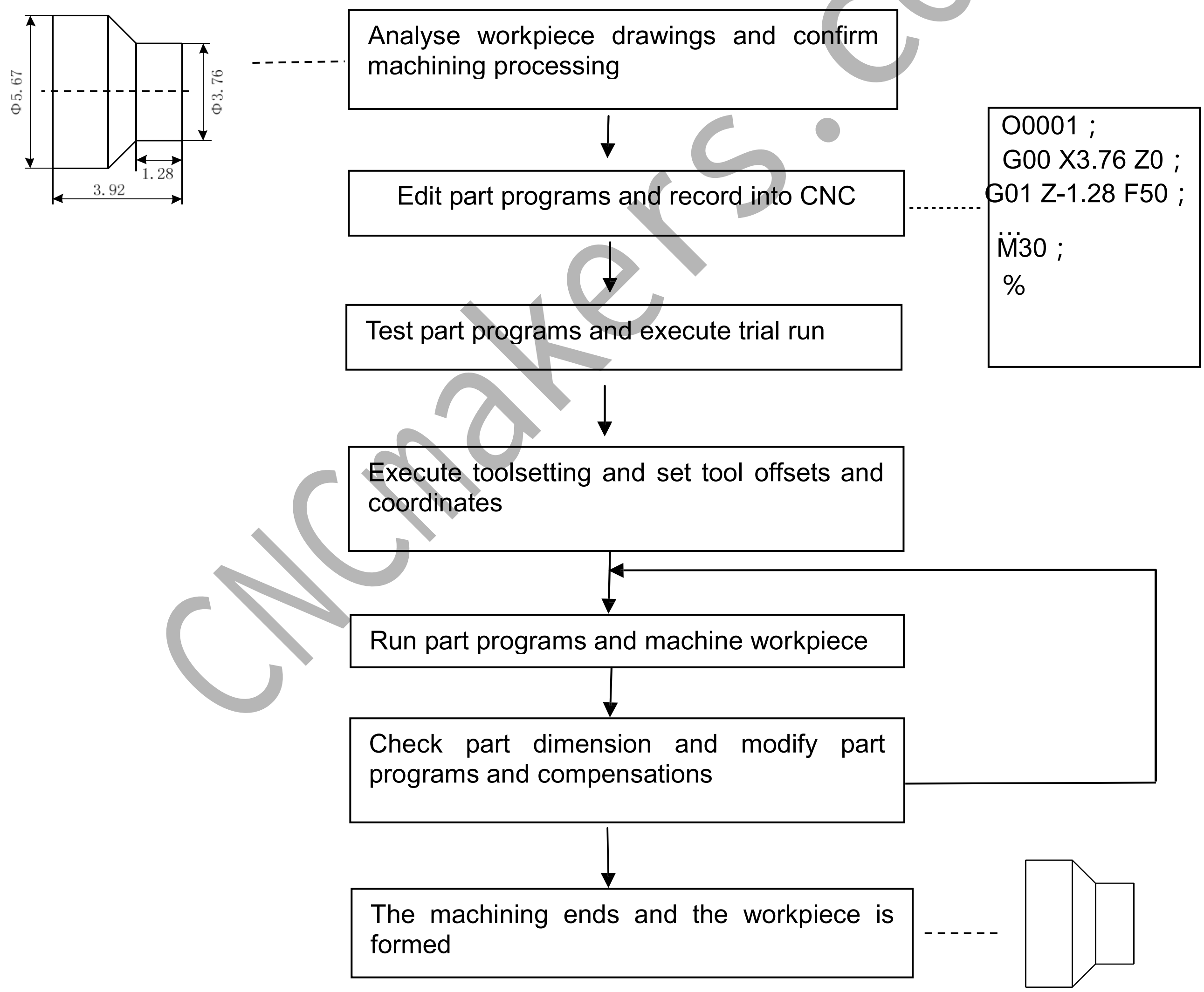


Fig. 1-2

1.3 Programming fundamentals

1.3.1 Coordinates definition

Sketch map of CNC turning machine is as follows:

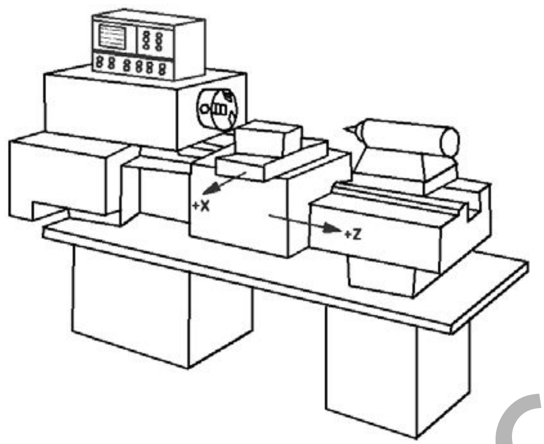


Fig. 1-3

C1000T uses a rectangular coordinate system composed of X, Z axis. X axis is perpendicular with axes of spindle and Z axis is parallel with axes of spindle; negative directions of them approach to the workpiece and positive ones are away from it.

There is a front tool post and a rear tool post of NC turning machine according to their relative position between the tool post and the spindle, Fig. 1-5 is a coordinate system of the front tool post and Fig. 1-6 is a rear tool post one. It shows exactly the opposite of X axes, but the same of Z axes from figures. In the manual, it will introduce programming application with the front tool post coordinate system in the following figures and examples.

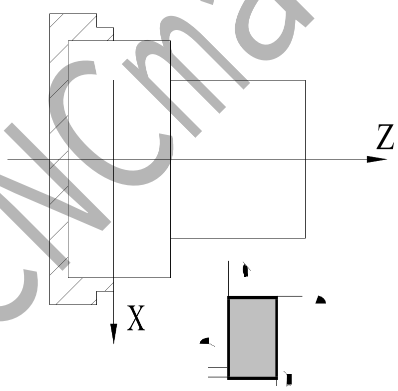


Fig.1-4 Front tool post coordinate system

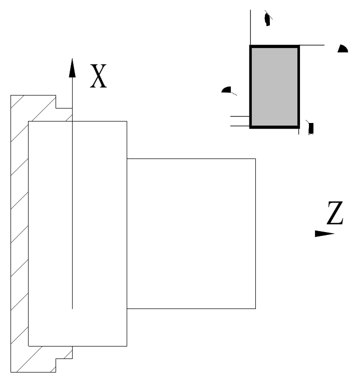


Fig.1-5 Rear tool post coordinate system

1.3.2 Machine coordinate system, Machine Zero and machine reference point

Machine tool coordinate system is a benchmark one used for CNC counting coordinates and a fixed one on the machine tool. **Machine tool zero** is a fixed point which position is specified by zero switch or zero return switch on the machine tool. Usually, the zero return switch is installed on max. stroke in X, Z positive direction. Machine reference point is located at the position at which the machine zero value adding the data parameter No.114/No.115 value. When No.114/No.115 value is 0, the machine reference point coincides with the machine zero. The coordinates of machine reference point is the No.120/No.121 value. Machine zero return/G28 zero return is to execute the machine reference point return. After the machine zero return/machine reference point return is completed, C1000T machine coordinate system which takes No.120/No.121 value as the reference point. **Note: Do not execute the machine reference point return without the reference point switch installed on the machine tool, otherwise, the motion exceeds the travel limit and the machine to be damaged.**

1.3.3 Workpiece coordinate system and Program Zero

The workpiece coordinate system is a rectangular coordinate system based on the part drawing, also called floating coordinate system. After the workpiece is installed on the machine, the absolute coordinates of tool's current position is set by G50 according to the workpiece's measure, and so the workpiece coordinate system is established in CNC. Generally, Z axis of the workpiece coordinate system coincides with the spindle axis. The established workpiece is valid till it is replaced by a new one.

The current position of workpiece coordinate system set by G50 is the program zero.

Note: Do not execute the machine reference point return without using G50 to set the workpiece coordinate system after power on, otherwise, the alarm occurs.

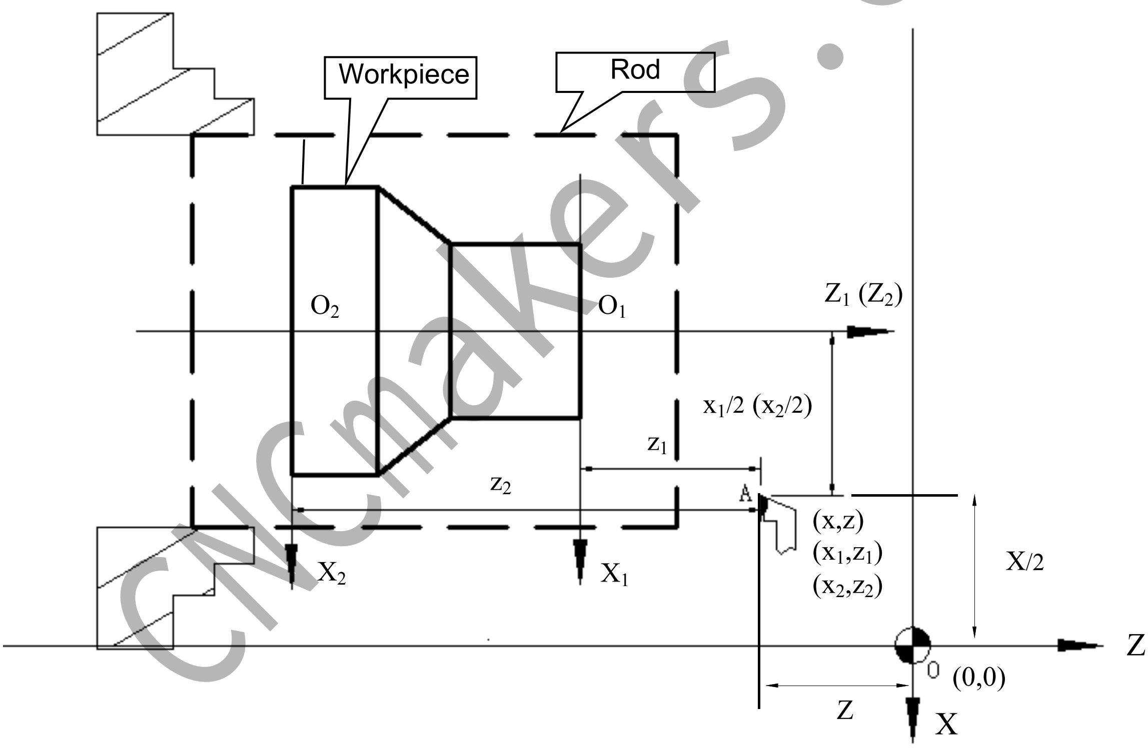


Fig. 1-6

In the above figure, XOZ is the coordinate system of machine tool, $X_1O_1Z_1$ is the workpiece coordinate system of X axis located at the heading of workpiece, $X_2O_2Z_2$ is the one of X axis located at the ending of workpiece, O point is the machine reference point, A point is the tool nose and coordinates of A point in the above-mentioned coordinate systems is as follows:

A point in the machine tool coordinate system: (x,z);

A point in $X_1O_1Z_1$ coordinate system: (x_1,z_1);

A point in $X_2O_2Z_2$ coordinate system: (x_2,z_2).

1.3.4 Interpolation function

Interpolation is defined as a planar or three dimensional contour formed by path of 2 or multiple axes moving at the same time, also called **Contour control**. The controlled moving axis is called link axis when the interpolation is executed. The moving distance, direction and speed of it are controlled synchronously in the course of running to form the required Composite motion path. Positioning control is defined that motion end point of one axis or multiple axes instead of the motion path in the course of running is controlled.

C1000T X and Z axis are link axes and 2 axes link CNC system. The system possesses linear, circular and thread interpolation function.

Linear interpolation: Composite motion path of X, Z axis is a straight line from starting point to end point.

Circular interpolation: Composite motion path of X, Z axis is arc radius defined by R or the circle center (I, K) from starting point to end point.

Thread interpolation: Moving distance of X or Z axis or X and Z axis is defined by rotation angle of spindle to form spiral cutting path on the workpiece surface to realize the thread cutting. For thread interpolation, the feed axis rotates along with the spindle, the long axis moves one pitch when the spindle rotates one rev, and the short axis and the long axis directly interpolate.

Example:

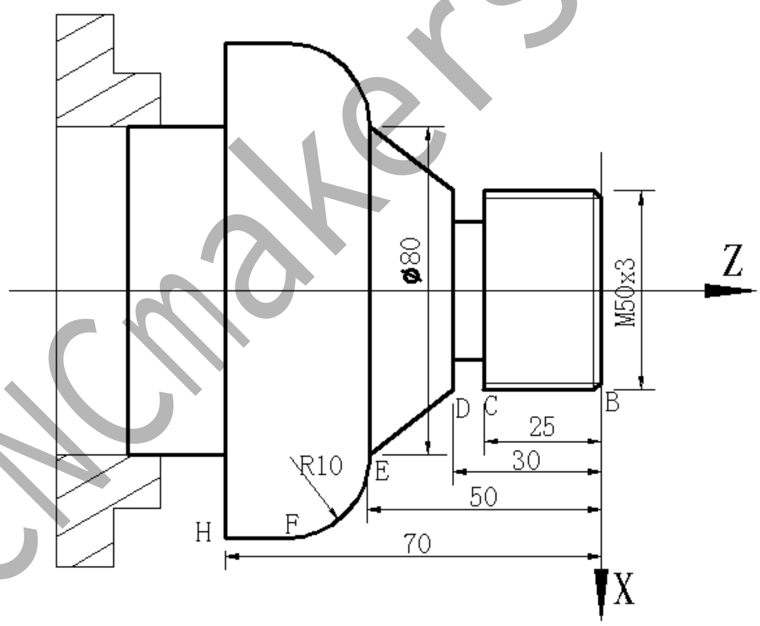


Fig. 1-7

```
…
G32 W-27 F3;         (B→C; thread interpolation)
G1 X50 Z-30 F100;
G1 X80 Z-50;         (D→E; linear interpolation)
G3 X100 W-10 R10;    (E→F; circular interpolation)
…
M30;
```


Volume I Programming

1.3.5 Absolute programming and incremental programming

Specify coordinate values of path's end point or target position in programming and there are 3 kinds of programming method according to coordinate values in programming: absolute programming, incremental programming and compound programming.

Programming with X/Z axis absolute coordinate value to program (present with X, Z) is defined to be the absolute programming;

Programming with X/Z axis incremental movement (present with U, W) is defined to be the incremental programming;

In the system, X, Z axis separately uses the absolute programming and incremental program, which is called the compound programming.

Example: A→B linear interpolation

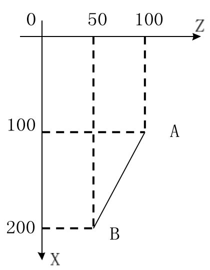


Fig.1-8

Absolute programming: G01 X200 Z50;
Incremental programming: G01 U100 W-50;
Compound programming: G01 X200 W-50; or G01 U100 Z50

Note: When there are command address X/ U or Z/ W at the same time, X/Z command value is valid.

Example: G50 X10 Z20;
G01 X20 W30 U20 Z30; 【End point of the block **(X20, Z30)**】

1.3.6 Diameter programming and radius programming

Programming methods of X coordinate values are divided into: diameter programming and radius programming.

Diameter programming: when NO.001 Bit2 is 0, X input command value is in diameter and X coordinate is in diameter at the moment;

Radius programming: when NO.001 Bit2 is 1, X input command value is in radius and X coordinate is in radius at the moment.

Table 1-3 Addresses related to diameter or radius programming

	Address	Explanation	Diameter programming	Radius programming
Addresses related to diameter or radius programming	X	X coordinate	In diameter	In radius
		G50 setting X coordinate		
	U	X increment	In diameter	In radius
		X finishing allowance in G71, G72, G73	In diameter	In radius
	R			
		Moving distance of tool retraction when cutting to the end point in G74	In diameter	In diameter

Except for addresses and data in Table 1-1, others (arc radius, taper in G90) are unrelated to diameter or radius programming, and their input values in X direction are defined by the radius.

Note: The diameter programming is used except for the special description in the following explanation.

1.4 Structure of an NC program

User needs to compile part programs (called program) according to command formats of CNC system. CNC system executes programs to control the machine tool movement, the spindle starting/stopping, the cooling and the lubricant ON/OFF to complete the machine of workpiece.

Program example:

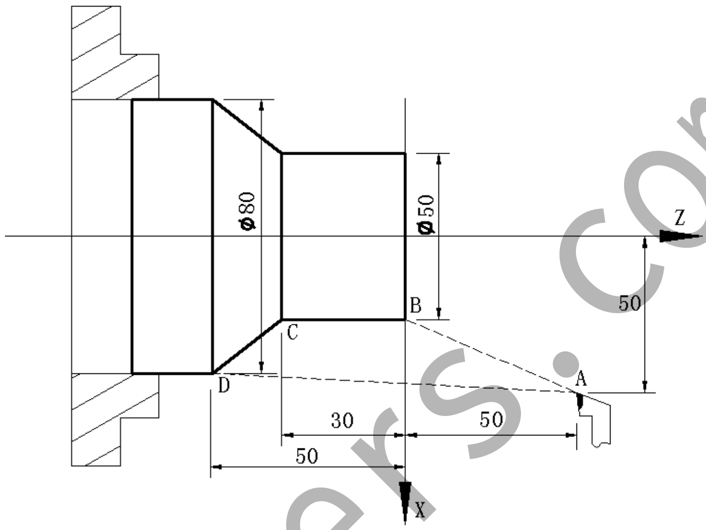


Fig. 1-9

O0001	;	(Program name)
N0005	G0 X100 Z50;	(Rapidly positioning to A point)
N0010	M12;	(Clamping workpiece)
N0015	T0101;	(Changing No.1 tool and executing its offset)
N0020	M3 S600;	(Starting the spindle with 600 r/min)
N0025	M8	(Cooling ON)
N0030	G1 X50 Z0 F600;	(Approaching B point with 600mm/min)
N0040	W-30 F200;	(Cutting from B point to C point)
N0050	X80 W-20 F150;	(Cutting from C point to D point)
N0060	G0 X100 Z50;	(Rapidly retracting to A point)
N0070	T0100;	(Canceling the tool offset)
N0080	M5 S0;	(Stopping the spindle)
N0090	M9;	(Cooling OFF)
N0100	M13;	(Releasing workpiece)
N0110	M30;	(End of program, spindle stopping and Cooling OFF)
N0120	%	

The tool leaves the path of A→B→C→D→A after the above-mentioned programs are executed.

1.4.1 General structure of a program

A program consists of a sequence of blocks, beginning with “OXXXX”(program name)and ending with “%”; a block begins with block number (omitted) and ends with “;” or “*”. See the general structure of program as follows:

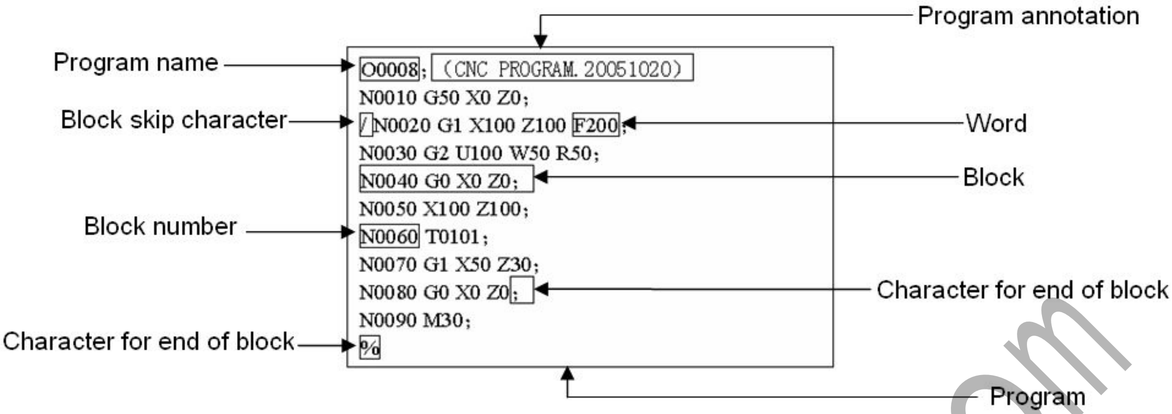
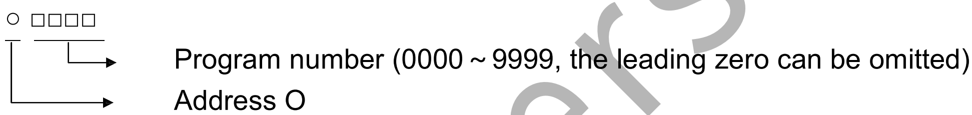


Fig. 1-10 Structure of a program

Program name

There are most 10000 programs stored in C1000T. To identify it, each program has only one program name (there is no the same program name) beginning with command address O and the following 4 digits.



Word

A word is the basic command unit to command CNC system to complete the control function, composed of an English letter (called command address) and the following number (operation command with/without sign). The command address describes the meaning of its following operation command and there may be different meaning in the same command address when the different words are combined together. All words of C1000T are in Table 1-4.

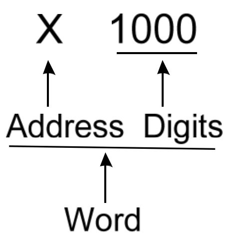


Table 1-4 Word table

Address	Command value range	Function meaning	Unit
O	0 ~ 9999	Program name	
N	0 ~ 9999	Block number	
G	00 ~ 99	Preparatory function	
X	-99999999 ~ 99999999	X coordinate	Related to IS-B, IS-C
	0 ~ 99999.999(s)	Pause time	
Z	-99999999 ~ 99999999	Z coordinate	Related to IS-B, IS-C
Y	-99999999 ~ 99999999	Y coordinate	Related to IS-B, IS-C

Address	Command value range	Function meaning	Unit
U	-99999999～99999999	X increment	Related to IS-B, IS-C
	0～99999.999(s)	Pause time	
	-99999～99999	X finishing allowance in G71,G72, G73	Related to IS-B, IS-C
	1～99999	Cutting depth in G71	Related to IS-B, IS-C
	-99999999～99999999	X tool retraction clearance in G73	Related to IS-B, IS-C
W	-99999999～99999999	Z increment	Related to IS-B, IS-C
	1～99999	Cutting depth in G72	Related to IS-B, IS-C
	-99999～99999	Z finishing allowance in G71,G72, G73	Related to IS-B, IS-C
	-99999999～99999999	Z tool retraction in G73	Related to IS-B, IS-C
V	-99999999～99999999	Y increment	Related to IS-B, IS-C
R	-99999999～99999999	Arc radius	Related to IS-B, IS-C
	1～99999	Tool retraction in G71, G72	Related to IS-B, IS-C
	1～9999 (times)	Roughing cycle times in G73	
	1～99999	Tool retraction clearance in G74, G75	Related to IS-B, IS-C
	1～99999	Tool retraction clearance from end point in G74, G75	Related to IS-B, IS-C
	1～99999999	Finishing allowance in G76	Related to IS-B, IS-C
	-99999999～99999999	Taper in G90, G92, G94, G96	Related to IS-B, IS-C
I	-99999999～99999999	X vector between arc center and starting point	Related to IS-B, IS-C
	0.06～25400（tooth/inch）	Metric thread teeth	
K	-99999999～99999999	Z vector between arc center and starting point	Related to IS-B, IS-C
F	0～8000（mm/min）	Feedrate per minute	
	0.0001～500(mm/r)	Feedrate per rev	
	0.001～500（mm）	Metric thread lead	
S	0～9999（r/min）	Spindle speed specified	
	00～04	Multi-gear spindle output	
T	01～32	Tool function	
M	00～99	Miscellaneous function output, program execution flow	
	9000～9999	Subprogram call	
P	0～9999999（0.001s）	Pause time	
	0～9999	Calling times of subprogram number	
	0～999	Subprogram call times	
	0～9999999	X circle movement in G74, G75	Related to IS-B, IS-C
		Thread cutting parameter in G76	

Address	Command value range	Function meaning	Unit
	0～9999	Initial block number of finishing in the compound cycle command	
Q	0～9999	End block number of finishing in the compound cycle	
	0～9999999	Z circle movement in G74, G75	Related to IS-B, IS-C
	1～9999999	First cut-in depth in G76	Related to IS-B, IS-C
	1～9999999	Min. cut-in depth in G76	Related to IS-B, IS-C
	0～360000	Offset angle between one-turn signal and starting point of thread cutting at the initial angle in G32	
H	01～99	Operand in G65	

Block

A block which is basic unit of CNC program consists of a sequence of words, ending with ";" or "*" . There is the character ";" or "*" between blocks. ";" is used to separate blocks in the manual as follows:

/ N0030 G0 X20 Z30 ;

End of block

Block number

Block skip

One block may be with a number of words or only with "; "ending character(EOB) instead of words. There must be one or more blank space between many words.

There is only one for other addresses except for N, G, S, T, H, L in one block, otherwise the system alarms. The last word in the same address is valid when there are more N, G, S, T, H, L in the same block. The last G code is valid when there are more G codes which are in the same group in one block.

Block number

A block number consists of an address N and its following 6-digit: N000000 ~ N999999, and the leading zero can be omitted. The block number must be at the beginning of block, otherwise the block is invalid.

The block number can be omitted, but there must be the block number when the program calls/skips the target block. The increment of block number is at will and it better to increase or decrease the sequence of block number in order to conveniently search and analyze programs.

When *"Automatic number"* in the switch window is set to "ON", block numbers will be automatically created incrementally and their increment is defined by N_o.42.

Character for block skip

Insert "/" in the front of block and startup 

when some block cannot be executed (cannot be deleted), and the system skips the block and executes the next one. The block with "/" in the front of it is executed if the block skip switch is not started.

Character for end of a program

"%" is an ending character of program. "%" is a mark of communication ended when the program is transmitted. The system will automatically insert "%" at the end of program.

1.4.2 Main program and subprogram

To simply the programming, when the same or similar machining path and control procedure is used many times, its program commands are edited to a sole program to call. A program which calls the program is the main program and the called program (end with M99) is subprogram. They both take up the program capacity and storage space of system. The subprogram has own name, and can be called at will by the main program and also can run separately. The system returns to the main program to continue when the subprogram ends as follows.

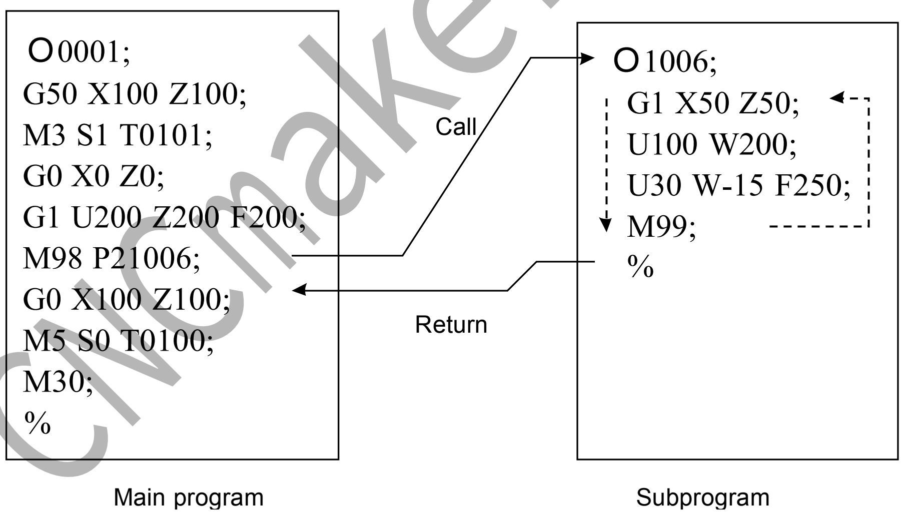


Fig. 1-12 Main program and subprogram

1.5 Program run

1.5.1 Sequence of program run

Running the current open program must be in Auto mode. C1000T cannot open two or more programs at the same, and runs only program any time. When the first block is open, the cursor is located in the heading of the first block and can be moved in Edit mode. In the run stop state in Auto mode

The program starts to run by the cycle start signal (CYCLE START is pressed or external cycle start signal)from a block pointed by current cursor, usually blocks are executed one by one according to their programming sequence, the program stops running till executing M02 or M30. The cursor moves along with program running and is located at the heading of the current block. Sequence and state of program running are changed in the followings:

* The program stops running after pressing RESET or emergent stop button;
* The program stops running when the system or PLC alarms;
* The program runs and single block stops (the program run stops after the current block runs completely) in Edit, MDI mode, and then a block pointed by the current cursor starts running after the system switches into Auto mode, CYCLE START is pressed or external cycle start signal is switched on;
* The program stops running in Manual(Jog), Handwheel (MPG), Single Block, Program Reference Point Return, Machine Reference Point Return mode and it continuously runs from current position after the system is switched into Auto mode and CYCLE START is pressed or the external cycle start signal is switched on;
* The program pauses after pressing FEED HOLD or the external cycle start signal is switched off, and it continuously runs from current position after pressing CYCLE START or the external cycle start signal is switched on;
* When Single Block is ON, the program pauses after every block is executed completely, and then it continuously runs from the next block after CYCLE START is pressed or the external cycle start signal is switched on;
* Block with "/" in the front of it is not executed when the block skipping switch is ON;
* The system skips to the target block to run after executing G65;
* Please see Section Three G Commands about execution sequence of G70~73;
* Call corresponding subprograms or macro program to run when executing M98 ; The system returns to main program to call the next block when executing M99(if M99 specifies a target block number, the system returns to it to run) after the subprograms or macro programs run completely;
* The system return to the first block to run and the current program is executed repetitively when M99 is executed in a main program.

1.5.2 Execution sequence of word

There are many words (G, X, Z, F, R, M, S, T and so on) and most of M, S, T is transmitted to PLC by NC explaining and others are directly executed by NC. M98, M99, S word used to specify the spindle speed r/min, m/min is directly executed by NC.

NC firstly executes G and then M commands when G codes and M00, M01, M02 and M30 are in the same block.

NC firstly executes G and then M commands(without transmitting M signal to PLC) when G codes and M98, M99 are in the same block.

When G codes and M, S, T executed by PLC are in the same block, PLC defines M, S, T and G to be executed simultaneously, or execute M, S ,T after G codes. Please see User Manual of machine manufacturer for execution sequence of commands.

Execution sequence of G, M, S, T in the same block defined by C1000T standard PLC program is as follows:

M3, M4, M8, M10, M12, M32, M41, M42, M43, M44, S□□, T□□□□ and G codes are executed simultaneously;

M5, M9, M11, M13, M33 after G codes are executed;

M00, M01, M02, M30 after other commands of current block are executed.

CHAPTER 2 MSTF COMMAND

2.1 M (miscellaneous function)

M command consists of command address M and its following 1～2 or 4 bit digits, used for controlling the flow of executed program or outputting M commands to PLC.

M □□□□

Command value(00~99, 9000~9999, the leading zero can be omitted)

Command address

M98, M99 is executed by NC separately and NC does not output M commands to PLC.

M02, M03 are for ending of programs defined by NC, and NC outputs M commands to PLC which can control spindle OFF, cooling OFF and so on.

M98, M99 are for calling programs, M02, M30 are for ending of program which are not changed by PLC. Other M commands output to PLC and their function are defined by PLC.
Please refer to User Manual from machine manufacturer.

There is only one M command in one block, otherwise the system alarms.

Table 2-1 M commands to control program execution

Commands	Functions
M02	End of program
M30	End of program
M98	Call subprograms
M99	Return from a subprogram; it is executed repeatedly when the program ends in M99(the current program is not called by other programs)

2.1.1 End of program M02

Command format: M02 or M2

Command function: In Auto mode, after other commands of current block are executed, the automatic run stops, the amount of workpiece is added 1,the tool nose radius compensation is cancelled and the cursor return to the start of program (whether return to the start of program or not is defined by parameters).

2.1.2 End of program run M30

Command format: M30

Command function: In Auto mode, after other commands of current block are executed in M30, the automatic run stops, the amount of workpiece is added 1, the tool nose radius compensation is cancelled and the cursor returns to the start of program (whether the cursor return to the start of program or not is defined by parameters).

If No.005 Bit 4 is set to 0, the cursor does not return to the beginning of program, and the cursor returns immediately after the program is executed completely when No.005 Bit 4 is set to 1.

2.1.3 Subprogram call M98

Command format:

M98 P○○○○□□□□

Called subprogram number (0000 ~ 9999). The leading zero of subprogram number can be omitted when the calling times is not input; the subprogram number must be with 4 digits when the calling times is input..

Call times: 1 ~ 9999. The calling times cannot be input when it is 1.

Command function: In Auto mode, after other commands are executed in M98, CNC calls subprograms specified by P, and subprograms are executed 9999 times at most. M98 is invalid in MDI mode.

2.1.4 Return from subprogram M99

Command format: M99 P○○○○

Executed block after returning to the main program is 0000 ~ 9999, and its leading zero can be omitted.

Command function: After other commands of current block in the subprogram are executed, the system returns to the main program and continues to execute next block specified by P, and calls a block following M98 of current subprogram when P is not input. The current program is executed repeatedly when M99 is defined to end of program (namely, the current program is executed without calling other programs). M99 is invalid in MDI mode.

Example: Execution path of calling subprogram (with P in M99) as Fig. 2-1.

Execution path of calling subprogram (without P in M99) as Fig. 2-2.

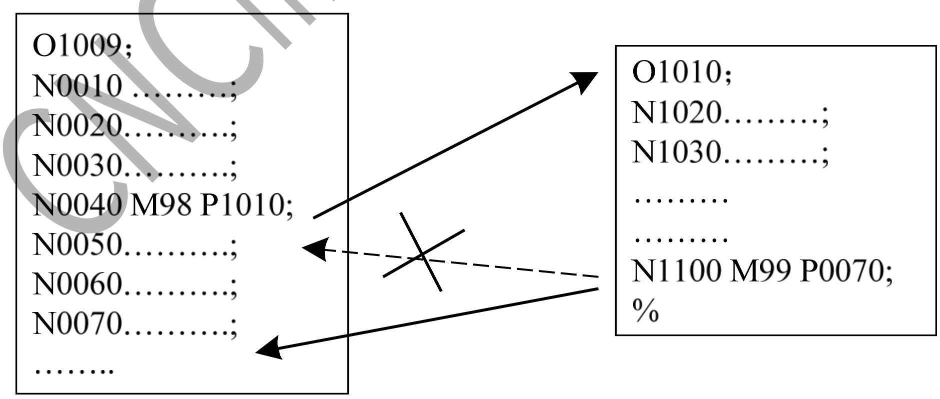


Fig. 2-1 Execution path of calling subprogram (with P in M99)

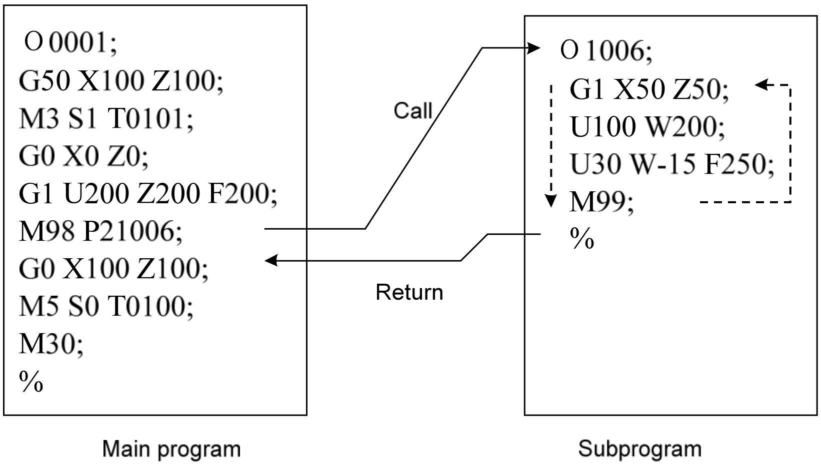


Fig. 2-2 Execution path of calling subprogram (without P in M99)

Subprogram calls can be nested up to four levels as shown in Fig. 2-3.

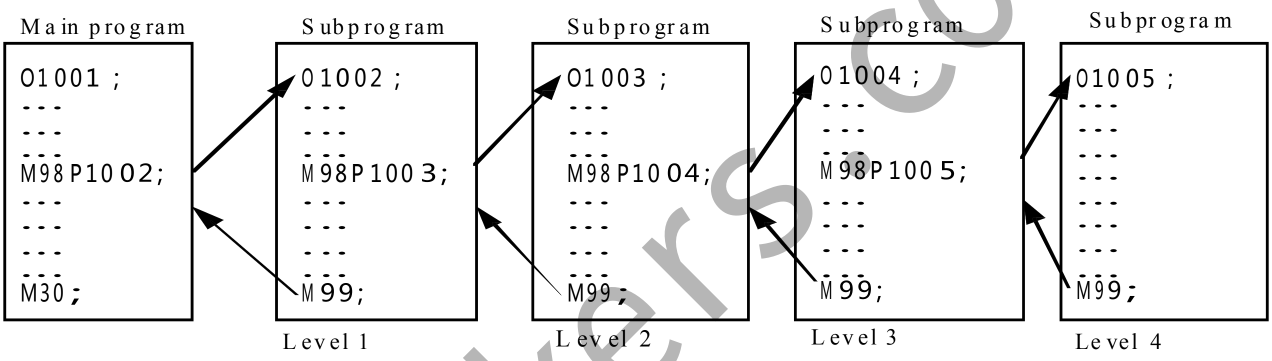


Fig. 2-3 Subprogram nesting

2.1.5 M commands defined by standard PLC ladder diagram

Other M commands are defined by PLC except for the above-mentioned ones(M02, M30, M98, M99, M9000～M9999). The following M commands are defined by standard PLC, and C1000T Turning Machine CNC system is used for controlling machine tool. Refer to commands of machine manufacturer about functions, significations, control time sequence and logic of M commands.

M commands defined by standard PLC ladder diagram.

Table 2-2 M commands

Command	Function	Remark
M00	Program pause	
M01	Program optional stop	
M03	Spindle clockwise (CW)	Functions interlocked and states reserved
M04	Spindle counterclockwise (CCW)	
*M05	Spindle stop	
M08	Cooling ON	Functions interlocked and states reserved
*M09	Cooling OFF	
M10	Tailstock forward	Functions interlocked and states reserved
M11	Tailstock backward	
M12	Chuck clamping	Functions interlocked and states reserved
M13	Chuck releasing	
M14	Spindle position control	Functions interlocked and states reserved
*M15	Spindle speed control	
M20	Spindle clamping	Functions interlocked and states reserved
*M21	Spindle releasing	
M24	The 2nd spindle position control	Functions interlocked and states reserved
*M25	The 2nd spindle speed control	
M32	Lubricating ON	Functions interlocked and states reserved
*M33	Lubricating OFF	
M63	The 2nd spindle rotation CCW	Functions interlocked and states reserved
M64	The 2nd spindle rotation CW	
*M65	The 2nd spindle stop	
*M41, M42, M43, M44	Spindle automatic gear shifting	Functions interlocked and states reserved

Note: Commands with "*" defined by standard PLC is valid when power on.

2.1.6 Program stop M00

Command format: M00 or M0

Command function: After M00 is executed, the program stops and the system displays "*Pause*", and then the program continuously runs after the cycle start key is pressed.

2.1.7 Program optional stop M01

Command format: M01 or M1

Command function: in AUTO, MDI mode, it is valid. Press  and its indicator lights and the system enters the optional stop state, at the moment, the program stops run and the system displays “PAUSE” after M01 is executed, after the cycle start key is pressed, the program continuously runs. When the program optional stop switch is not open, the program does not pause even if M01 runs.

2.1.8 Spindle CW, CCW and stop control M03, M04, M05

Command format: M03 or M3
M04 or M4;
M05 or M5.

Command function: M03: Spindle CW rotation;
M04: Spindle CCW rotation;
M05: Spindle stop.

Note: Refer to time sequence of output defined by standard PLC ladder in VOLUME Ⅲ INSTALLATION & CONNECTION.

2.1.9 Cooling control M08, M09

Command format: M08 or M8;
M09 or M9;

Command function: M08: Cooling ON;
M09: Cooling OFF.

Note: Refer to time sequence and logic of M08, M09 defined by standard PLC ladder in VOLUME Ⅲ INSTALLATION & CONNECTION.

2.1.10 Tailstock control M10, M11

Command format: M10;
M11;

Command function: M10: tailstock going forward;
M11: tailstock going backward.

Note: Refer to time sequence and logic of M10, M11 defined by standard PLC ladder in VOLUME Ⅲ INSTALLATION & CONNECTION

2.1.11 Chuck control M12, M13

Command format: M12;
M13;

Command function: M12: chuck clamping;
M13: chuck releasing.

Note: Refer to time sequence and logic of M12, M13 defined by standard PLC ladder in VOLUME Ⅲ INSTALLATION & CONNECTION.

2.1.12 Spindle position/speed control switch M14, M15

Command format：M14；
M15；

Command function：M14：spindle is in the position control mode from speed control mode;

M15：spindle is in speed control mode from the position control mode. **Note: Refer to time sequence and logic of M14, M15 defined by standard PLC ladder in VOLUME Ⅲ INSTALLATION & CONNECTION.**

2.1.13 Spindle clamped/released M20, M21

Command format： M20；
M21；

Command function：M20：spindle clamped
M21：spindle released

Note: Refer to time sequence and logic of M20, M21 defined by standard PLC ladder in VOLUME Ⅲ INSTALLATION & CONNECTION.

2.1.14 The 2nd spindle position/speed switch M24, M25

Command format：M24；
M25；

Command function：M24：The 2nd spindle is switched from the speed control mode to the position control mode;
M25：The 2nd spindle is switched from the position control mode to the speed control mode.

Note: Refer to time sequence and logic of M24, M25 defined by standard PLC ladder in VOLUME Ⅲ INSTALLATION&CONNECTION.

2.1.15 Lubricating control M32, M33

Command format： M32；
M33；

Command function：M32：lubricating ON;
M33：lubricating OFF.

Note: Refer to time sequence and logic of M32, M33 defined by standard PLC ladder in VOLUME Ⅲ INSTALLATION&CONNECTION.

2.1.16 Spindle automatic gear change M41, M42, M43, M44

Command format：M4n；(n=1, 2, 3, 4)

Command function：When the system executes M4n, the spindle changes to gear n.

Note: Refer to time sequence and logic of M41, M42, M43, M44 defined by standard PLC ladder in VOLUME Ⅲ INSTALLATION&CONNECTION.

2.1.17 The 2nd spindle rotation CCW, rotation CW , stop M63, M64, M65

Command format: M63 ;

M64 ;

M65 ;

Command function:

M63: spindle rotation CCW;

M64: spindle rotation CW;

M65: spindle stop.

Note 1: The sequence of M63, M64, M65 defined by the standard PLC is the same that of M03, M04, M05.

Note 2: The function is enabled when the 2nd spindle function is valid.

2.2 Spindle function

S command is used for controlling spindle speed and this C1000T has two modes to control it:

Spindle speed switching value control: S□□(2 digits command value)is executed by PLC, and PLC outputs switching value signal to machine tool to change spindle speed with grades.

Spindle speed analog voltage control: S□□□□(4 digits command value)specifies actual speed of spindle and NC outputs 0~10V analog voltage signal to spindle servo or converter to realize stepless spindle speed.

2.2.1 Spindle speed switching value control

Spindle speed is controlled by switching value when No.001 BIT4 is set to 0. There is only one S command in a block, otherwise the system alarms.

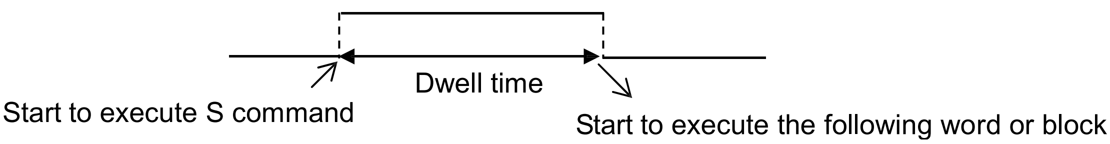
Their executing sequence is defined by PLC when S command and word for moving function are in the same block. Please refer to *User Manual* from machine manufacturer.

When spindle speed is controlled by switching value, C1000T Turning CNC system is used for machine tool and the time sequence and logic of executing S command is according to *User Manual* from machine manufacturer. Refer to S command defined by standard PLC of C1000T as follows:

Command format: S□□

00～04(the leading zero can be omitted): No.1～No.4 gear of spindle speed is controlled by switching value.

In spindle speed switching value control mode, after S signal transmits to PLC, the system dwells time defined by No.081, then return FIN signal, and the dwell time is called runtime of S command.



S01, S02, S03, S04 output are reserved when resetting CNC.

S1～S4 output are invalid when CNC is switched on. The corresponding S signal output is valid and reserved, and others are cancelled at the same time when executing one of S01, S02, S03, S04. When executing S00, S1～S4 output are cancelled and only one of S1～S4 is valid at the same time.

2.2.2 Spindle speed analog voltage control

Spindle speed is controlled by analog voltage when No.001 BIT4 is set to 1.

Command format: S OOOO

0000 ~ 9999 (the leading zero can be omitted.):Spindle speed analog voltage control

Command function: The spindle speed is defined, and the system outputs 0 ~ 10V analog voltage to control spindle servo or converter to realize the stepless timing. S command value is not reserved, and it is 0 after the system is switched on.

When the spindle speed analog voltage control is valid, there are 2 methods to input the spindle speed: the spindle fixed speed is defined by S command(r/min), and is invariant without changing S command value, which is called constant speed control(G97 modal); other is the tangent speed of tool relative to the outer circle of workpiece defined by S command, which is called constant surface speed control (G96 modal), and the spindle speed is changed along with the absolute coordinates value of X absolute coordinates in programming path when cutting feed is executed in the constant surface speed.

Please refer to Section 2.2.3**.**

The system can execute 4 gears spindle speed. Count the analog voltage value corresponding to the specified speed according to setting value(corresponding to No.037 ~ No.040) of max. spindle speed (analog voltage is 10V)of current gear, and then output to spindle servo or converter to ensure that the spindle actual speed and the requirement are the same.

After the system is switched on, the analog output voltage is 0V. The analog output voltage is reserved (except that the system is in cutting feed in the surface speed control mode and the absolute value of X absolute coordinates is changed) after S command is executed. The analog output voltage is 0V after S0 is executed. The analog output voltage is reserved when the system resets and emergently stops.

Parameters relative to the analog voltage control of spindle speed:

System parameter No.021: offset value of output voltage with max. spindle speed (the analog output voltage is 10V);

System parameter No.036: offset value of output voltage with spindle speed 0 (the analog output voltage is 10V);

System parameter No.037 ~ No.040: max. spindle speed (the analog output voltage is 10V) with spindle 1 ~ 4 gears(corresponding to M41 ~ M44).

2.2.3 Constant surface speed control G96, constant rotational speed control G97

Command format: G96 S__; (S0000 ~ S9999, the leading zero can be omitted.)

Command function: The constant surface speed control is valid, the cutting surface speed is defined (m/min) and the constant rotational speed control is cancelled. G96 is modal G code. If the current modal is G96, G96 cannot be input.

Command format: G97 S__; (S0000 ~ S9999, the leading zero can be omitted.)

Command function: The constant surface speed control is cancelled, the constant rotational speed control is valid and the spindle speed is defined (r/min). G96 is modal G code. If the current modal is G97, G97 cannot be input.

Command format: G50 S__; (S0000 ~ S9999, the leading zero can be omitted.)

Command function: define max. spindle speed limit (r/min) in the constant surface speed control and take the current position as the program reference point.

G96, G97 are the modal word in the same group but one of them is valid. G97 is the initial word and the system defaults G97 is valid when the system is switched on.

When the machine tool is turning it, the workpiece rotates based on the axes of spindle as the center line, the cutting point of tool cutting workpiece is a circle motion around the axes, and the instantaneous speed in the circle tangent direction is called **cutting surface** (for short **surface speed**). There are different surface speed for the different workpiece and tool with different material.

When the spindle speed controlled by the analog voltage is valid, the constant surface control is valid. The spindle speed is changed along with the absolute value of X absolute coordinates of programming path in the constant speed control. If the absolute value of X absolute coordinates adds, the spindle speed reduces, and vice verse, which make the cutting surface speed as S command value. The constant speed control to cut the workpiece makes sure all smooth finish on the surface of workpiece with diameter changing.

Surface speed=spindle speed × |X| × π ÷1000 (m/min)

Spindle speed: r/min

|X|: absolute value of X absolute coordinate value, mm

π ≈3.14

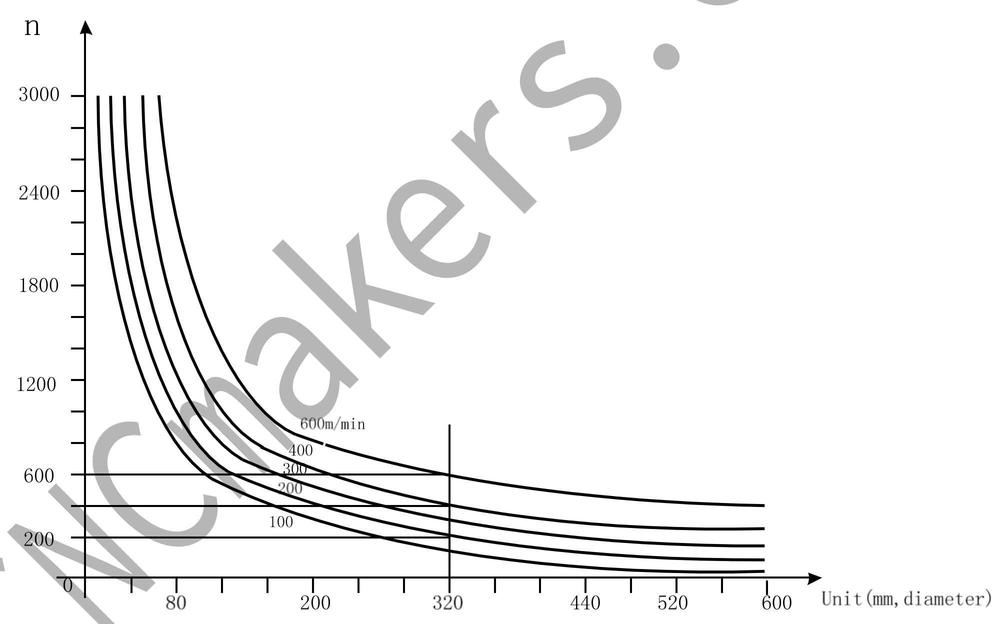


Fig. 2-4

In G96, the spindle speed is changed along with the absolute value of X absolute coordinates value of programming path in cutting feed (interpolation), but it is not changed in G00 because there is no actual cutting and is counted based on the surface speed of end point in the program block.

In G96 (constant surface speed control), Z coordinates axis of workpiece system must consist with the axes of spindle (rotary axis of workpiece), otherwise, there is different between the actual surface speed and the defined one.

G96 control is valid, G50 S_ can limit max. spindle speed (r/min). The spindle actual speed is the limit value of max. speed when the spindle speed counted by the surface speed and X coordinates value is more than the max. spindle speed set by G50 S_. After the system powers on, max. spindle speed limit value is not defined and its function is invalid. Max. spindle speed limit value defined by G50 S_ is reserved before it is defined again and its function is valid in G96. Max. spindle speed defined by G50 S_ is invalid in G97 but its limit value is reserved.

Note: When NO.043 (lowest spindle speed in constant surface speed control) is set to 0 and G50 S0 is executed, the spindle speed is limited to 0 r/min (the spindle does not rotate).

When the constant surface speed is controlled by the system parameter No.043, the spindle speed is lower limit, which is higher than one counted by the surface speed and X axis coordinates value.

Example:

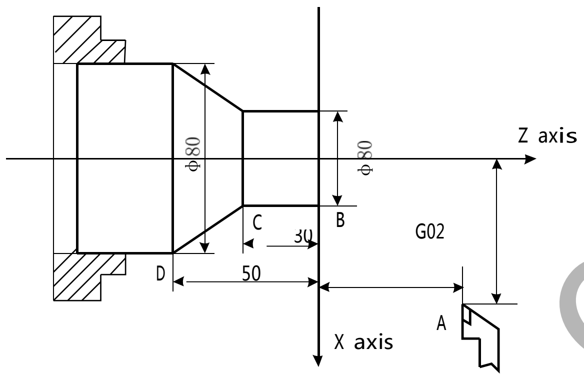


Fig. 2-5

O0001	;	(Program name)
N0010	M3 G96 S300;	(Spindle rotates clockwise, the constant surface speed control is valid and the surface speed is 300 m/min)
N0020	G0 X100 Z100;	(Rapid traverse to A point with spindle speed 955 r/min)
N0030	G0 X50 Z0;	(Rapid traverse to B point with spindle speed 1910 r/min)
N0040	G1 W-30 F200;	(Cut from B to C with spindle speed 1910 r/min)
N0050	X80 W-20 F150;	(Cut from C to D with spindle speed 1910 r/min and surface speed 1194 r/min)
N0060	G0 X100 Z100;	(Rapid retract to A point with spindle speed 955 r/min)
N0110	M30;	(End of program, spindle stopping and cooling OFF)
N0120	%	

Note 1: S value commanded in G96 is also reserved in G97. Its value is resumed when the system is in G96 again; Example:

G96 S50; (Cutting surface speed 50m/min)
G97 S1000; (Spindle speed 1000 r/min)
G96 X3000; (Cutting surface speed 50m/min)

Note 2: The constant surface speed control is valid when the machine tool is locked (X, Z do not move when their motion command are executed);

Note 3: To gain the precise thread machining, it should not be adopted with the constant surface speed control but the constant rotational speed (G97) in the course of thread cutting;

Note 4: From G96 to G97, if none of S command (r/min) is commanded in the program block in G97, the last spindle speed in G96 is taken as S command in G97, namely, the spindle speed is not changed at this time;

Note 5: In G96, when the spindle speed counted by the cutting surface speed is more than max. speed of current spindle gear (system parameter No.037 ~ No.040), at this time, the spindle speed is limited to max. one of current spindle gear.

2.2.4 Spindle override

When the spindle speed analog voltage control is valid, the spindle actual speed can be tuned real time by the spindle override and is limited by max spindle speed of current gear after the spindle override is tuned, and it also limited by limited values of max. and min. spindle speed in constant surface speed control mode.

The system supplies 8 steps for spindle override (50%～120% increment of 10%). The actual steps and tune of spindle override are defined by PLC ladder and introductions from machine manufacturer should be referred when using it. Refer to the following functions of CNCmaker Limited standard PLC ladder.

The spindle actual speed specified by C1000T standard PLC ladder can be tuned real time by the spindle override tune key at 8 steps in 50%～120% and it is not reserved when the spindle override is switched off. Refer to the operations of spindle override in **VOLUME Ⅱ OPERATION**.

2.3 Tool function

2.3.1 Tool control

T functions of C1000T: automatic tool change and executing tool offset. Control logic of automatic tool change is executed by PLC and tool offset is executed by NC.

Command format:

T □□ ○○

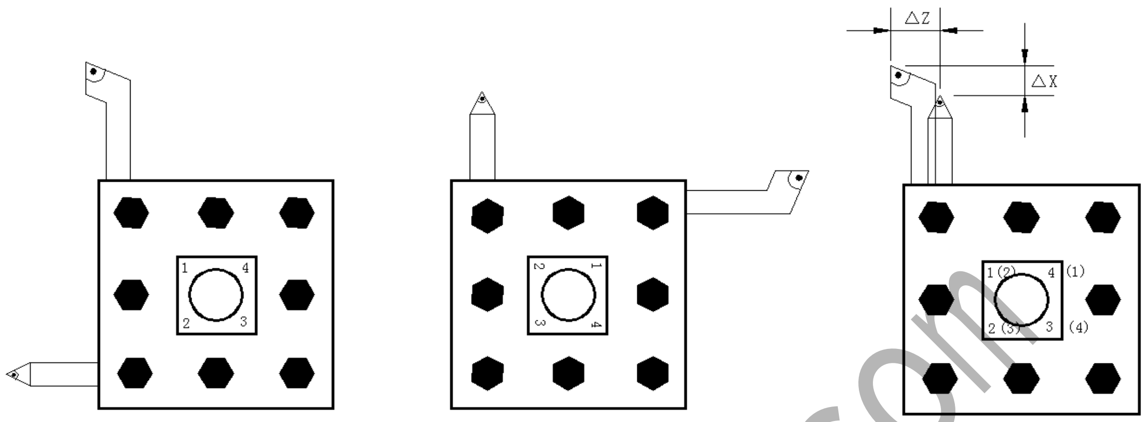
Tool offset number（00-32，the leading zero cannot be omitted）

Target tool number（01-32，the leading zero cannot be omitted）

Command function: The automatic tool post rotates to the target tool number and the tool offset of tool offset number commanded is executed. The tool offset number can be the same as the tool number, and also cannot be the same as it, namely, one tool can corresponds to many tool offset numbers. After executing tool offset and then T□□00, the system reversely offset the current tool offset and the system its operation mode from the executed tool length compensation into the non-compensation, which course is called the canceling tool offset, called canceling tool compensation. When the system is switched on, the tool offset number and the tool offset number displayed by T command is the state before the system is switched off.

Only one T command is in a block, otherwise the system alarms.

Toolsetting is executed to gain the position offset data before machining (called tool offset), and the system automatically executes the tool offset after executing T command when programs are running. Only edit programs for each tool according to part drawing instead of relative position of each tool in the machine coordinate system. If there is error caused by the wearing of tool, directly modify the tool offset according to the dimension offset.



The tool offset is used for the programming. The offset corresponding to the tool offset number in T command is added or subtracted on the end point of each block. Tool offset in X direction in diameter or radius is set by No.004 Bit4. For tool offset in diameter or radius in X direction, the external diameter is changed along with diameter or radius when the tool length compensation is changed.

Example: When the state parameter No.004 Bit4 is set to 0 and X tool length compensation value is 10mm, the diameter of workpiece external diameter is 20mm as Fig.2-5 :

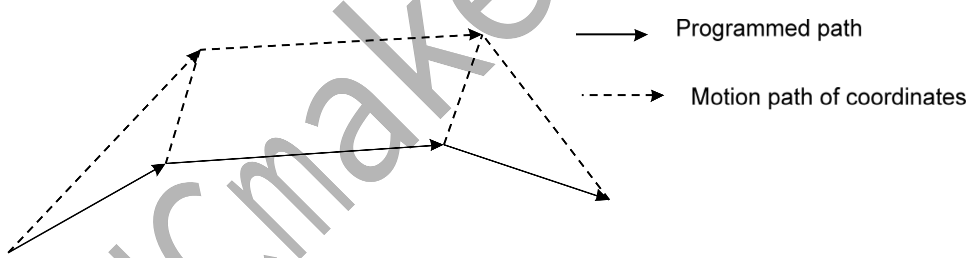


Fig. 2-5 Creation, execution and cancellation of tool length compensation

G01 X100 Z100 T0101; (Block 1, start to execute the tool offset)
G01 W150; (Block 2, tool offset Block 2, tool offset)
G01 U150 W100 T0100; (Block 3, canceling tool offset)

There are two methods defined by No.003 Bit4 to execute the tool length compensation:

Bit4=0: The tool length compensation is executed by the tool traversing;

Bit4=1: The tool length compensation is executed by modifying the coordinates;

Example:

Table 2-4

Tool offset number	X	Z
00	0.000	0.000
01	0.000	0.000
02	12.000	-23.000
03	24.560	13.452

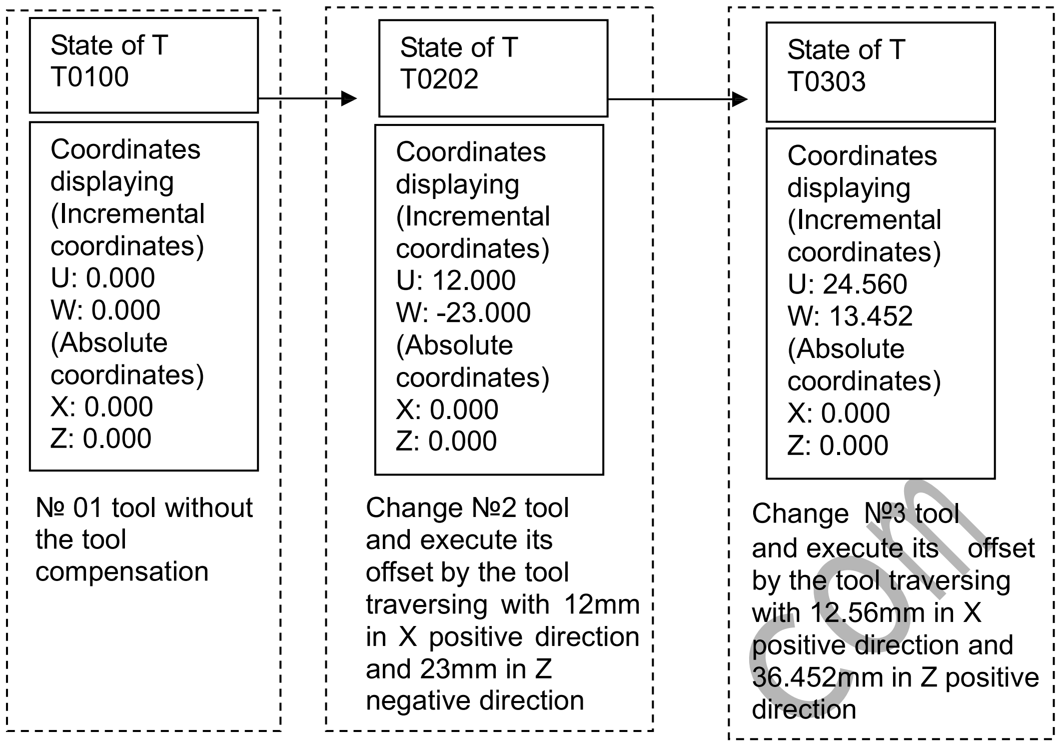


Fig. 2-6 Tool traversing mode

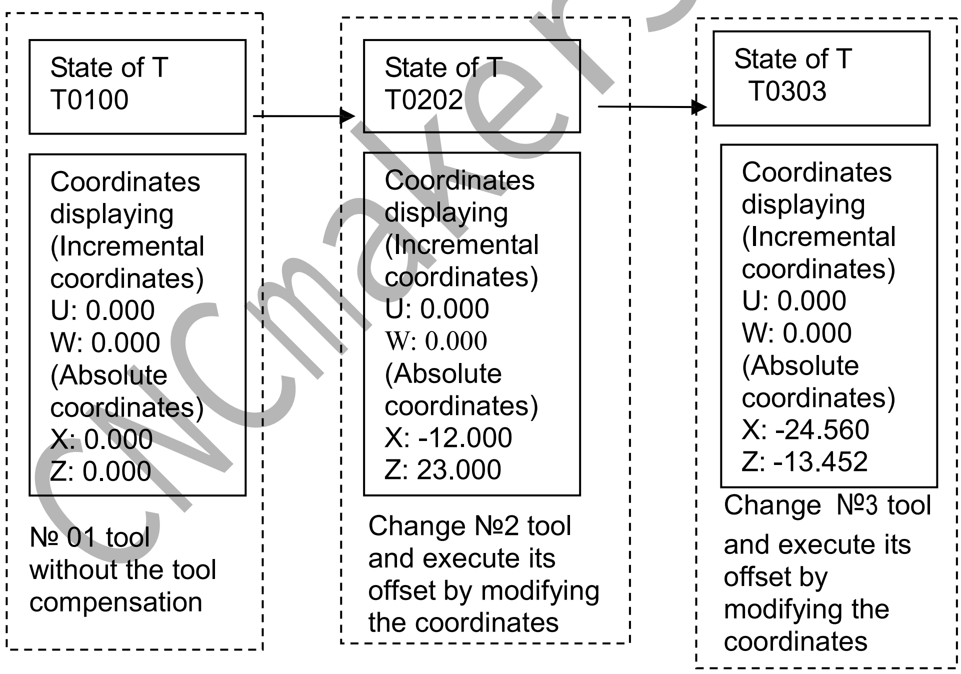


Fig. 2-7 Modifying the coordinates mode

In Edit and Auto mode, a sole T word in executing tool offset (it is not with the motion command in the same block) is relative to No.004 BIT3 setting (as Fig.2-6 and Fig.2-7).

When T command and the motion command are in the same block and execute tool offset by modifying coordinates, the motion command and T command are executed at the same time, the system executes by adding the current tool offset to coordinates of motion command and whether the traverse speed is employed the cutting feedrate or the rapid traverse speed defined by the motion command.

When T command and the motion command are in the same block and execute tool offset by traversing tool, the motion command or T command is executed separately. Firstly tool change is executed and then the motion command is executed. The tool offset is executed at current rapid traverse speed.

The tool offset is cancelled after one of the following operations is executed:

1. Execute T□□00 command;
2. Execute G28 or manual machine reference point return (only the tool offset of coordinate axis which is executed machine reference point return is cancelled, and another one which is not executed machine reference point return is not cancelled);

When No.084 is not 1 (2～32) and target tool number is not equal to current display tool number, the control sequence and logic of tool post is defined by PLC ladder diagram after commanding T command, please see User Manual of machine tool manufacturer. C1000T standard PLC ladder diagram defines as follows: clockwise rotation for selecting tool, counterclockwise rotation for tool post clamping, directly inputting tool selection signal for tool change. Please refer to **VOLUME Ⅲ INSTALLATION&CONNECTION.**

When the system is employed with line-up tool post, No.084 should be set to 1 and different tool number is executed by different tool offset as T0101, T0102, T0103.

standby tools. Executing the counting in MDI mode is determined by No.002 Bit3 (MDITL).

CHAPTER 3 G COMMANDS

3.1 Commands

G command consists of command address G and its following 1 ~ 2 bits command value, used for defining the motion mode of tool relative to the workpiece, defining the coordinates and so on. Refer to G commands as Fig. 3-1.

G □□

→ Command value (00 ~ 99 , the leading zero can be omitted)

→ Command address G

G words are divided into 9 groups (00, 01, 02, 03, 05 、 06 、 07 、 16 、 21). Except that commands in the group 01 and 00 are not in the same block, G words in the different groups can be input to the same block and the last one is valid when two or more G words in the same group are input. The words in the different groups without the same parameter (word) can be in the same block and their functions are valid without sequence at the same time. The system alarms when G words do not belong to Table 3-1 or they are optional functions without being supplied.

Table 3-1 G command list

Word	Group	Function	Remark
G00	01	Rapid traverse movement	Initial modal G command
G01		Linear interpolation	Modal G commands
G02		Circular interpolation(CW)	
G03		Circular interpolation(CCW)	
G32		Thread cutting	
G33		Z tapping cycle	
G34		Variable pitch thread cutting	
G90		Axial cutting cycle	
G92		Thread cutting cycle	
G94		Radial cutting cycle	
G04	00	Dwell time preset	Non-modal G commands
G10		Data input valid	
G11		Data input cancel	
G12		Storage stroke detection ON	
G13		Storage stroke detection OFF	
G27		Reference point return check	
G28		Machine 1st reference point return	
G29		Machine reference point automatic return	
G30		Machine 2nd, 3rd,4th reference point return	
G31		Skip interpolation	

G50		Setting workpiece coordinates system	
G65		Macro command	
G70		Finishing cycle	
G71		Axial roughing cycle	
G72		Radial roughing cycle	
G73		Closed cutting cycle	
G74		Axial grooving cycle	
G75		Radial grooving cycle	
G76		Multiple thread cutting cycle	
G54	05	Workpiece coordinates 1	Modal G command
G55		Workpiece coordinates 2	
G56		Workpiece coordinates 3	
G57		Workpiece coordinates 4	
G58		Workpiece coordinates 5	
G59		Workpiece coordinates 6	
G20	06	Inch select	Modal G command
G21		Metric select	Initial mode G command
G96	02	Constant surface speed ON	Modal G command
G97		Constant surface speed OFF	Initial mode G command
G98	03	Feed per minute	Initial mode G command
G99		Feed per rev	Modal G command
G40	07	Cancel cutter radius compensation	Initial mode G command
G41		Tool nose radius compensation left contour (option)	Modal G command
G42		Tool nose radius compensation right contour(option)	
G17	16	XY plane	Modal G command
G18		ZX plane	Initial mode G command
G19		YZ plane	Modal G command

3.1.1 Modal, non-modal and initial mode

G commands are divided into group 00, 01, 02, 03, 06, 07 , 16, 21.

After G commands are executed, their defined functions and states are valid until they are changed by others in the same group, which commands are called modal G commands. After the modal G words are executed, and before their defined functions and states are changed, the G command cannot be input again when they are executed by the following block.

The defined function and state are valid one time after G command is executed, and the G word must be input again when it is executed every time, which command is called non-modal G command.

After the system is switched on, the valid modal G commands which are not executed their functions or states are called initial mode G command. Take it as the initial mode G command to be executed when it is not be input after the system is switched on.

3.1.2 Omitting words

To simplify the programming, their command values are reserved after executing words in Table 3-2. If the words are contained in the previous blocks, they cannot be input when the words are used with the same values and definitions in the following blocks.

Table 3-2

Command address	Function	Initial value when power-on
U	Cutting depth in G71	No.51 parameter value
U	Move distance of X tool retraction in G73	No.53 parameter value
W	Cutting depth in G72	No.51 parameter value
W	Move distance of X tool retraction in G73	No.54 parameter value
R	Move distance of tool retraction in G71, G72 cycle	No.52 parameter value
R	Cycle times of stock removal in turning in G73	No.55 parameter value
R	Move distance of tool retraction after cutting in G74, G75	No.56 parameter value
R	Allowance of finishing in G76	No.60 parameter value
R	Taper in G90, G92, G94, G96	0
(G98) F	Feedrate per minute(G98)	No.30 parameter value
(G99) F	Feedrate per rev (G99)	0
F	Metric pitch(G32, G92, G76)	0
I	Inch pitch(G32, G92)	0
S	Spindle speed specified(G97)	0
S	Spindle surface speed specified(G96)	0
S	Spindle speed switching value output	0
P	Finishing times of thread cutting in G76; Tool retraction width of thread cutting in G76 Angle of tool nose of thread cutting in G76;	No.57 parameter value No.19 parameter value No.58 parameter value
Q	Min. cutting value in G76	No.59 parameter value

Note 1: For the command addresses with functions (such as F, used for feedrate per minute, feedrate per revolution and metric pitch and so on), they can be omitted not to input when executing the same function to definite words after the words are executed. For example, after executing G98 F_ without executing the thread command, the pitch must be input with F word when machining metric thread.

Note 2: They can be omitted not to input when the address characters X(U) , Z(W) are the coordinates of end point of block and the system defaults the current absolute coordinates in X or Z direction to the coordinate value of end point of block.

Note 3: The corresponding words must be input when the command addresses which are not in Table 3-2 are used.

Example 1:

O0001;
G0 X100 Z100; (rapid traverse to X100 Z100; the modal G0 is valid)
X20 Z30; (rapid traverse to X20 Z30; the modal G0 is not input)
G1 X50 Z50 F300; (linear interpolation to X50 Z50, feedrate 300mm/min; the modal G1 is valid)
X100; (linear interpolation to X100 Z50, feedrate 300mm/min; Z coordinate is not input and is the current coordinates Z50; F300 is kept, G1 is modal and is not input)
G0 X0 Z0; (rapid traverse to X0 Z0 and the modal G0 is valid)
M30;

Example 2:

O0002;
G0 X50 Z5; (rapid traverse to X50 Z5)
G04 X4; (dwell 4 seconds)
G04 X5; (dwell 5 seconds again, G04 is non-modal and is needed to input again)
M30;

Example 3 (the first run after power-on) :

O0003;
G98 F500 G01 X100 Z100; (Feedrate per minute 500mm/min in G98)
G92 X50 W-20 F2 ; (F value is a pitch and must be input in thread cutting)
G99 G01 U10 F0.01 (Feedrate per revolution in G99 must be input again)
G00 X80 Z50 M30;

3.1.3 Related definitions

In the user manual, the definitions of Word are as follows except for the especial explanations:

Starting point: position before the current block runs;
End point: position after the current block ends;
X: X absolute coordinates of end point;
U: different value of absolute coordinates between starting point and end point;
Z: Z absolute coordinates of end point;
W: different value of absolute coordinates between starting point and end point;
F: cutting feedrate.

3.2 Rapid traverse movement G00

Command format: G00 X(U) __ Z(W) __;

Command function: X, Z rapidly traverses at the respective traverse speed to the end points from their starting point. G00 is initial command as Fig.3-1.
X, Z traverses at the respective traverse speed, the short axis arrives the end point and the length axis continuously moves to the end point and the compound path may be not linear.

Command specification: G00 is initial mode;
X, U, Z, W range: ±99999999× least input increment；

Can omit one or all command addresses X(U), Z(W). The coordinate values of starting point and end point are the same when omitting one command address; the end point and the starting point are in the same position when all are omitted. X, Z are valid, and U, W are invalid when X, U, Z and W are in the same one block.

Command path:

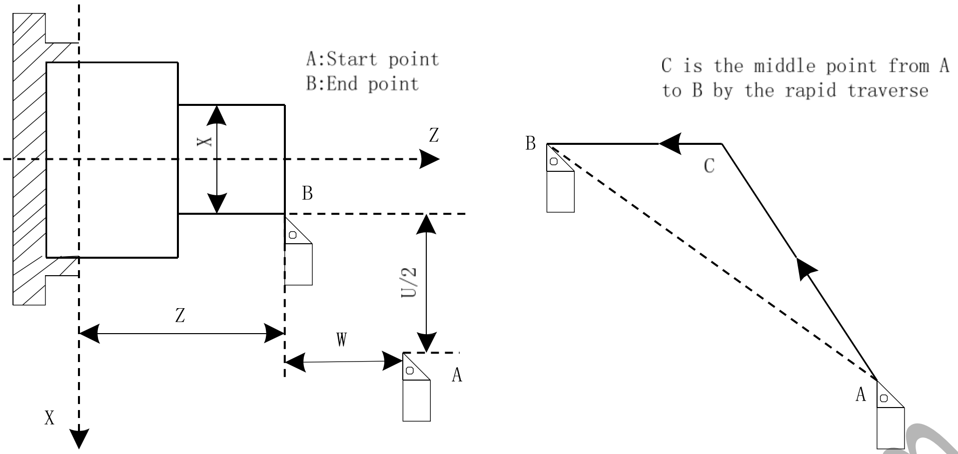


Fig. 3-1

The respective rapid traverse speed of X, Z is defined by the system parameter No.022, No.023, and their traverse speed can changed by rapid override key on the machine control panel.

Example: The tool rapidly traverses to B from A as Fig. 3-2.

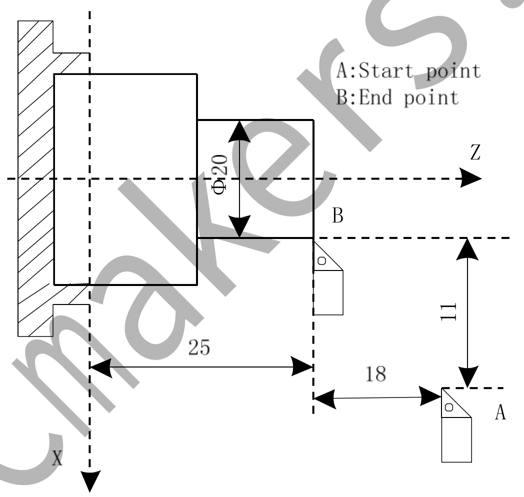


Fig. 3-2

G0 X20 Z25; (absolute programming)
G0 U-22 W-18; (incremental programming)
G0 X20 W-18; (compound programming)
G0 U-22 Z25; (compound programming)

3.3 Linear interpolation G01

Command format: G01 X(U) _ Z(W) _ F_;

Command function: The movement path is a straight line from starting point to end point as Fig.3-3.

Command specification: G01 is modal.

Can omit one or all command addresses X (U), Z (W). The coordinate values of starting point and end point are the same when omitting one command address; the end point and the starting point are in the same position when all are omitted.

F command value is the vector compound speed of X and Z instantaneous speed and the actual cutting feedrate is the product between the feedrate override and F command value.

After F command value is executed, it has been reserved unless the new one is executed. Do not repeat it when the following G commands adopt functions of F word. Its range is referred to Table 1-10.

Note: In G98, F max. value cannot exceed the value set by the data parameter No.027, otherwise, the system alarms.

Command path:

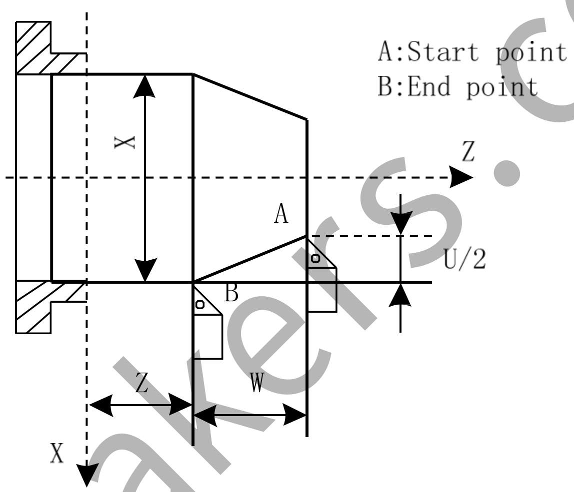
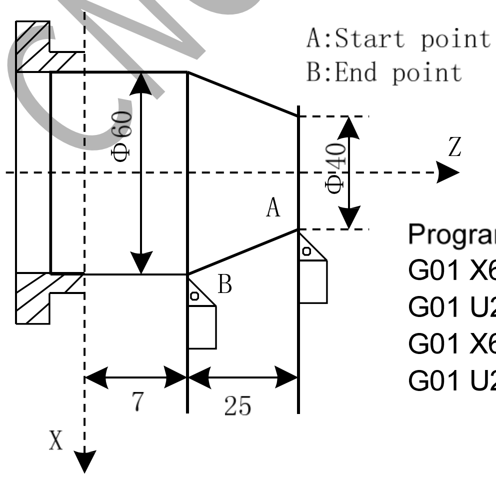


Fig. 3-3

Example: Cutting path from Φ40 to Φ60 as Fig.3-4:



Program:

G01 X60 Z7 F500; （Absolute programming）

G01 U20 W-25; （Incremental programming）

G01 X60 W-25; （Compound programming）

G01 U20 Z7; （Compound programming）

Fig. 3-4

3.4 Circular interpolation G02, G03

Command format:

G02	X(U)__ Z(W)__		R__
G03			I__ K__

Command function:

G02 movement path is clockwise (rear tool post coordinate system)/counterclockwise (front tool post coordinate system) arc from starting point to end point as Fig. 3-5(a).

G03 movement path is counterclockwise (rear tool post coordinate system/clockwise (front tool post coordinate system) arc from starting point to end point as Fig. 3-5(b).

Command path:

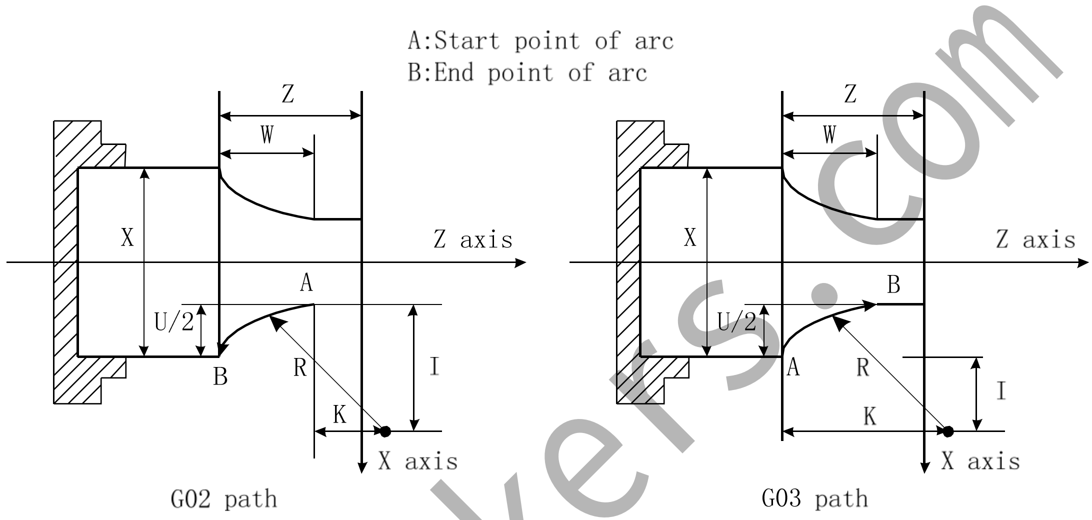


Fig. 3-5 G02 and G03 path

Command specification:

G02, G03 are modal,

R is arc radius, range: ±99999999× least input increment;

I: X difference value between circle center and starting point of arc in radius;

K: Z difference value between circle center and starting point of arc;

Center point of arc is specified by address I, K which separately corresponds to X, Z, I, K expresses the vector (it is the increment value) from starting point to center point of arc as the following figure;

I=Coordinates of center point-that of starting point in X direction; K= Coordinates of center point-that of starting point in Z direction;

I, K are with sign symbol. When directions of I, K are the same as those of X, Z, they are positive, otherwise, they are negative.

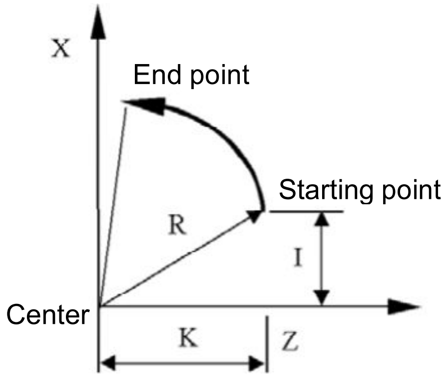


Fig. 3-6

Arc direction: G02/G03 direction (clockwise/counterclockwise) is opposite on the front tool post coordinate system and the rear one as Fig.3-7:

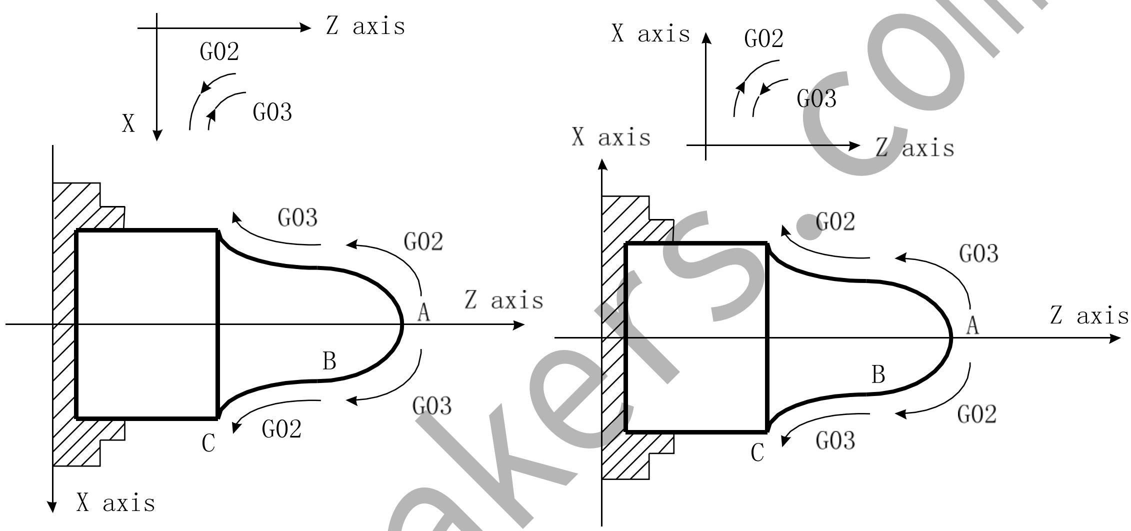
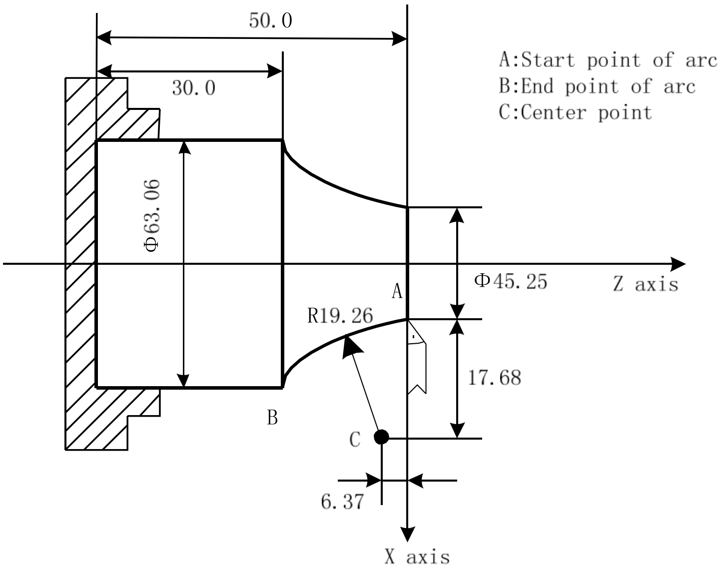


Fig. 3-7

Notes:

* When I = 0 or K = 0, they can be omitted; one of I, K or R must be input, otherwise the system alarms.
* R is valid and I, K are invalid when they are input at the same time.
* R value must be equal to or more than half distance from starting point to end point, and the system alarms if the end point is not on the arc defined by R command;
* Omit all or one of X(U), Z(W); coordinates of starting point and end point of this axis are the same when omitting ones, the path is a full circle(360°) in G02/G03 when center point are specified by I,K; the path is 0(0°) when center point is specified by R.
* R should be used for programming. The system executes in R= $\sqrt{I^2+K^2}$ to ensure starting point and end point of arc path are the specified ones in I, K programming.
* When the distance from center point to end point is not equal to R(R = $\sqrt{I^2+K^2}$) in I,K programming, the system automatically adjusts position of center point to ensure starting point and end point of arc path are the specified ones; when the distance from center point to end point is more than 2R, and the system alarms.
* Arc is less than 360° when R is commanded, the arc is more than 180° when R is negative, and it is less than or equal to 180° when R is positive.

Example: Arc cutting path from Φ45.25 to Φ63.06 shown in Fig. 3-8.



Program:
G02 X63.06 Z-20.0 R19.26 F300； or
G02 U17.81 W-20.0 R19.26 F300； or
G02 X63.06 Z-20.0 I17.68 K-6.37； or
G02 U17.81 W-20.0 I17.68 K-6.37 F300

Fig. 3-8

Compound programming in G02/G03:

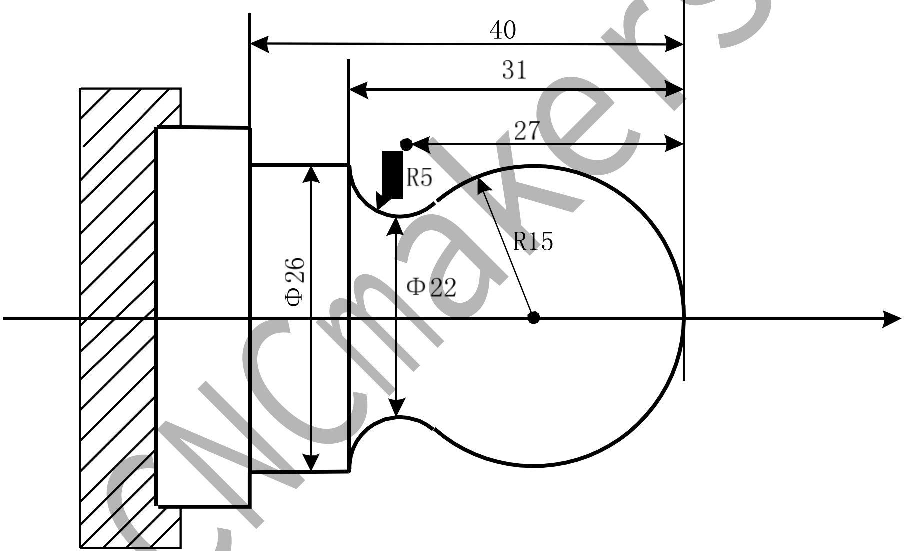


Fig. 3-9 Circular programming example

Program: O0001
N001 G0 X40 Z5; (Rapidly traverse)
N002 M03 S200; (Start spindle)
N003 G01 X0 Z0 F900; (Approach workpiece)
N005 G03 U24 W-24 R15; (Cut R15 arc)
N006 G02 X26 Z-31 R5; (Cut R5 arc)
N007 G01 Z-40; (Cut φ26)
N008 X40 Z5; (Return to starting point)
N009 M30; (End of program)

3.5 Plane selection G17 ~ G19

Command format：

G17……XY plane
G18……ZX plane
G19……YZ plane

Command function: use G commands to select the plane of the arc interpolation or the one of the cutter compensation

Command explanation: G17, G18, G19 are modal, and the plane does not change in the block without the command.

Notes:

- ＊ Firstly set the basic axis Y when the system selects G17, G19 plane;
- ＊ Cannot switch the planes in C tool compensation;
- ＊ G71~G76，G90，G92，G94 can be used in G18 plane;
- ＊ The plane selection code can be in the same block with G codes in the other groups;
- ＊ The movement command is not related to the plane selection;
- ＊ Diameter or radius programming: currently, because there is only one bit parameter No 1.2 to select the diameter or the radius programming and is valid to only X axis, Z and Y axis use the only radius programming in G2, G3, and X axis is selected by the parameter;
- ＊ The tool nose direction of C tool compensation is 0 in G17, G19.

3.6 Chamfering function

Chamfering function is to insert one straight line or circular between two contours to make the tool smoothly transmit from one contour to another one. C1000T uses the linear and circular chamfering functions.

3.6.1 Linear chamfering

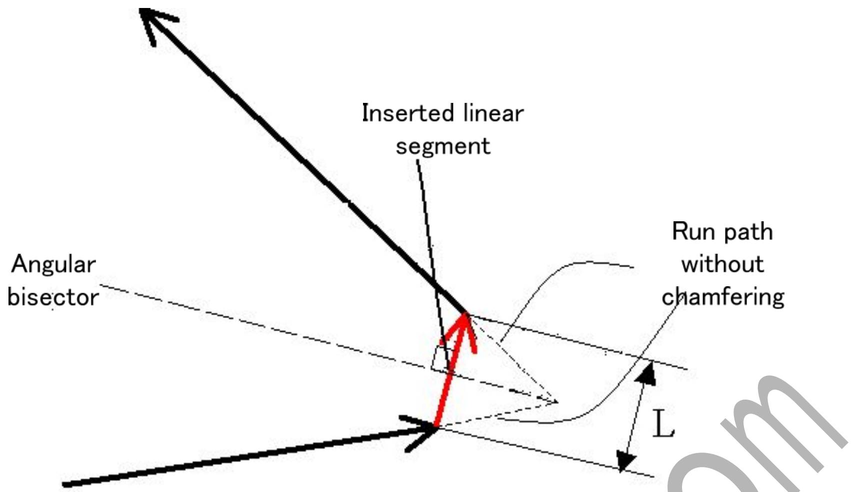
Linear chamfering: insert one straight line in the linear contours, arc contours, linear contour and arc contour. The command address of linear chamfering is L, behind which data is the length of chamfering straight line. The linear chamfering must be used in G01, G02 or G03 command.

A. Linear to linear

Command format: G01 X(U)_ Z(W)_ L_ ;
G01 X(U)_ Z(W)_ ;

Command function: insert one straight line between two linear interpolation blocks

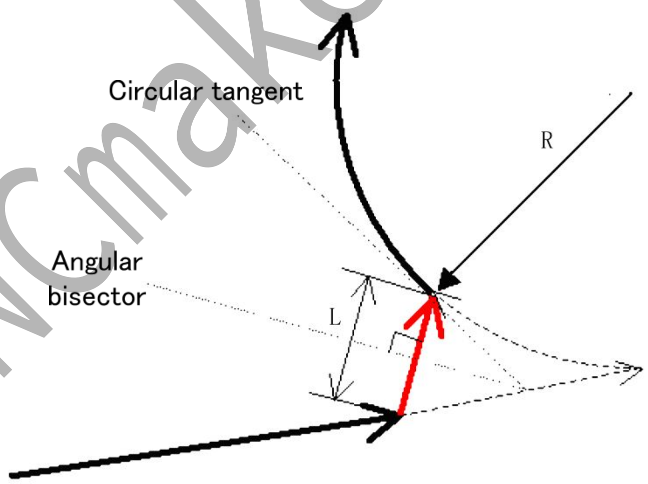
.



B. Linear to circular

Command format: G01 X(U)_ Z(W)_ L_;
G02/G03 X(U)_ Z(W)_ R_;
Or
G01 X(U)_ Z(W)_ L_;
G02/G03 X(U)_ Z(W)_ I_ K_;

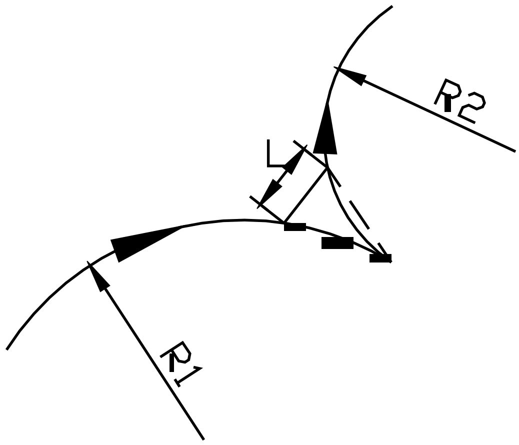
Command function: insert one straight line between the linear and circular interpolation blocks.



C. Circular to circular

Command format: G02/G03 X(U)_ Z(W)_ R_ L_;
G02/G03 X(U)_ Z(W)_ R_;
Or
G02/G03 X(U)_ Z(W)_ I_ K_ L_;
G02/G03 X(U)_ Z(W)_ I_ K_;

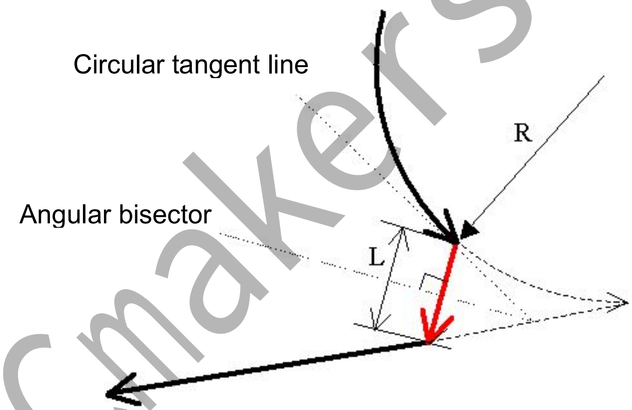
Command function: insert one straight line between two circular interpolation blocks.



D. Circular to linear

Command format: G02/G03 X(U)_ Z(W)_ R_ L_;
G01 X(U)_ Z(W)_;
Or
G02/G03 X(U)_ Z(W)_ I_ K_ L_;
G01 X(U)_ Z(W)_;

Command function: insert one straight line block between circular and linear interpolation block.



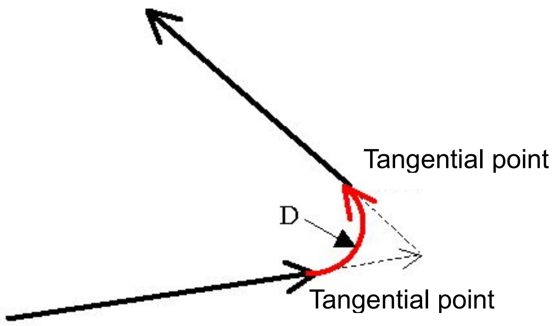
3.6.2 Circular chamfering

Circular chamfering: insert one circular between linear contours, circular contours, linear contour and circular contour, the circular and the contour line are transited by the tangent. The command of circular chamfering is D, and the data behind the command is the radius of chamfering circular. The circular chamfering must be used in G01, G02 or G03.

A. Linear to linear

Command format: G01 X(U)_ Z(W)_ D_;
G01 X(U)_ Z(W)_;

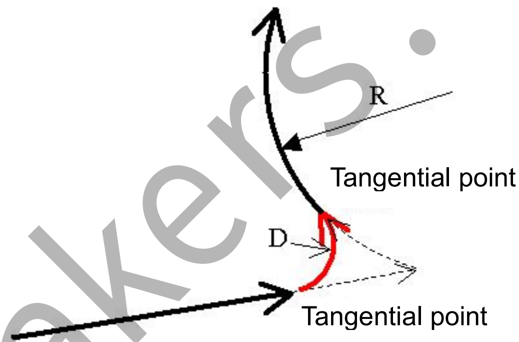
Command function: insert one circular between two straight lines, the inserted circular block and two straight lines are tangent, the radius is the data behind the command address D.



B. Linear to circular

Command format: G01 X(U)_ Z(W)_ D_;
G02/G03 X(U)_ Z(W)_ R_;
or
G01 X(U)_ Z(W)_ D_;
G02/G03 X(U)_ Z(W)_ I_ K_;

Command function: insert one circular between linear and circular, the inserted circular is tangent to the linear and the circular, and the radius is the data behind the command address D.

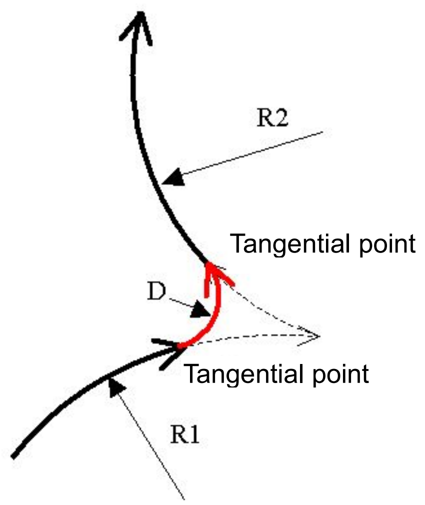


C. Circular to circular

Command format: G02/G03 X(U)_ Z(W)_ R_ D_;
G02/G03 X(U)_ Z(W)_ R_;

or
G02/G03 X(U)_ Z(W)_ R_ D_;
G02/G03 X(U)_ Z(W)_ I_ K_;
or
G02/G03 X(U)_ Z(W)_ I_ K_ D_;
G02/G03 X(U)_ Z(W)_ I_ K_;
or
G02/G03 X(U)_ Z(W)_ I_ K_ D_;
G02/G03 X(U)_ Z(W)_ R_;

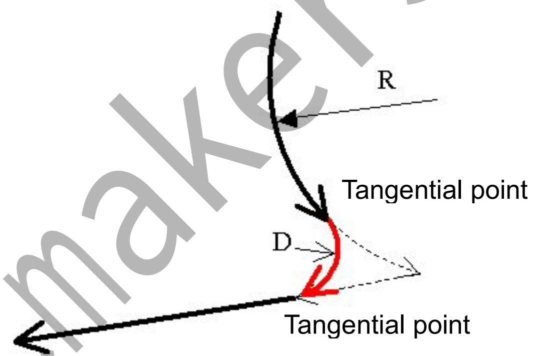
Command function: insert one circular between two circular blocks, the inserted circular is tangent to the two circular blocks, and the radius is the data behind the command address D.



D. Circular to linear

Command format: G02/G03 X(U)_ Z(W)_ R_ D_;
G01 X(U)_ Z(W)_;
Or
G02/G03 X(U)_ Z(W)_ I_ K_ D_;
G01 X(U)_ Z(W)_;

Command function: insert one circular block between the circular and the linear, the inserted circular block is tangent to the circular and the linear, and the radius is the data behind the command address D.



3.6.3 Special cases

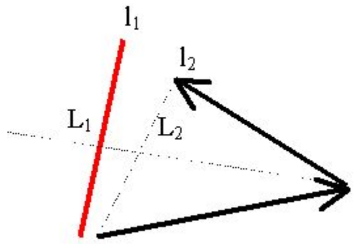
The chamfering function is invalid or alarms as follows:

1) Linear chamfering

A. The chamfering function is invalid when two interpolation straight lines are in the same linear.

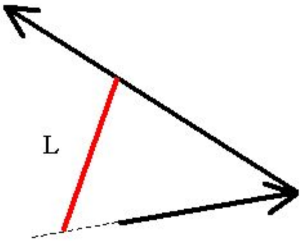
B. CNC alarms when the chamfering linear is too long.

L1 i is the chamfering linear, and the length is $L_{1;}$ l_2 is the third edge of the triangle which is formed by two interpolation straight lines, the length is L_2, CNC alarms when L_1 is bigger than L_2 as follows:



C. Some linear block is too short

The chamfering linear length is L, CNC alarms when other end of the caculated chamfering linear is not in the interpolation linear(in the extension line of the interpolation linear).

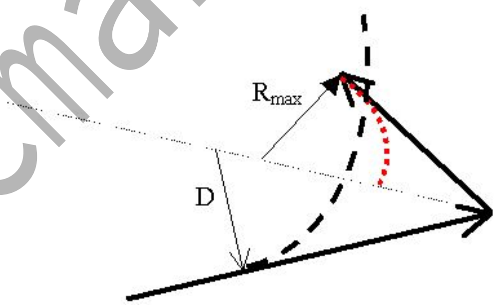


2) Circular chamfering

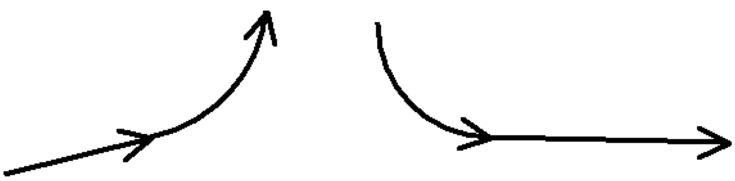
A. The circular chamfering function is invalid when two interpolation straight lines are in the same block.

B. CNC alarms when the chamfering circular radius is too big.

CNC alarms when the chamfering circular radius is D, max. circular radius of the tangential linear lines is R_{max} which is less than D as follows.

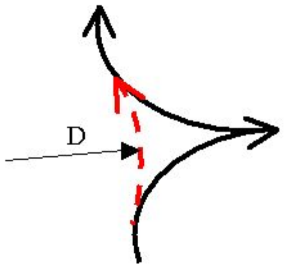


C. The circular chamfering function is invalid when the linear and the circular, or the circular and the linear are tangential.



D. The circular chamfering function is invalid when one circular and another one are tangential.

The circular chamfering function is valid when the circular tangency is as follows:



3.7 Dwell G04

Command format: G04 P__ ; or
G04 X__ ; or
G04 U__ ; or
G04;

Command function: each axis stops the motion, the modal of G commands and the reserved data, state are not changed, and execute the next block after dwelling the defined time.

Command specification: G04 is non-modal.
G04 dwell time is defined by the word P__, X__ or U__.
P range is 0 ~ 99999 (unit: ms) .
X, U range is 0 ~ 9999.999 x least input unit (unit: s)

Notes:
- The system exactly stop a block when P, X, U are not input
- P, X, U can not be in the same block;

3.8 Machine Zero function

3.8.1 Machine 1st reference point G28

Command format: G28 X/U Z/W ;

Command function: the tool rapid traverses to the middle point defined by X/U、Z/W from starting point and then return to the machine zero.

Command specifications:

G28 is non-modal.

X, Z: absolute coordinates of middle point;

U,W: Difference value of absolute coordinates between middle point and starting point in Z direction

.

Omit all or one of X/U ,Z/W as follows:

Table 3-4

Command	Function
G28 X/U	X returns to machine zero and Z/Y axis remain in the previous position
G28 Z/W	Z returns to machine zero and X/ Y axis remain in the previous position
G28	in the previous positions and continuously execute the next block
G28 X/U Z/W	X, Z axis return to machine zero simultaneously

Running path(as Fig. 3-12) :

(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 reference point return is completed.

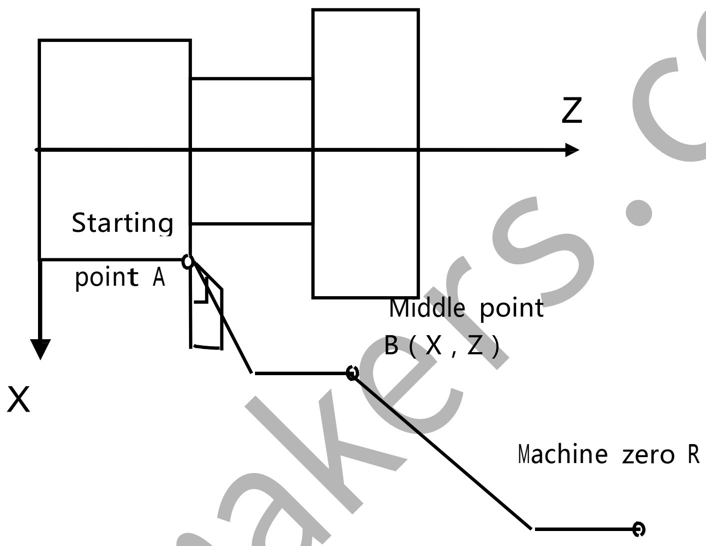


Fig.3-12

Note 1: Machine zero returns in Jog mode and in G28 are the same and their deceleration signals and signals per rev must be detected;

Note 2: X and Z move 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: The system cancels the tool length compensation after executing G28 to perform the machine zero return;

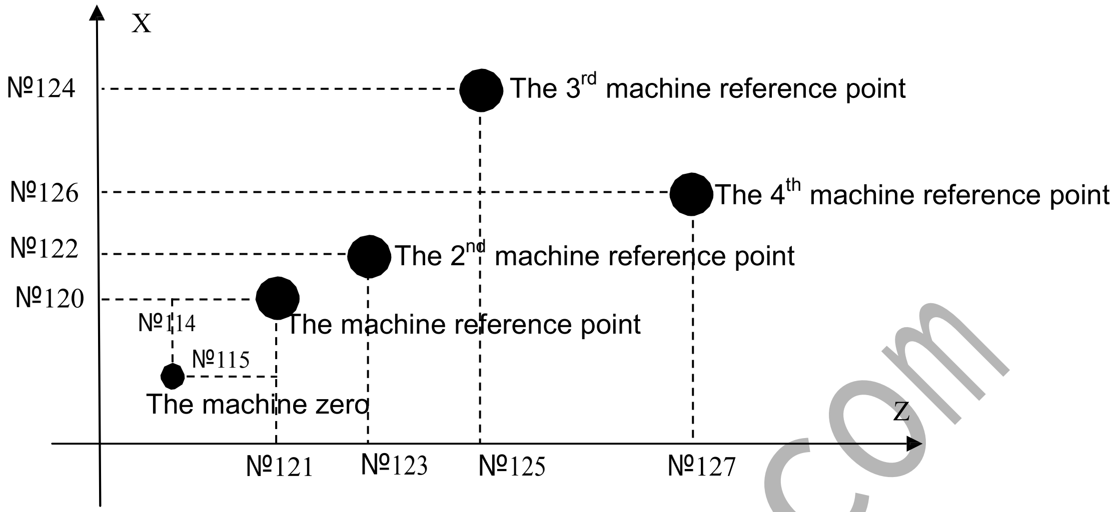
Note 4: Do not execute G28 and machine zero return without the zero switch on the machine.

3.8.2 Machine 2nd, 3rd, 4th reference point G30

Machine zero is fixed point in the machine tool, decided by the zero switch and zero return switch installed on the machine tool.The coordinates of machine reference point are No.120, No.121 setting value.

C1000T has machine 2^{nd}, 3^{rd}, 4^{th} reference point functions. Use separately No.122 ~ No.127 to set X, Z machine coordinates of the machine 2^{nd}, 3^{rd}, 4^{th} reference point.

The relationship between the machine zero, machine reference point, machine 2^{nd}, 3^{rd}, 4^{th} reference point is as follows:



Command format:

G30 P2 X/U ___ Z/W __;

G30 P3 X/U___ Z/W __;

G30 P4 X/U __ Z/W _ ;

Command function: the tool rapidly traverses with the rapid traverse speed to the middle point specified by X/U , Z/W and then return to machine 2^{nd}, 3^{rd}, 4^{th} reference point

Command specifications: G30 is non-modal.

X: X absolute coordinate of the middle point;

U: X relative coordinate of the middle point;

Z: Z absolute coordinate of the middle point;

W: Z relative coordinate of the middle point;

Omit one or all of X/U , Z/W as follows:

Command	Function
G30 Pn X/U ___	X returns to the machine nth reference point, Z axis retains
G30 Pn Z/W ___	Z return to the nth machine reference point, X axis retains
G30	X and Z retain, go on executing the next program block
G30 Pn X/U __ Z/W ___	X and Z return to the machine nth reference point simultaneously

Note 1: n in the above table is 2, 3 or 4;

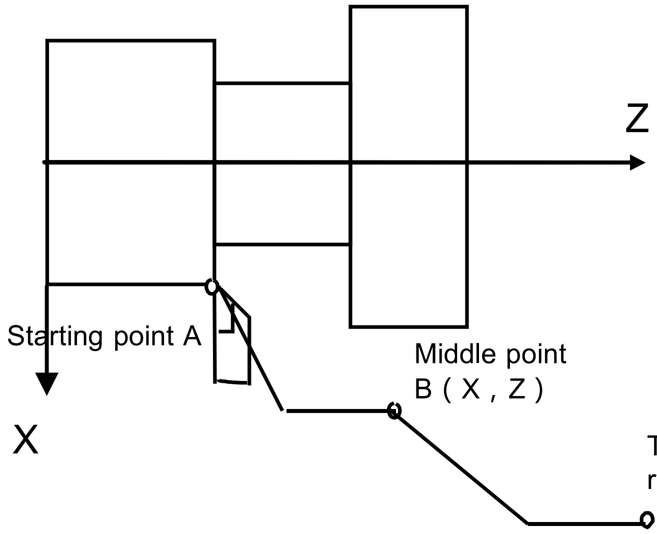
Note 2: Do not check the deceleration, zero signal when you execute the machine 2^{nd}, 3^{rd}, 4^{th} reference point.

Command operations: (taking example of returning to machine 2^{nd} reference point as follows):

(1) Rapidly traverse to the middle position of command axis from the current position (A point →B point);

(2) Traverse from the middle point with the speed set by No.113 to the 2nd reference point set by No.122 and No.123 (B point →R2 point);

(3) When CNC is not in the machine lock state, the completion signal of reference point return ZP21 Bit0, Bit1 is high.



Note 1: Execute the machine 2nd, 3rd, 4th reference point return after you manually execute the machine reference point return or G28 (machine reference point return).

Note 2: A→B and B→R2, two axes separately traverse, and so their trails are linear or not.

Note 3: CNC cancels the tool length compensation after you execute G30 to return 2nd, 3rd, and 4th reference point.

Note 4: Must not execute G30 (machine 2nd, 3rd, 4th reference point return) when the zero switch is not installed on the machine.

Note 5: Do not set the workpiece coordinate system when you execute the 2nd, 3rd, and the machine 4th reference point return.

3.9 Skip interpolation G31

Command format: G31 X/U_ Z/W_ F_;

Command function: in executing the command, when the outside skip signal (X3.5) is input, the system stops the command to execute the next block. The function is used to the dynamic measure (such as milling machine), toolsetting measure and so on of workpiece measure.

Command explanations: non-modal G command (00 group);

Its address format and usage are same that of G01;

Cancel the tool nose radius compensation before using it;

Feedrate should not be set to too big to get the precise stop position;

a. following block execution after skip:

1. The next block of G31 is the incremental coordinate programming shown in Fig. 3-13:

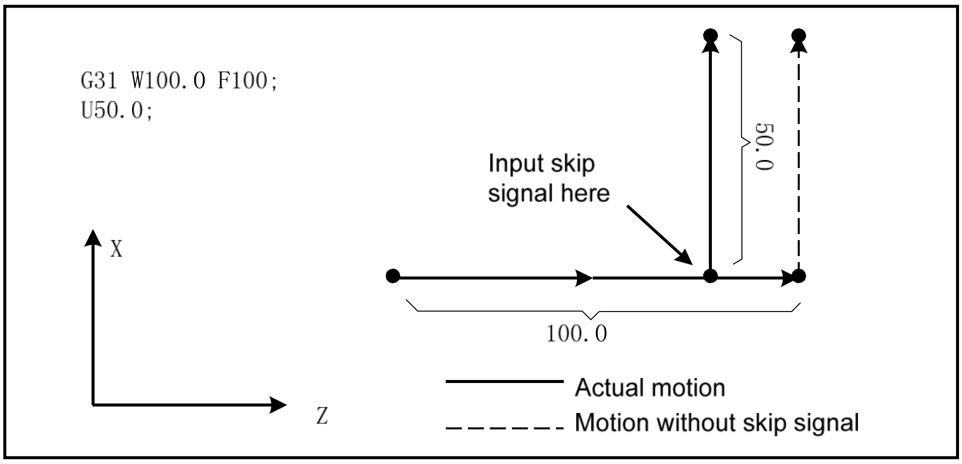


Fig. 3-13

2. The next block of G31 is the absolute coordinate programming of one axis as Fig. 3-14:

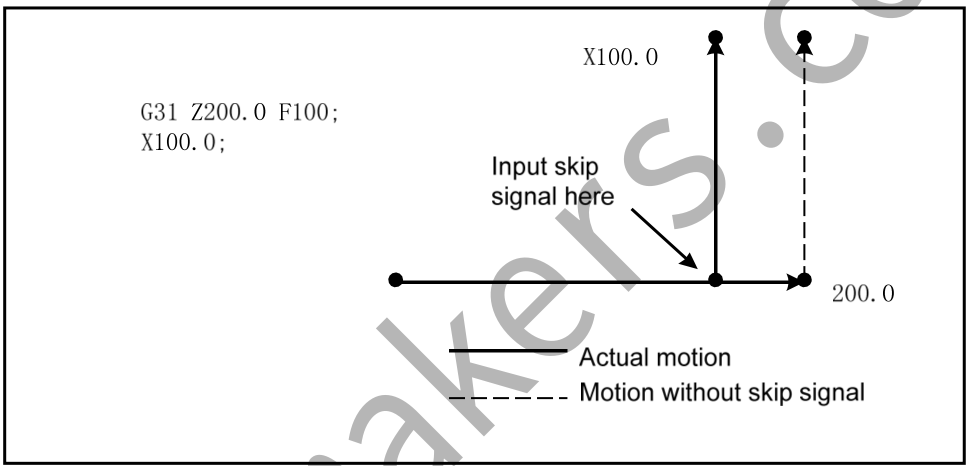


Fig. 3-14

3. The next block of G31 is the absolute coordinate programming of two axes shown in Fig. 3-15:

Program: G31 Z200 F100

G01 X100 Z300

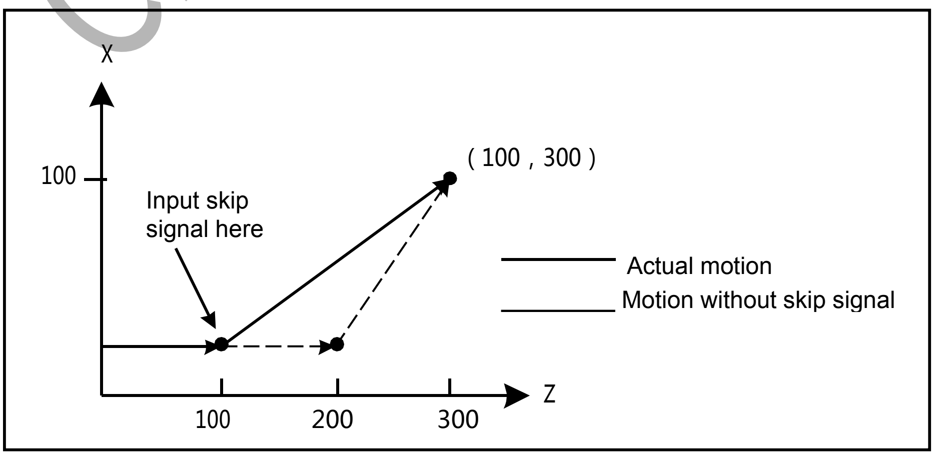


Fig. 3-15

b. Signals related to G31

Skip signal:

SKIP: G6.6

Type: input signal

Function: G6.6 ends the skip cutting. I.e. in a block containing G31, the skip signal becoming the absolute coordinate position of “1” is to be stored in the macro variable (#5011～#5015 separately corresponds to X, Z, Y,4th,5th)

Operation: when the skip signal becomes “1”, CNC executes as follows:

When the block is executing G31, CNC stores the current absolute coordinates of each axis. CNC stops G31 to execute the next block, the skip signal detects its state instead of its RISING EDGE. So when the skip signal is “1”, it meets the skip conditions.

Note: CNC immediately stops the feed axis (without acceleration/deceleration execution), and G31 feedrate should be as low as possible below 1000 mm/min to get the precise stop position.

3.10 Workpiece coordinate system G50

Command format: G50 X/U Z/W ;

Command function: define the absolute coordinates of current position and create the workpiece coordinates system (called floating coordinates system) by setting the absolute coordinates of current position in the system. After G50 is executed, the system takes the current position as the program zero (program reference point), and the system returns to the point after executing the program zero return. After the workpiece coordinate system is created, input the coordinate values with the coordinate system in the absolute coordinates programming until the next workpiece coordinate system is created again (using G50).

Command specifications:

G50 is non-modal;

X: New absolute coordinates of current position in X direction;

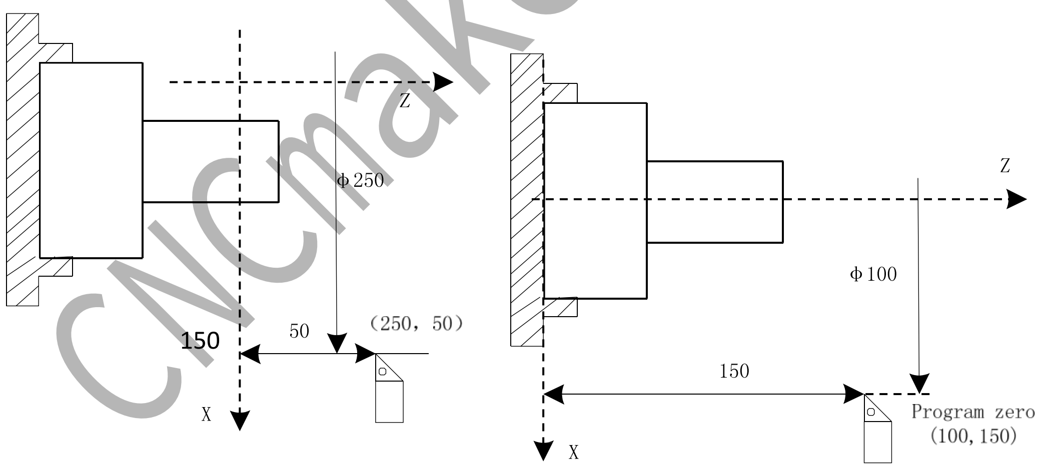
U: Different value between the new absolute coordinates of current position in X direction and the absolute coordinates before executing commands;

Z: New absolute coordinates of current position in Z direction;

W: Different value between the new absolute coordinates of current position in X direction and the absolute coordinates before executing commands;

In G50, when X/U、Z/W are not input, the system does not change current coordinates position as program zero; (In G50 SXXXX,not set program zero)

Example:



Before setting coordinate system with G50 **After setting coordinate system with G50**

Fig.3-16

As Fig.3-16, create the above-mentioned workpiece coordinate system and set (X100 Z150) to program zero point after executing “G50 X100 Z150”.

3.11 Workpiece coordinate system G54 ~ G59

Format: G54 ~ G59

Function: It specifies the current workpiece coordinate system. It is used to select workpiece coordinate system by specifying workpiece coordinate system G code in program.

Explanation:

1. No instruction parameter.
2. 6 workpiece coordinate systems can be set in the system, any of which can be selected

b by G54~G59 instruction.

G54 ---------------- Workpiece coordinate system 1

G55 ---------------- Workpiece coordinate system 2

G56 ---------------- Workpiece coordinate system 3

G57 ---------------- Workpiece coordinate system 4

G58 ---------------- Workpiece coordinate system 5

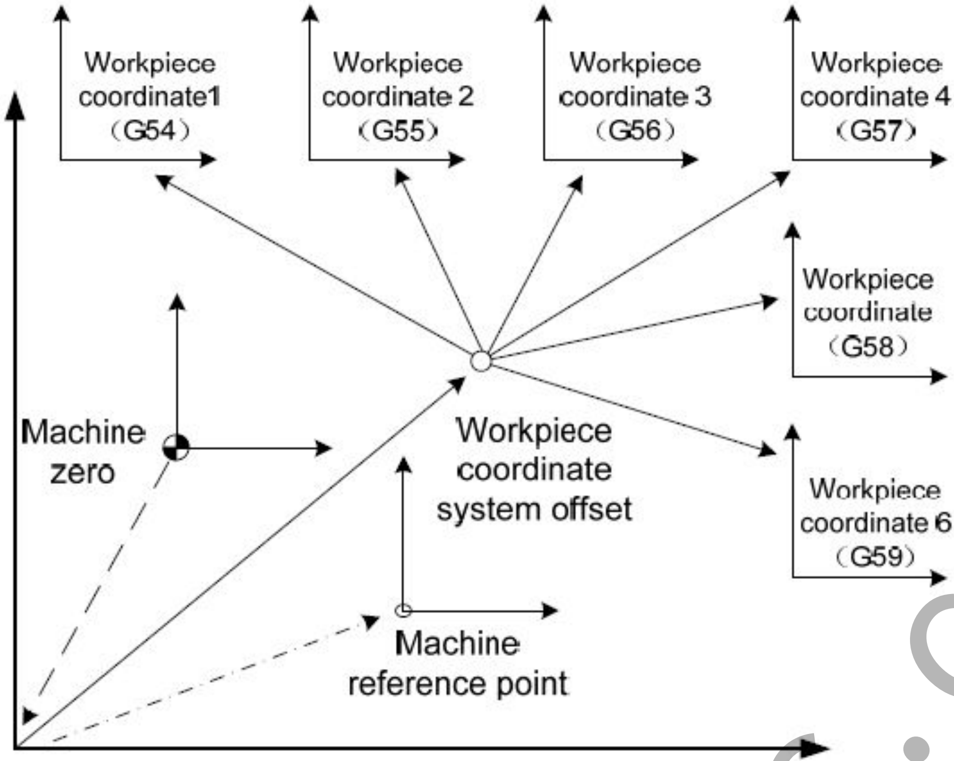
G59 ---------------- Workpiece coordinate system 6

3. When different workpiece coordinate system is called by block, the axis for move by instruction will be located in the new workpiece coordinate system; for the coordinate of the axis not move, It turns to the corresponding coordinate in the new workpiece coordinate system and the actual machine position doesn't change.

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

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

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



As shown in Fig. 4-2-8-1, after power-on, the machine returns to machine zero by manual zero return. The machine coordinate system is set up by machine zero with the machine reference point generating and workpiece coordinate system to be defined. The corresponding values of offset number parameter P270～274 in workpiece coordinate system are the integral offset of the 6 workpiece coordinate system. The 6 workpiece coordinate system origins can be specified by coordinate offset input in MDI mode or set by number parameter P128～139,P275~P292. These 6 workpiece coordinate systems are set up by the distances from machine zero to each coordinate system origin

***Example*：**

N10 G55 G90 G00 X100 Y20；

N20 G56 X80.5 Z25.5；

For the example above, when N10 block is being executed, it rapidly traverses to a position （X=100，Y=20）in G55 workpiece coordinate system.

When N20 block is being executed, the absolute coordinate value automatically turns to the coordinate value （X=80.5，Z=25.5）in G55 workpiece coordinate system for rapid positioning.

3.12 Fixed cycle command

To simplify programming, the system defines G command of single machining cycle with one block to complete the rapid traverse to position, linear/thread cutting and rapid traverse to return to the starting point:

G90: axial cutting cycle;

G92: thread cutting cycle;

G94: radial cutting cycle;

G92 will be introduced in section Thread Function.

3.12.1 Axial cutting cycle G90

Command format: G90 X/U_ Z/W_ F_ ; (cylinder cutting)

G90 X/U_ Z/W_ R_ F_ ; (taper cutting)

Command function: From starting point, the cutting cycle of cylindrical surface or taper surface is completed by radial feeding(X) and axial (Z or X and Z) cutting.

Command specifications:

G90 is modal;

Starting point of cutting: starting position of linear interpolation(cutting feed)

End point of cutting: end position of linear interpolation(cutting feed)

X: X absolute coordinates of cutting end point

U: different value of X absolute coordinate between end point and starting point of cutting

Z: Z absolute coordinates of cutting end point

W: different value of Z absolute coordinate between end point and starting point of cutting

R: different value (radius value) of X absolute coordinates between end point and start point of cutting. When the signs of R is not the same that of U, R│≤│U/2│; when R＝0 or the input is default, the cylinder cutting is executed as Fig.3-17, otherwise, the cone cutting is executed as Fig. 3-18; unit: mm.

Cycle process:

① X rapidly traverses from starting point to cutting starting point;

② Cutting feed (linear interpolation) from the cutting starting point to cutting end point;

③ X executes the tool retraction at feedrate (opposite direction to the above-mentioned ①), and return to the position which the absolute coordinates and the starting point are the same;

④ Z rapidly traverses to return to the starting point and the cycle is completed.

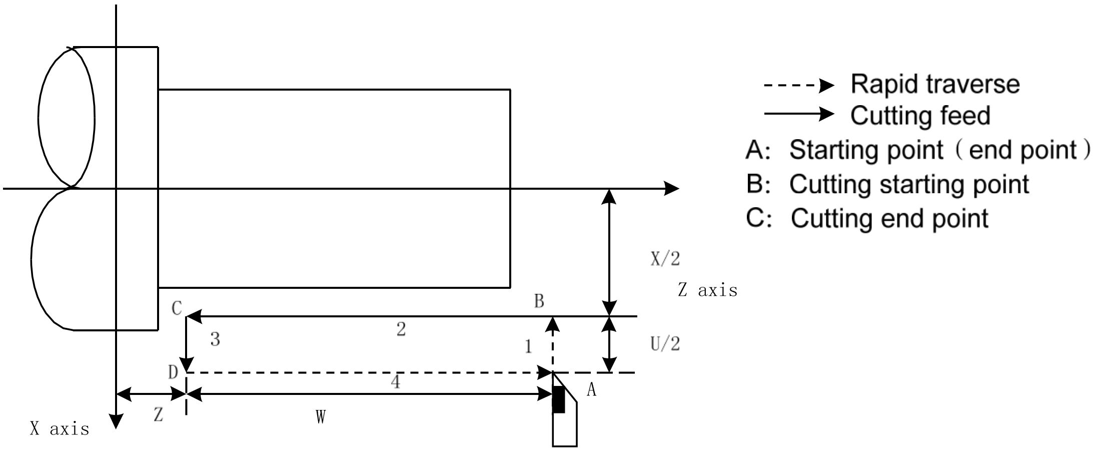


Fig. 3-17

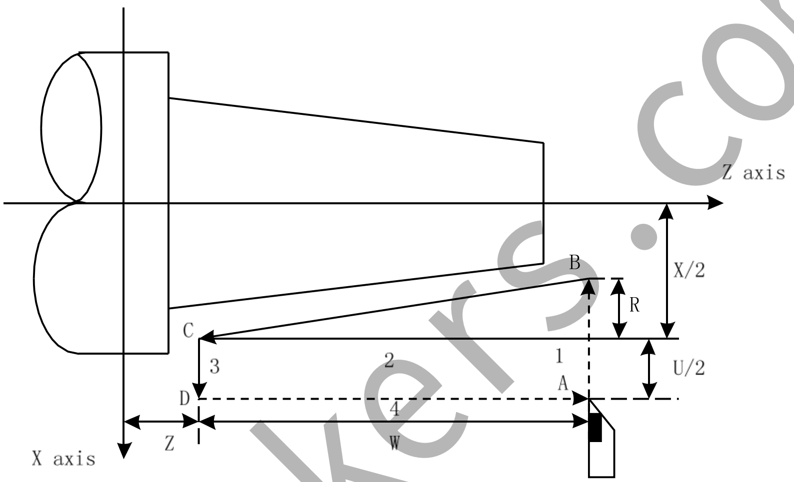
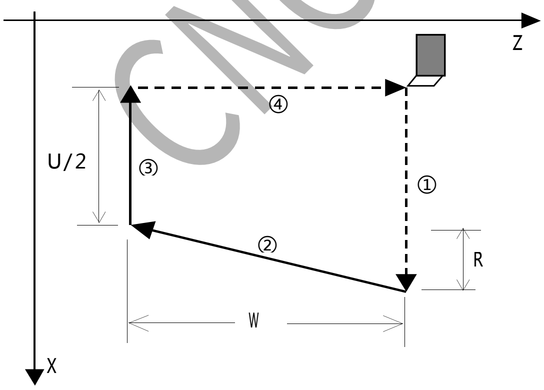


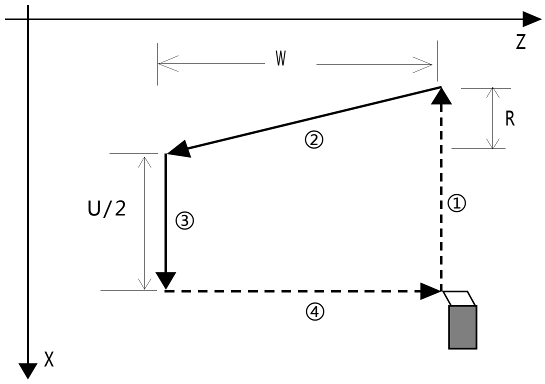
Fig. 3-18

Cutting path: Relative position between cutting end point and starting point with U, W, R, and tool path of U, W, R with different signs are shown in Fig. 3-19:

1） U>0，W<0，R>0



2） U<0，W<0，R<0



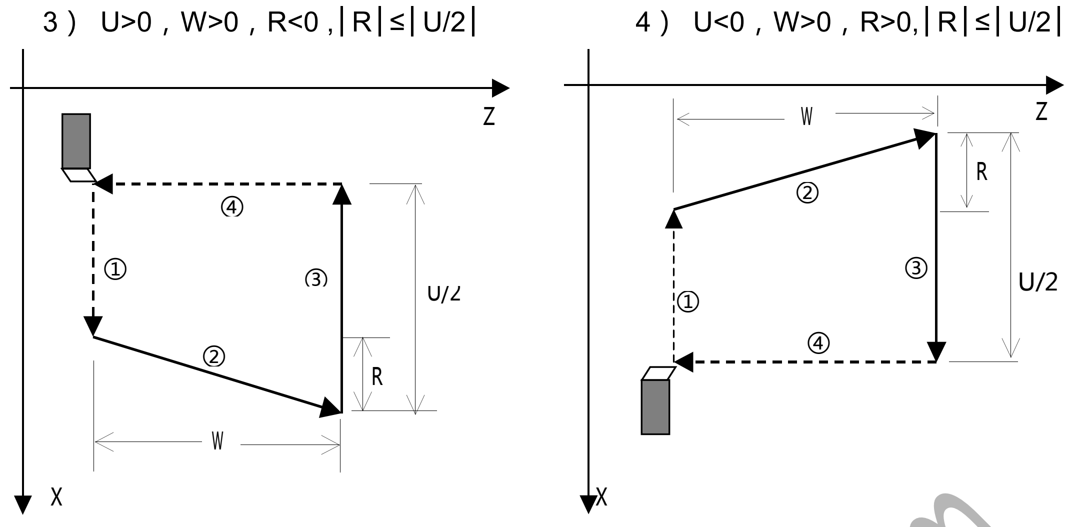


Fig. 3-19

Example: Fig. 3-20 rod Φ125×110

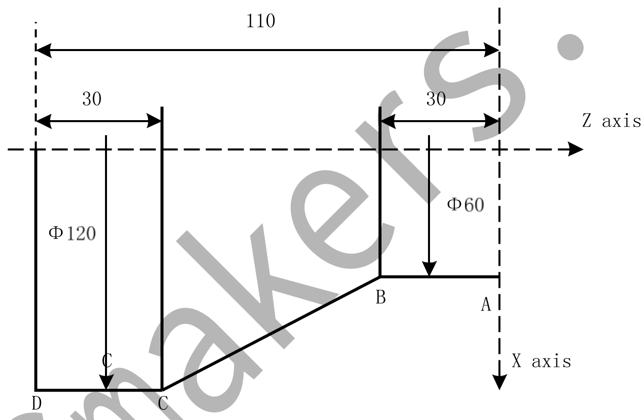


Fig. 3-20

Program：O0002;
M3 S300 G0 X130 Z3;
G90 X120 Z-110 F200; (A→D, cut Φ120)
X110 Z-30;
X100;
X90;
X80;
X70;
X60;
(A→B，6 times cutting cycle Φ60, increment of 10mm)
G0 X120 Z-30;
G90 X120 Z-44 R-7.5 F150;
Z-56 R-15
Z-68 R-22.5
Z-80 R-30
(B→C，4 times taper cutting)
M30;

3.12.2 Radial cutting cycle G94

Command format: G94 X/U __ Z/W __ F__; (face cutting)

G94 X/U __ Z/W __ R__ F__; (taper face cutting)

Command function: From starting point, the cutting cycle of cylindrical surface or taper surface is completed by axial feeding(Z) and radial (X or X and Z) cutting.

Command specifications:

G94 is modal;

Starting point of cutting: starting position of linear interpolation (cutting feed). Unit: mm;

End point of cutting: end position of linear interpolation (cutting feed). Unit: mm;

X: X absolute coordinate of end point of cutting. Unit: mm;

U: Different value of absolute coordinate from end point to starting point of cutting in X direction .Unit: mm;

Z: Z absolute coordinates of end point of cutting, Unit: mm;

W: Different value of X absolute coordinate from end point to starting point of cutting, Unit: mm;

R: Different value(R value) of X absolute coordinates from end point to starting point of cutting. When the sign of R is not the same as that of U, R, │R│≤│W│.

Radial linear cutting is shown in Fig. 3-21, radial taper cutting is as Fig. 3-22. Unit: mm

Cycle process:

① Z rapidly traverses from starting point to cutting starting point;

② Cutting feed (linear interpolation) from the cutting starting point to cutting end point;

③ Z executes the tool retraction at the cutting feedrate (opposite direction to the above-mentioned ①), and returns to the position which the absolute coordinates and the starting point are the same;

④ X rapidly traverses to return to the starting point and the cycle is completed.

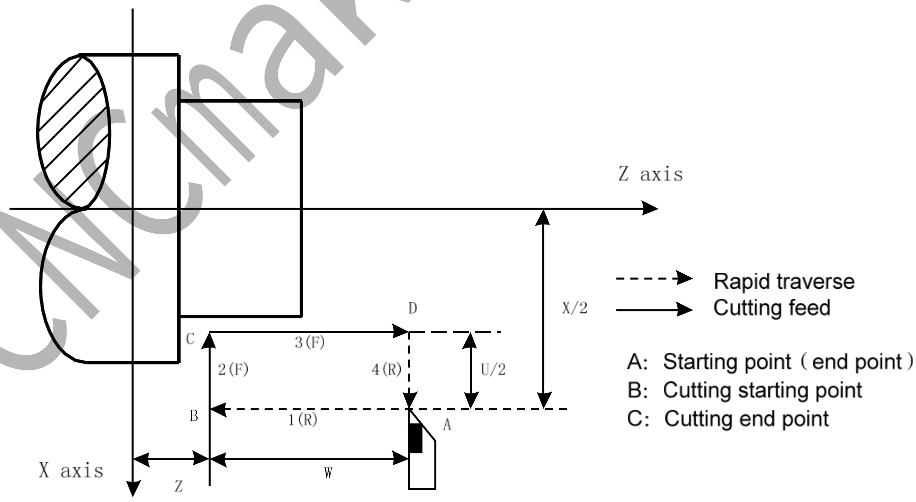


Fig. 3-21

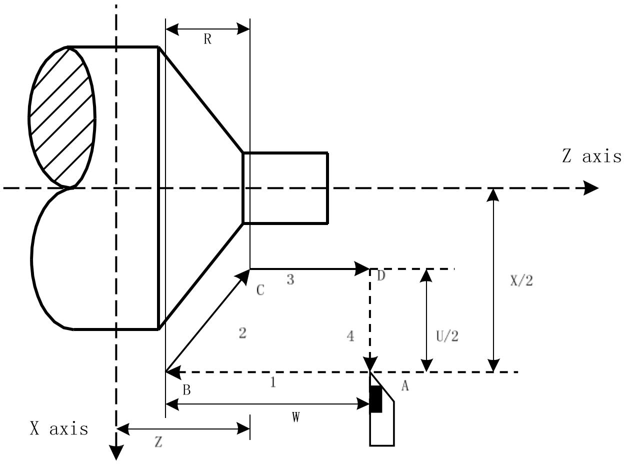


Fig. 3-22

Cutting path: Relative position between cutting end point and starting point with U, W,R is shown in Fig.3-23:

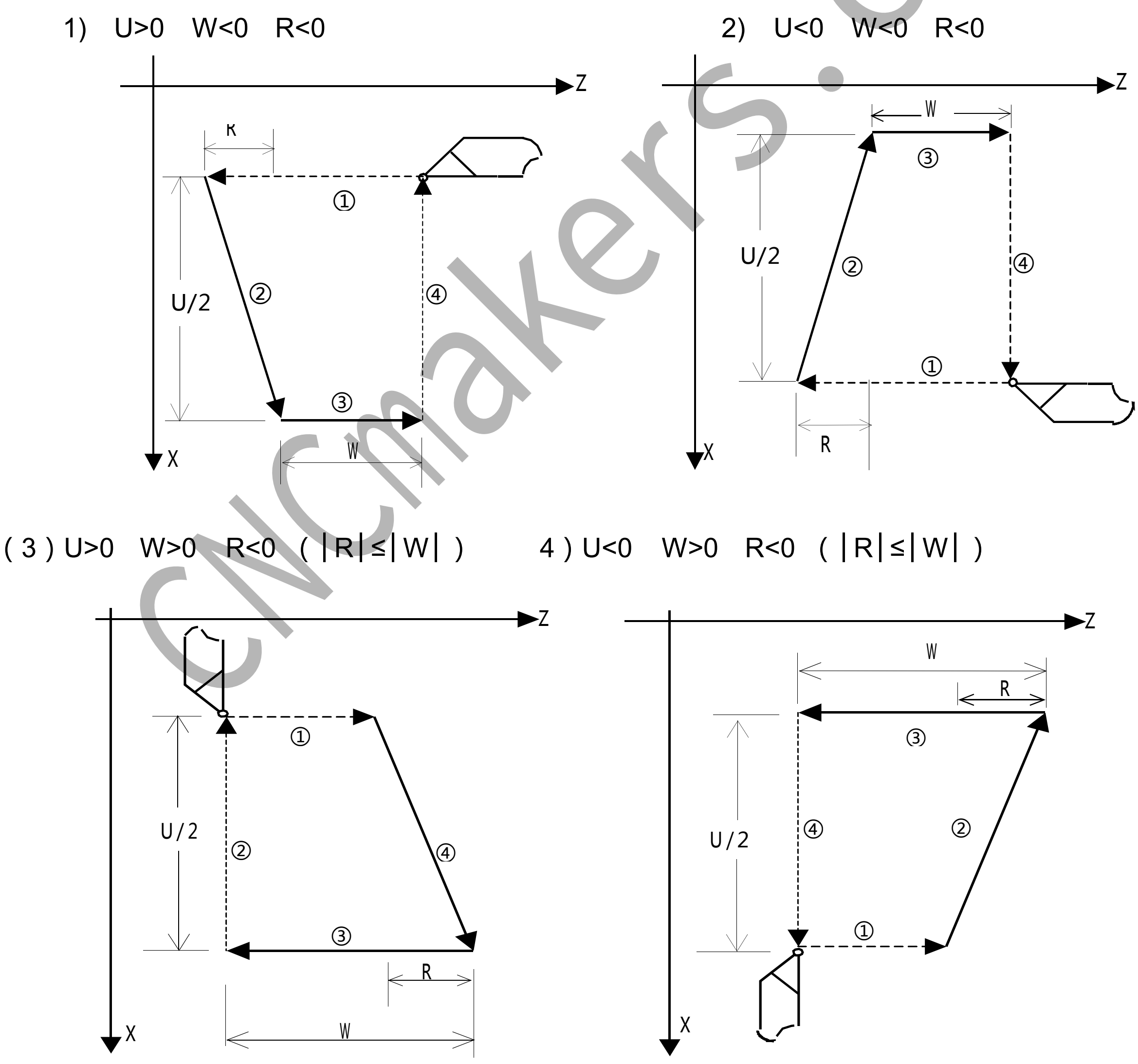


Fig. 3-23

Example: Fig. 3-24, rod Φ125×112

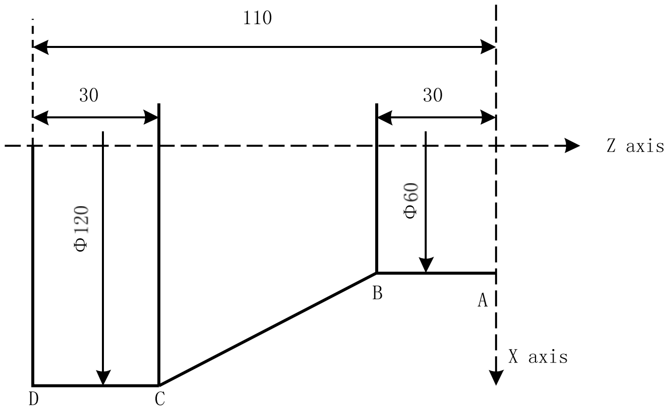


Fig. 3-24

Program: O0003;

```
G00 X130 Z5 M3 S1;
G94 X0 Z0 F200            End face cutting
X120 Z-110 F300;          ( Outer cutting Φ120 )
G00 X120 Z0
G94 X108 Z-30 R-10
X96 R-20
X84 R-30                  ( C→B→A , cutting Φ60 )
X72 R-40
X60 R-50;
M30;
```

3.12.3 Caution of fixed cycle commands

1) After X(U) , Z(W) , R are executed in the canned cycle command, their command values are valid if X(U) , Z(W) ,R are not redefined by executing a new canned cycle commands. The command values of X(U) ,Z(W) ,R are cleared if non-modal G command(00 Group) except for G04 or G00, G01, G02, G03, G32 is executed.

2) Pause or single block is executed in G90, G94, the single block stops after the tool moves end point of current path.

3.13 Multiple cycle commands

Multiple cycle commands 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 system executes these commands, it automatically counts the cutting times and the cutting path according to the programmed path, travels of tool infeed and tool retraction, executes multiple machining cycle (tool infeed → cutting→retract tool→tool infeed), automatically completes the roughing, finishing workpiece and the starting point and the end point of command are the same one.

3.13.1 Axial roughing cycle G71

Command format：G71 U(Δd) R(e) F__ S__ T__ ; (1)

G71 P(ns) Q(nf) U(Δu) W(Δw) K0/1 ; (2)

```
N(ns) G0/G1 X(U) . . ;    ┐
 . . . . . . . . . ;      │
   . . . . F ;            │
   . . . . S ;            ├ (3)
   . . . .                │
N(nf) . . . . . ;         ┘
```

Command function: G71 is divided into three parts:

(1) 1st blocks for defining the travels of tool infeed and retract tool, the cutting feedrate, the spindle speed and the tool function when roughing;

(2) 2nd blocks for defining the block interval, finishing allowance;

(3) 3rd blocks for some continuous finishing path, counting the roughing path without being executed actually when executing G71.

According to the finishing path, the finishing allowance, the path of tool infeed and tool retract, the system automatically counts the path of roughing, the tool cuts the workpiece in paralleling with Z, and the roughing is completed by multiple executing the cutting cycle tool infeed→cutting→tool retraction. The starting point and the end point are the same one. The command is applied to the formed roughing of non-formed rod.

Relevant definitions:

Finishing path: The above-mentioned Part 3 of G71**(**ns ~ nf block)defines the finishing path, and the starting point of finishing path (starting point of ns block)is the same these of starting point and end point of G71, called A point; the first block of finishing path(ns block)is used for X rapid traversing or tool infeed, and the end point of finishing path is called to B point; the end point of finishing path(end point of nf block)is called to C point. The finishing path is A→B→C.

Roughing path: The finishing path is the one after offsetting the finishing allowance(Δu,Δw) and is the path contour formed by executing G71. A, B, C point of finishing path after offset corresponds separately to A', B', C' point of roughing path, and the final continuous cutting path of G71 is B'→C' point.

Δd: It is each travel(unit: mm, radius value) of X tool infeed in roughing, its value: 0.001 （IS_B） /0.0001 （IS_C） ~99.999(unit: mm,radius value) without sign, and the direction of tool infeed is defined by move direction of ns block. The command value Δd is reserved after executing U(Δd) and the value of system parameter No.051 is rewritten to Δd×1000(unit: 0.001 mm) . The value of system parameter No.051 is regarded as the travel of tool infeed when U(Δd) is not input.

e: It is travel(unit: mm, radius value) of X tool retraction in roughing its value: 0~99.999(unit: mm,radius value) without sign, and the direction of tool retraction is opposite to that of tool infeed, the command value e is reserved and the value of system parameter No.052 is rewritten to e×1000(unit: 0.001 mm) after R(e) is executed. The value of system parameter No.052 is regarded as the travel of tool retraction when R(e) is not input.

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

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

Δu: X finishing allowance is ±99999.999× least input increment with sign symbol (diameter). X coordinate offset of roughing path compared to finishing path, i.e. the different value of X absolute coordinates between A'and A. The system defaults Δu=0 when U(Δu) is not input, i.e. there is no finishing allowance in X direction for roughing cycle.

Δw: Z finishing allowance is ±99999.999× least input increment with sign symbol (diameter). the Z coordinate offset of roughing path compared to finishing path, i.e. the different value of Z absolute coordinate between A' and A. The system defaults Δw=0 when W(Δw) is not input, i.e. there is no Z finishing allowance for roughing cycle.

K: When K is not input or is not 1, the system does not check the program monotonicity except that the Z value of starting point and end point of the arc or ellipse or parabola or the arc is more than 180 degree; K=1, the system checks the program monotonicity.

F: Feedrate; S: Spindle speed; T: Tool number, tool offset number.

M, S, T, F: They can be specified in the first G71 or the second ones or program ns～nf. M, S, T, F functions of M, S, T, F blocks are invalid in G71, and they are valid in G70 finishing blocks.

Type I：

1）Execution process: (Fig. 3-25)

① X rapidly traverses to A' from A point, X travel is Δu, and Z travel is Δw;

② X moves from A'is Δd(tool infeed), ns block is for tool infeed at rapid traverse speed with G0, is for tool infeed at feedrate F with G71, and its direction of tool infeed is that of A→B point;

③ Z executes the cutting feeds to the roughing path, and its direction is the same that of Z coordinate B→C point;

④ X, Z execute the tool retraction e (45° straight line) at feedrate, the directions of tool retraction is opposite to that of too infeed;

⑤ Z rapidly retracts at rapid traverse speed to the position which is the same that of Z coordinate;

⑥ After executing X tool infeed (Δd+e)again, the end point of traversing tool is still on the middle point of straight line between A' and B'(the tool does not reach or exceed B'), and after executing the tool infeed (Δd+e)again, execute ③; after executing the tool infeed (Δd+e)again, the end point of tool traversing reaches B' point or exceeds the straight line between A'→B' point and X executes the tool infeed to B' point, and then the next step is executed;

⑦ Cutting feed from B' to C' point along the roughing path;

⑧ Rapid traverse to A from C' point and the program jumps to the next clock following nf block after G71 cycle is ended.

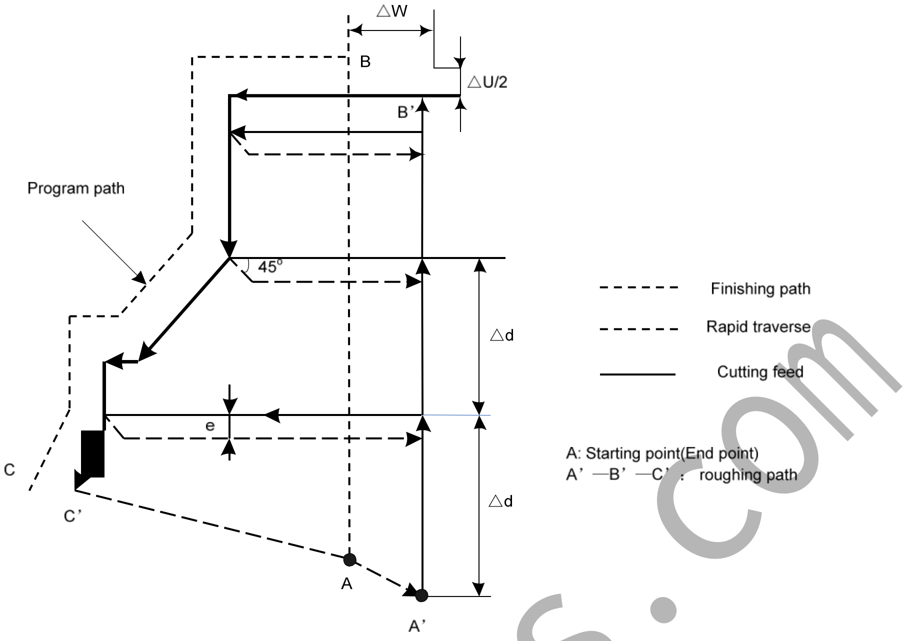


Fig. 3-25 G71 cycle path

2）Coordinate offset direction with finishing allowance:

Δu, Δw define the coordinate offset and cut-in direction in finishing, and their sign symbol are as follows Fig. 3-26: B→C for finishing path, B'→C' for roughing path and A is the tool start-up point.

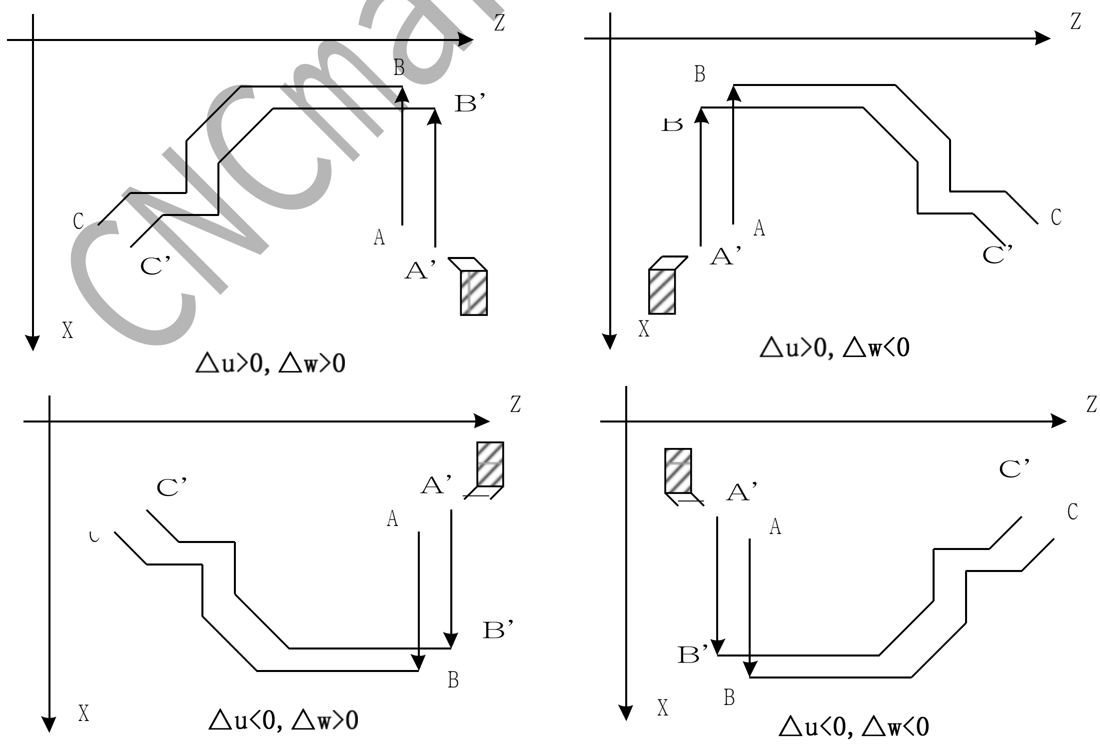


Fig.3-26

Notes：

- ns block is only G00, G01.
- For the finishing path(ns～nf block)，Z dimension must be monotonous change(always increasing or decreasing)
- ns～nf blocks in programming must be followed G71 blocks.
- ns～nf blocks are used for counting the roughing path and the blocks are not executed when G71 is executed. F, S, T commands of ns～nf blocks are invalid when G71 is executed, at the moment, F, S, T commands of G71 blocks are valid. F, S, T of ns～nf blocks are valid when executing ns～nf to command G70 finishing cycle;
- In ns～nf blocks, there are only G commands: G00, G01, G02, G03, G04, G96, G97, G98, G99, G40, G41, G42 and the system cannot call subprograms(M98/M99);
- G96, G97, G98, G99, G40, G41, G42 are invalid when G71 is executed, and are valid when G70 is executed;
- When G71 is executed, the system can stop the automatic run and manual traverse
- When the system is executing the feed hold or single block, the program pauses after the system has executed end point of current path;
- △d,△u are specified by the same U and different with or without being specified P, Q commands;
- G71 cannot be executed in MDI, otherwise, the system alarms;
- There are no the same block number in ns~nf when compound cycle commands are executed repetitively in one program;
- The tool retraction point should be high or low as possible to avoid crashing the workpiece

. Example：Fig. 3-27（Type I）

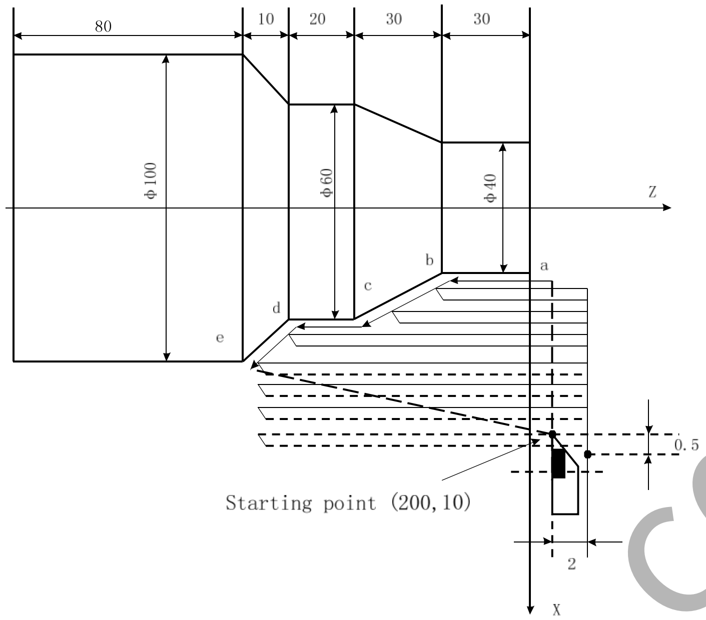


Fig. 3-27

Program: O0004;

G00 X200 Z10 M3 S800; (Spindle clockwise with 800 r/min)

G71 U2 R1 F200; (Cutting depth each time 4mm, tool retraction 2mm [in diameter])

G71 P80 Q120 U0.5 W0.2; (roughing a---e, machining allowance: X, 1mm;Z, 2mm)

N80 G00 X40 S1200; (Positioning)

G01 Z-30 F100 ; (a→b)

X60 W-30; (b→c)

W-20; (c→d)

N120 X100 W-10; (d→e)

a→b→c→d→e blocks for finishing path

G70 P80 Q120; (a---e blocks for finishing path)

M30; (End of block)

3.13.2 Radial roughing cycle G72

Command format：G72 W(Δd) R(e) F__ S__ T__； (1)

G72 P(ns) Q(nf) U(Δu) W(Δw) K0/1 (2)

N__(ns) ．．．．．．；

．．．．．．．．．；

．．．．．F；

．．．．．S；

．．．．．；

．

N__(nf)．．．．．．； (3)

Command function: G72 is divided into three parts:

(1) 1st blocks for defining the travels of tool infeed and tool retraction, the cutting speed, the spindle speed and the tool function in roughing;

(2) 2nd blocks for defining the block interval, finishing allowance;

(3) 3rd blocks for some continuous finishing path, counting the roughing path without being executed actually when G72 is executed.

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 Z, and the roughing is completed by multiple executing the cutting cycle tool infeed→cutting feed→tool retraction. The starting point and the end point of G72 are the same one. The command is applied to the formed roughing of non-formed rod.

Relevant definitions:

Finishing path: the above-mentioned Part (3) of G71(ns～nf block)defines the finishing path, and the starting point of finishing path (i.e. starting point of ns block)is the same these of starting point and end point of G72, called A point; the first block of finishing path(ns block)is used for Z rapid traversing or cutting feed, and the end point of finishing path is called to B point; the end point of finishing path(end point of nf block)is called to C point. The finishing path is A→B→C.

Roughing path: The finishing path is the one after offsetting the finishing allowance(Δu, Δw) and is the path contour formed by executing G72. A, B, C point of finishing path after offset corresponds separately to A', B', C' point of roughing path, and the final continuous cutting path of G72 is B'→C' point.

Δd: it is Z cutting in roughing, its value: 0.001~99.999(unit: mm) without sign symbol, and the direction of tool infeed is determined by ns block traverse direction. the specified value Δd is reserved and the data value is switched to the corresponding value to save to No.051 after W(Δd) is executed. The value of system parameter No.051 is regarded as the tool infeed clearance when R(e) is not input.

e: it is Z tool retraction clearance in roughing, its value: 0~99.999(unit: mm) without sign symbol, and the direction of tool retraction is opposite to that of tool infeed, the specified value e is reserved and the data value is switched to the corresponding value to save to No.052 after R(e) is executed. The value of system parameter No.052 is regarded as the tool retraction clearance when R(e) is not input.

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

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

Δu : it is X finishing allowance in roughing, its range: ±99999999×least input increment(X coordinate offset of roughing contour corresponding to the finishing path, i.e. X absolute coordinate difference between A' and A, in diameter with sign symbol).

Δw : it is Z finishing allowance in roughing, its range: ±99999999×least input increment(Z coordinate offset of roughing contour corresponding to the finishing path, i.e. Z absolute coordinate difference between A' and A, in diameter with sign symbol).

F: Cutting feedrate; S: Spindle speed; T: Tool number, tool offset number.

M, S, T, F: They can be specified in the first G72 or the second ones or program ns ~ nf. M, S, T, F functions of M, S, T, F blocks are invalid in G72, and they are valid in G70 finishing blocks.

Execution process: Fig. 3-28

①X rapidly traverses to A' from A point, X travel is Δu, and Z travel is Δw;

②X moves from from A' is Δd(tool infeed), ns block is for tool infeed at rapid traverse speed with G0, is for tool infeed at G72 feedrate F in G1, and its direction of tool infeed is that of A→B point;

③X executes the cutting feeds to the roughing path, and its direction is the same that of X coordinate B→C point;

④X, Z execute the tool retraction e (45°straight line)at feedrate, the directions of tool retraction is opposite to that of tool infeed ;

⑤X rapidly retracts at rapid traverse speed to the position which is the same that of Z coordinate;

⑥After Z tool infeed (Δd+e)again is executed, the end point of traversing tool is still on the middle point of straight line between A'and B'(the tool does not reach or exceed B'), and after Z executes the tool infeed (Δd+e)again, ③ is executed; after the tool infeed (Δd+e) is executed again, the end point of tool traversing reaches B' point or exceeds the straight line between A'→B' point and Z executes the tool infeed to B' point, and then the next step is executed;

⑦Cutting feed from B' to C' point along the roughing path;

⑧Rapidly traverse to A from C' point and the program jumps to the next clock following nf block after G71 cycle is completed.

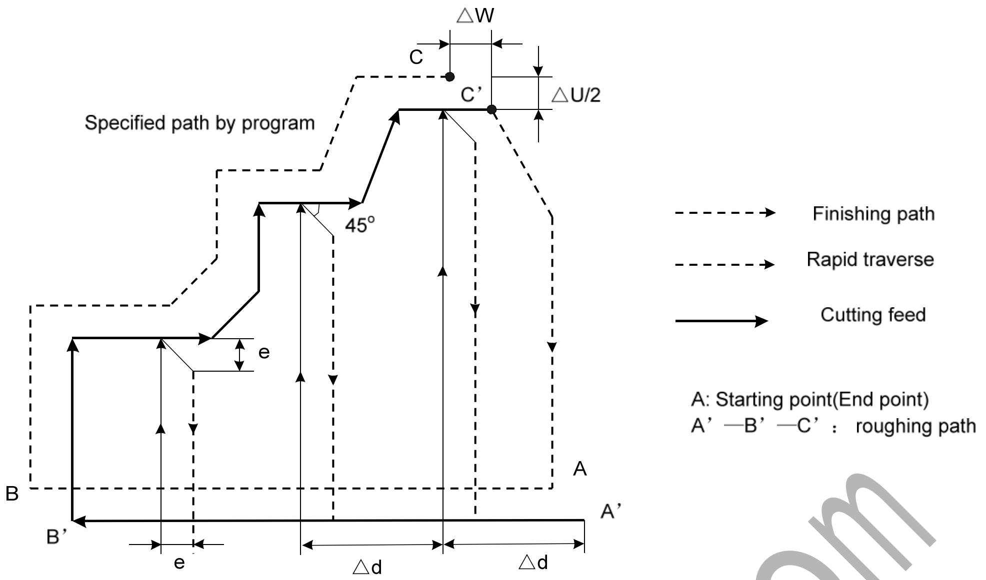


Fig. 3-28

Command specifications:

- ns～nf blocks in programming must be followed G72 blocks.
- ns～nf blocks are used for counting the roughing path and the blocks are not executed when G72 is executed. F, S, T commands of ns～nf blocks are invalid when G72 is executed, at the moment, F, S, T commands of G72 blocks are valid. F, S, T of ns～nf blocks are valid when executing ns～nf to command G70 finishing cycle;
- There are G00, G01 without the word X(U) in ns block, otherwise the system alarms;
- The dimensions in X, Z direction must be changed monotonously (always increasing or reducing) for the finishing path;
- In ns～nf blocks, there are only G commands: G01, G02, G03, G04, G96, G97, G98, G99, G40, G41, G42 and the system cannot call subprograms(M98/M99);
- G96, G97, G98, G99, G40, G41, G42 are invalid when G72 is executed, and are valid when G70 is done;
- When G72 is executed, the system can stop the automatic run and manual traverse
- When the system is executing the feed hold or single block, the program pauses after the system has executed end point of current path;
- △d, △w are specified by the same W and different with or without being specified P, Q commands;
- There are no the same block number in ns~nf when compound cycle commands are executed repetitively in one program;
- G72 cannot be executed in MDI, otherwise, the system alarms;
- The tool retraction point should be high or low as possible to avoid crashing the workpiece.

Coordinate offset direction with finishing allowance:

Δu, Δw define the coordinate offset and its direction of cut-in in finishing, and their sign symbol are as follows Fig. 3-29: B→C for finishing path, B'→C' for roughing path and A is the tool start-up point.

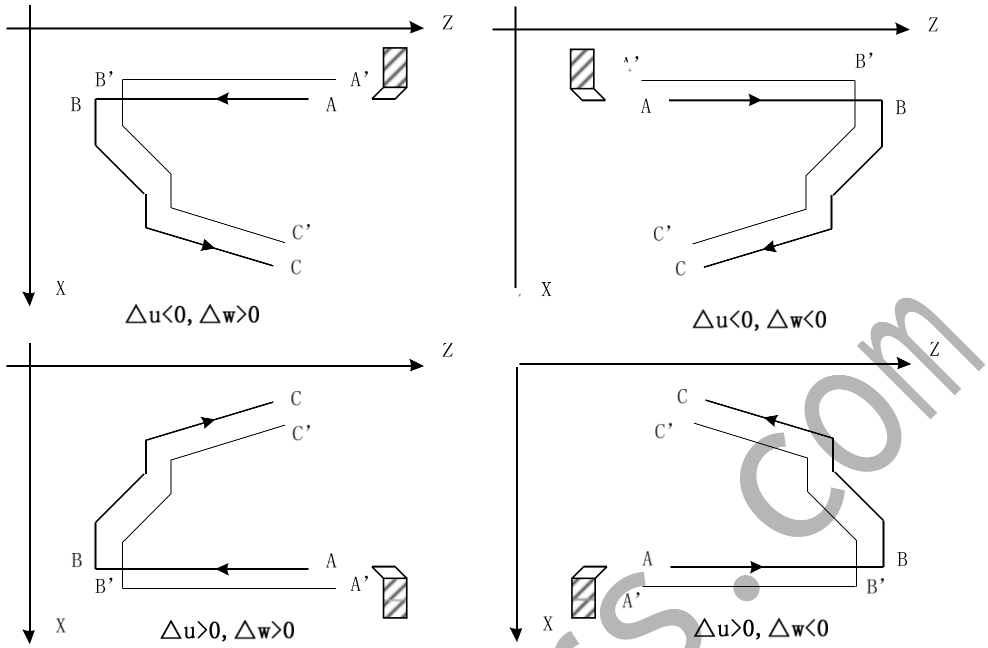


Fig.3-29

Example：Fig. 3-30

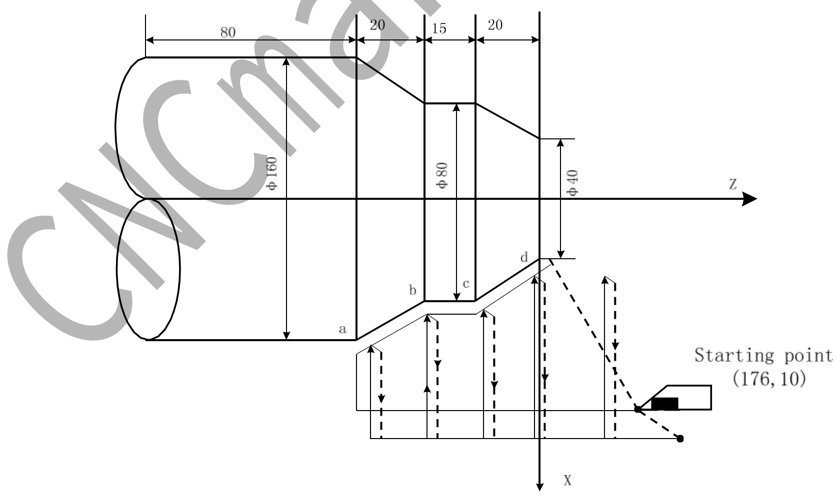


Fig.3-30

Program:

O0005;

G00 X176 Z10 M03 S500 (Change No.2 tool and execute its compensation, spindle CW rotation with 500 r/min)

G72 W2.0 R0.5 F300; (Tool infeed 2mm, tool retraction 0.5mm)

G72 P10 Q20 U0.2 W0.1; (Roughing a--d,X roughing allowance 0.2mm and Z 0.1mm)

```
N10 G00 Z-55 S800 ;     (Rapid traverse)
G01 X160 F120;          (Infeed to a point)
X80 W20;                (Machining a—b)       Blocks for finishing path
W15;                    (Machining b—c)
N20 X40 W20 ;           (Machining c—d)
G70 P050 Q090 M30;      (Finishing a—d)
```

3.13.3 Closed cutting cycle G73

Command format:

```
G73 U(Δi) W (Δk) R (d) F__ S__ T__;          (1)
G73 P(ns) Q(nf) U(Δu) W(Δw) ;                (2)
N__(ns) . . . . . . ;
  . . . . . . . . ;
  . . . . F;
  . . . . S;                                 (3)
  . . . . ;
  .
N__ (nf) . . . . . . ;
```

Command functions: G73 is divided into three parts:

(1) Blocks for defining the travels of tool infeed and tool retraction, the cutting speed, the spindle speed and the tool function when roughing;

(2) Blocks for defining the block interval, finishing allowance;

(3) Blocks for some continuous finishing path, counting the roughing path without being executed actually when executing G73.

According to the finishing allowance, the travel of tool retraction and the cutting times, the system automatically counts the travel of roughing offset, the travel of each tool infeed and the path of roughing, the path of each cutting is the offset travel of finishing path, the cutting path approaches gradually the finishing one, and last cutting path is the finishing one according to the finishing allowance. The starting point and end point of G73 are the same one, and G73 is applied to roughing for the formed rod. G73 is non-modal and its path is shown in Fig.3-77.

Relevant definitions:

Finishing path: The above-mentioned Part 3 of G73 **(**ns～nf block)defines the finishing path, and the starting point of finishing path (start point of ns block)is the same these of starting point and end point of G73, called A point; the end point of the first block of finishing path(ns block)is called B point; the end point of finishing path(end point of nf block) is called C point. The finishing path is A→B→C.

Roughing path: It is one group of offset path of finishing one, and the roughing path times are the same that of cutting. After the coordinates offset, A, B, C of finishing path separately corresponds to A_n, B_n , C_n of roughing path(n is the cutting times, the first cutting path is A_1, B_1, C_1 and the last one is A_d, B_d, C_d). The coordinates offset value of the first cutting compared to finishing path is (Δi×2+Δu, Δw+Δk) (diameter programming) , the coordinates offset value of the last cutting compared to finishing path is(Δu, Δw) , the coordinates offset value of each cutting compared to the previous one is as follows:

Δi: It is X tool retraction clearance in roughing, and its range is ±99999.999× least input increment (radius, with sign symbol) , Δi is equal to X coordinate offset value (radius value) of A1 point compared to Ad point. The X total cutting travel(radius value) is equal to |Δi| in roughing, and X cutting direction is opposite to the sign of Δi: Δi > 0, the system executes X negative cutting in roughing. It is reserved after Δi specified value is executed and the data is switched to the corresponding value to save to NO.053. The No.053 value is regarded as X tool retraction clearance in roughing when U(Δi) is not input.

Δk: It is Z tool retraction clearance in roughing, and its range is ±99999.999× least input increment (radius, with sign symbol) , Δk is equal to Z coordinate offset value (radius value) of A1 point compared to Ad point. Z total cutting travel(radius value) is equal to |Δk| in roughing, and Z cutting direction is opposite to the sign of Δk: Δi > 0, the system executes Z negative cutting in roughing. It is reserved after Δk specified value is executed and the data is switched to the corresponding value to save to NO.054. The No.054 value is regarded as Z tool retraction clearance in roughing when W(Δk) is not input.

d: It is the cutting times 1~9999 (unit: times). R5 means the closed cutting cycle is completed by 5 times cutting. R (d) is reserved after it is executed and NO.055 value is rewritten to d (unit: times). No.055 value is regarded as the cutting times when R(d) is not input. When the cutting times is 1, the system completes the closed cutting cycle based on 2 times cutting.

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

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

Δu: It is X finishing allowance and its range is ±99999.999× least input increment (diameter, with sign symbol) and is the X coordinate offset of roughing path compared to finishing path, i.e. the different value of X absolute coordinates of A_1 compared to A. Δu > 0,it is the offset of the last X positive roughing path compared to finishing path. The system defaults Δu=0 when U(Δu) is not input, i.e. there is no X finishing allowance for roughing cycle.

Δw: It is Z finishing allowance and its range is ±99999.999× least input increment (diameter, with sign symbol) and is the X coordinate offset of roughing path compared to finishing path, i.e. the different value of Z absolute coordinates of A_1 compared to A. Δw > 0,it is the offset of the last X positive roughing path compared to finishing path. The system defaults Δw=0 when W(Δw) is not input, i.e. there is no Z finishing allowance for roughing cycle.

F: Feedrate; S: Spindle speed; T: Tool number, tool offset number.

M, S, T, F: They can be specified in the first G73 or the second ones or program ns ~ nf. M, S, T, F functions of M, S, T, F blocks are invalid in G73, and they are valid in G70 finishing blocks.

Execution process: (Fig. 3-31)

① A→A_1: Rapid traverse;

② First roughing A_1→B_1→C_1 :

A_1→B_1: Rapid traverse speed in ns block in G0, cutting feedrate specified by G73 in ns block in G1;

B_1→C_1: Cutting feed.

③ C_1→A_2: Rapid traverse.

④ Second roughing A_2→B_2→C_2 :

A_2→B_2: Rapid traverse speed in ns block in G0, cutting feedrate specified by G73 in ns block in G1;

$B_2 \rightarrow C_2$: Cutting feed.

⑤ $C_2 \rightarrow A_3$: Rapid traverse:

…………

No. n times roughing, $A_n \rightarrow B_n \rightarrow C_n$:

$A_n \rightarrow B_n$: ns Rapid traverse speed in ns block in G0, cutting feedrate specified by G73 in ns block in G1;

$B_n \rightarrow C_n$: Cutting feed.

$C_n \rightarrow A_{n+1}$: Rapid traverse;

…………

Last roughing, $A_d \rightarrow B_d \rightarrow C_d$:

$A_d \rightarrow B_d$: Rapid traverse speed in ns block in G0, cutting feedrate specified by G73 in ns block in G1;

$B_d \rightarrow C_d$: Cutting feed.

$C_d \rightarrow A$: Rapid traverse to starting point;

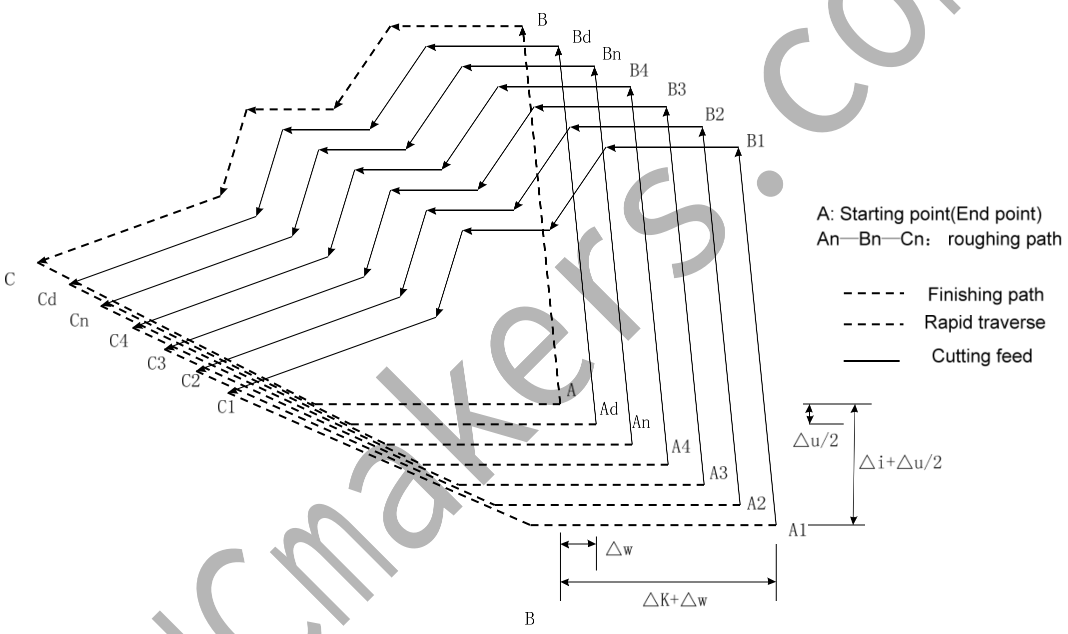


Fig. 3-31 G73 path

Command specifications:

- ns～nf blocks in programming must be followed G73 blocks.
- ns～nf blocks are used for counting the roughing path and the blocks are not executed when G73 is executed. F, S, T commands of ns～nf blocks are invalid when G71 is executed, at the moment, F, S, T commands of G73 blocks are valid. F, S, T of ns～nf blocks are valid when executing ns～nf to command G70 finishing cycle.
- There are only G00, G01 in ns block.
- I n ns～nf blocks, there are only G commands: G00, G01, G02, G03, G04, G96, G97, G98, G99, G40, G41, G42 and the system cannot call subprograms(M98/M99)

- G96, G97, G98, G99, G40, G41, G42 are invalid when G73 is executed, and are valid when G70 is executed.
- When G73 is executed, the system can stop the automatic run and manual traverse
- When the system is executing the feed hold or single block, the program pauses after the system has executed end point of current path.
- △i, △u are specified by the same U and Δk, Δw are specified by the same U, and they are different with or without being specified P,Q commands.
- G73 cannot be executed in MDI, otherwise, the system alarms.
- There are no the same block number in ns~nf when compound cycle commands are executed repetitively in one program.
- The tool retraction point should be high or low as possible to avoid crashing the workpiece.

Coordinate offset direction with finishing allowance:

Δi, Δk define the coordinates offset and its direction of roughing; Δu, Δw define the coordinate offset and the cut-in direction in finishing, and their sign symbols are as follows Fig. 3-32: A is tool start-up point, B→C for workpiece contour, B'→C' for roughing contour and B''→C'' for finishing path.

1）Δi<0 Δk>0， △u<0 Δw>0；

2）Δi>0 Δk>0， △u>0 Δw>0；

3）Δi<0 Δk<0， △u<0 Δw<0；

4）Δi>0 Δk<0， △u>0 Δw<0；

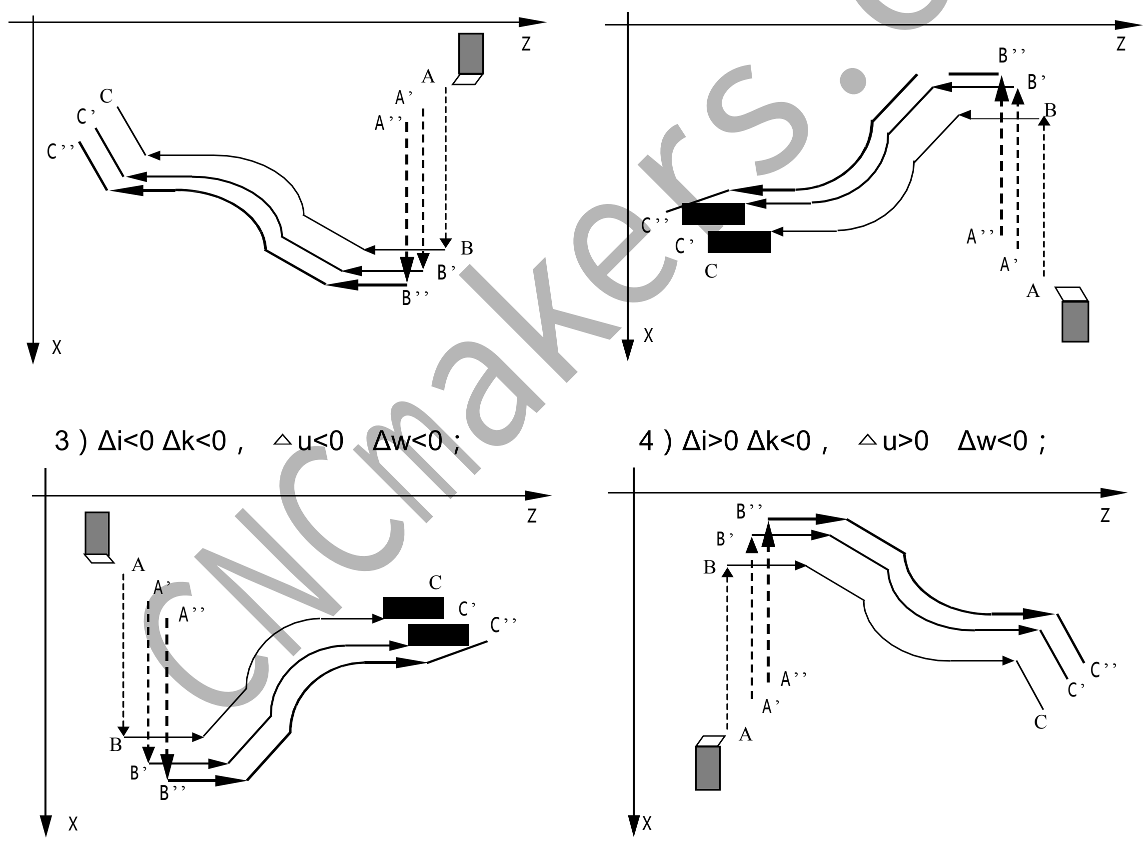


Fig.3-32

Example：Fig. 3-33

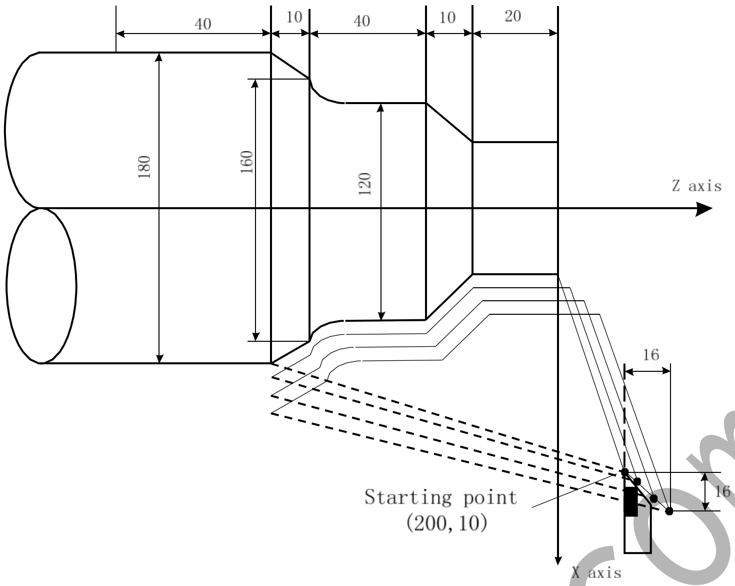


Fig.3-33


```
Program: O0006;
G99 G00 X200 Z10 M03 S500;    (Specify feedrate per rev and position starting point and
                                start spindle)
G73 U1.0 W1.0 R3 ;            (X tool retraction with 2mm, Z 1mm)
G73 P14 Q19 U0.5 W0.3 F0.3 ;  (X roughing with 0.5 allowance and Z 0.3mm)
N14 G00 X80 W-40 ;
G01 W-20 F0.15 S600 ;
X120 W-10 ;
W-20 ;                         Blocks for finishing
G02 X160 W-20 R20 ;
N19 G01 X180 W-10 ;
G70 P14 Q19 M30;              (Finishing)
```

3.13.4 Finishing cycle G70

Command format: G70 P(ns) Q(nf) ;

Command function: The tool executes the finishing of workpiece from starting point along with the finishing path defined by ns～nf blocks. After executing G71, G72 or G73 to roughing, execute G70 to finishing and single cutting of finishing allowance is completed. The tool returns to starting point and execute the next block following G70 block after G70 cycle is completed.

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

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

G70 path is defined by programmed one of ns～nf blocks. Relationships of relative position of ns, nf block in G70～G73 blocks are as follows:

```
 . . . . . . . .
G71/G72/G73 …… ;
N__(ns) . . . . . .        ⎫
 . . . . . . . .           ⎪
       · F                 ⎪
       · S                 ⎬ Blocks for finishing path
       ·                   ⎪
       ·                   ⎪
N__(nf)……                  ⎭
       . . .
G70 P(ns) Q(nf) ;
        . . .
```

Command specifications:

- ns～nf blocks in programming must be followed G70 blocks.
- F, S, T in ns～nf blocks are valid when executing ns～nf to command G70 finishing cycle.
- G96, G97, G98, G99, G40, G41, G42 are valid in G70;
- When G70 is executed, the system can stop the automatic run and manual traverse
- When the system is executing the feed hold or single block, the program pauses after the system has executed end point of current path.
- G70 cannot be executed in MDI, otherwise, the system alarms.
- There are no the same block number in ns~nf when compound cycle commands are executed repetitively in one program.
- The tool retraction point should be high or low as possible to avoid crashing the workpiece.

3.13.5 Axial grooving multiple cycle G74

Command format: G74 R(e) ;

G74 X/U __ Z/W __ P(Δi)_ Q(Δk)_ R(Δd)_ F__;

Command function: Axial (X axis) tool infeed cycle compounds radial discontinuous cutting cycle: Tool infeeds from starting point in radial direction(Z), retracts, infeeds again, and again and again, and last tool retracts in axial direction, and retracts to the Z position in radial direction, which is called one radial cutting cycle; tool infeeds in axial direction and execute the next radial cutting cycle; cut to end point of cutting, and then return to starting point (starting point and end point are the same one in G74), which is called one radial grooving compound cycle. Directions of axial tool infeed and radial tool infeed are defined by relative position between end point X/U Z/W and starting point of cutting. G74 is used for machining radial loop groove or column surface by radial discontinuously cutting, breaking stock and stock removal.

Relevant definitions:

Starting point of axial cutting cycle: starting position of axial tool infeed for each axial cutting cycle, defining with A_n(n=1,2,3……), Z coordinate of A_n is the same that of starting point A, the different value of X coordinate between A_n and A_{n-1} is Δi. The starting point A_1 of the first axial cutting cycle is the same as the starting point A, and the X coordinate of starting point (A_f) of the last axial cutting cycle is the same that of cutting end point.

End point of axial tool infeed: starting position of axial tool infeed for each axial cutting cycle, defining with B_n(n=1,2,3……), Z coordinate of B_n is the same that of cutting end point, X coordinate of B_n is the same that of A_n , and the end point (B_f) of the last axial tool infeed is the same that of cutting end point.

End point of radius tool retraction: end position of radius tool infeed (travel of tool infeed is Δd) after each axial cutting cycle reaches the end point of axial tool infeed, defining with C_n(n=1,2,3……), Z coordinate of C_n is the same that of cutting end point, and the different value of X coordinate between C_n and A_n is Δd;

End point of axial cutting cycle: end position of axial tool retraction from the end point of radius tool retraction, defining with D_n(n=1, 2, 3……), Z coordinate of D_n is the same that of starting point, X coordinate of D_n is the same that of C_n (the different value of X coordinate between it and A_n is Δd);

Cutting end point: it is defined by X/U __ Z/W ____, and is defined with B_f of the last axial tool infeed.

R(e) : it is the tool retraction clearance after each axial(Z) tool infeed, and its range is 0~99.999(unit : mm) without sign symbols. The specified value is reserved validly after R(e) is executed and the data is switched to the corresponding value to save to NO.056. The NO.056 value is regarded as the tool retraction clearance when R(e) is not input.

X: X absolute coordinate value of cutting end point B_f (unit: mm).

U: Different value of X absolute coordinate between cutting end point B_f and starting point.

Z: Z absolute coordinate value of cutting end point B_f (unit: mm).

W: Different value of Z absolute coordinates between cutting end point B_f and starting point.

P(Δi) : radial(X) cutting for each axial cutting cycle , range: 0＜Δi≤ 9999999× least input increment (diameter value), without sign symbol.

Q(Δk) : radial(Z) cutting for each axial cutting cycle, range: 0＜Δk≤ 9999999× least input increment (diameter value), without sign symbol.

R(Δd) : radial (X) tool retraction after cutting to end point of axial cutting,range:0~99999.999× least input increment (diameter value) without sign symbol. The radial (X) tool retraction clearance is 0 when the system defaults the axial cutting end point. The system defaults the tool retraction is executed in positive direction when X/U and P(Δi) are omitted.

Execution process: (Fig. 3-34)

① Axial (Z) cutting feed △k from the starting point of axial cutting cycle, feed in Z negative direction when the coordinates of cutting end point is less than that of starting point in Z direction, otherwise, feed in Z positive direction;

② Axial (Z) rapid tool retraction e and its direction is opposite to the feed direction of ①;

③ X executes the cutting feed (Δk+e) again, the end point of cutting feed is still in it between starting point A_n of axial cutting cycle and end point of axial tool infeed, Z executes the cutting feed (Δk+e)again and execute ②; after Z executing the cutting feed (Δk+e)again, the end point of cutting feed is on B_n or is not on it between A_n and

B_n cutting feed to B_n in Z direction and then execute ④;

④ Radial(X) rapid tool retraction △d(radius value) to C_n , when X coordinate of B_f (cutting end point) is less than that of A (starting point), retract tool in X positive, otherwise, retract tool in X negative direction;

⑤ Axial(Z axial) rapid retract tool to Dn, No. n axial cutting cycle is completed. If the current axial cutting cycle is not the last one, execute ⑥ ; if it is the previous one before the last axial cutting cycle, execute ⑦;

⑥ Radial(X axial)rapid tool infeed, and it direction is opposite to ④ retract tool. If the end point of tool infeed is still on it between A and A_f (starting point of last axial cutting cycle) after X executes the tool infeed (△d+△i) (radius value) , i.e. $D_n \rightarrow A_{n+1}$ and then execute ① (start the next axial cutting cycle); if X end point of tool infeed is not on it between D_n and A_f after tool infeed (△d+△i) (radius value), rapidly traverse to A_f and execute ① to start the first axial cutting cycle;

⑦ X rapidly traverse to return to A, and G74 is completed.

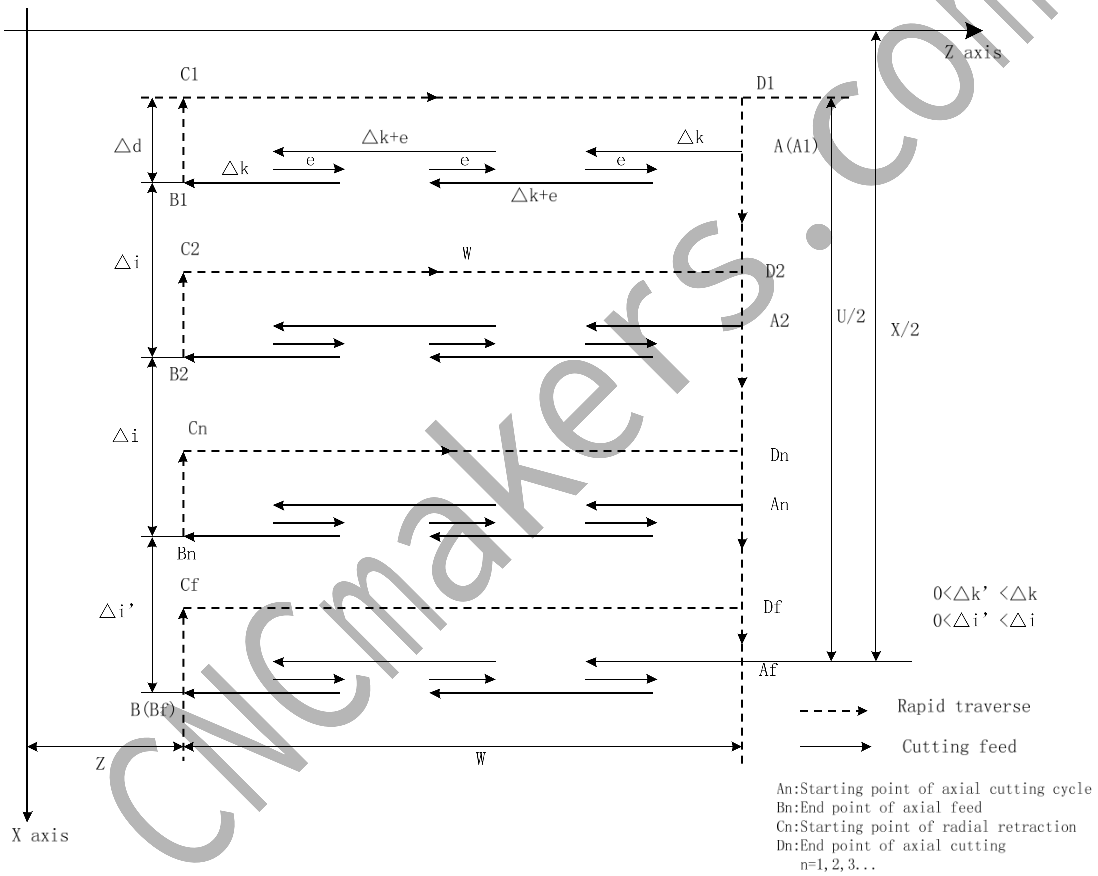


Fig. 3-34 G74 path

Command specifications:

- The cycle movement is executed by Z/W and P(Δk) blocks of G74, and the movement is not executed if only “G74 R(e) ; ” block is executed;
- Δd and e are specified by the same address and whether there are Z/W and P(Δk) word or not in blocks to distinguish them;
- When G74 is executed, the system can stop the automatic run and manual traverse

- When the single block is running, programs dwell after each axial cutting cycle is completed.
- R(Δd) must be omitted in blind hole cutting, and so there is no distance of tool retraction when the tool cuts to axial end point of cutting.

Example：Fig. 3-35

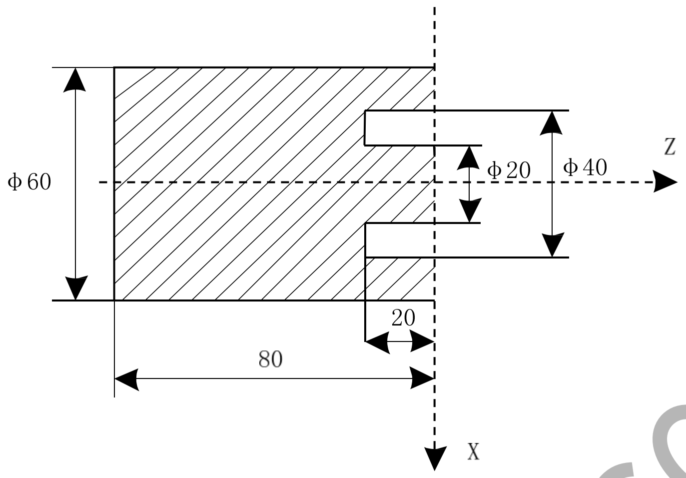


Fig.3-35

Program (suppose that the grooving tool width is 4mm, system least increment is 0.001mm):

O0007;

G0 X40 Z5 M3 S500; (Start spindle and position to starting point of machining)

G74 R0.5 ; (Machining cycle)

G74 X20 Z60 P3000 Q5000 F50; (Z tool infeed 5mm and tool retraction 0.5mm each time; rapid return to starting point (Z5) after cutting feed to end point (Z-20), X tool infeed 3mm and cycle the above-mentioned steps)

M30; (End of program)

3.13.6 Radial grooving multiple cycle G75

Command format：G75 R(e) ;

G75 X/U__ Z/W__ P(Δi) Q(Δk) R(Δd) F__ ;

Command function: Axial (Z) tool infeed cycle compounds radial discontinuous cutting cycle: Tool infeeds from starting point in radial direction, retracts, infeeds again, and again and again, and last tool retracts in axial direction, and retracts to position in radial direction, which is called one radial cutting cycle; tool infeeds in axial direction and execute the next radial cutting cycle; cut to end point of cutting, and then return to starting point (starting point and end point are the same one in G75), which is called one radial grooving compound cycle. Directions of axial tool infeed and radial tool infeed are defined by relative position between end point X(U) Z(W) and starting point of cutting. G75 is used for machining radial loop groove or column surface by radial discontinuously cutting, breaking stock and stock removal.

Relevant definitions:

Starting point of radial cutting cycle: Starting position of axial tool infeed for each radial cutting cycle, defined by A_n(n=1, 2, 3……), X coordinate of A_n is the same that of starting point A, the different value of X coordinate between A_n and A_{n-1} is Δk. The starting point A_1 of the first radial cutting cycle is the same as the starting point A, and Z starting point (A_f) of the last axial cutting cycle is the same that of cutting end point.

End point of radial tool infeed: Starting position of radial tool infeed for each radial cutting cycle, defined by B_n (n=1, 2, 3……), X coordinates of B_n is the same that of cutting end point, Z coordinates of B_n is the same that of A_n, and the end point (B_f) of the last radial tool infeed is the same that of cutting end point.

End point of axial tool retraction: End position of axial tool infeed (travel of tool infeed is Δd) after each axial cutting cycle reaches the end point of axial tool infeed, defining with C_n(n=1,2,3……), X coordinate of C_n is the same that of cutting end point, and the different value of Z coordinate between C_n and A_n is Δd;

End point of radial cutting cycle: End position of radial tool retraction from the end point of axial tool retraction, defined by D_n(n=1, 2, 3……), X coordinate of D_n is the same that of starting point, Z coordinates of D_n is the same that of C_n (the different value of Z coordinate between it and A_n is Δd);

Cutting end point: It is defined by X/U __ Z/W ____ , and is defined with B_f of the last radial tool infeed.

R(e) : It is the tool retraction clearance after each radial(X) tool infeed, its range is 0~99.999 (unit: mm, radius value) without sign symbols. The specified value is reserved validly after R(e) is executed and the data is switched and saved to No.056. NO.056 value is regarded as the tool retraction clearance when R(e) is not input.

X: X absolute coordinate value of cutting end point B_f (unit:mm).

U: Different value of X absolute coordinate between cutting end point B_f and starting point.

Z: Z absolute coordinate value of cutting end point B_f (unit:mm).

W: Different value of Z absolute coordinate between cutting end point B_f and starting point A(unit: mm).

P(Δi) : Radial(X) discontinuous tool infeed of each axial cutting cycle, its range: 0 < Δi≤ 9999999 x least input increment without sign.

Q(Δk) : Axial(Z) discontinuous tool infeed of each radial cutting cycle, its range: 0 < Δk≤ 9999999× least input increment without sign symbol.

R(Δd) : Axial (Z) tool retraction clearance after cutting to end point of radial cutting, its range: 0~99999.999×least input increment without sign symbol.

The system defaults the tool retraction clearance is 0 after the radial cutting end point is completed when R(Δd) is omitted.

The system defaults it executes the positive tool retraction when Z/W and Q(Δk) are omitted.

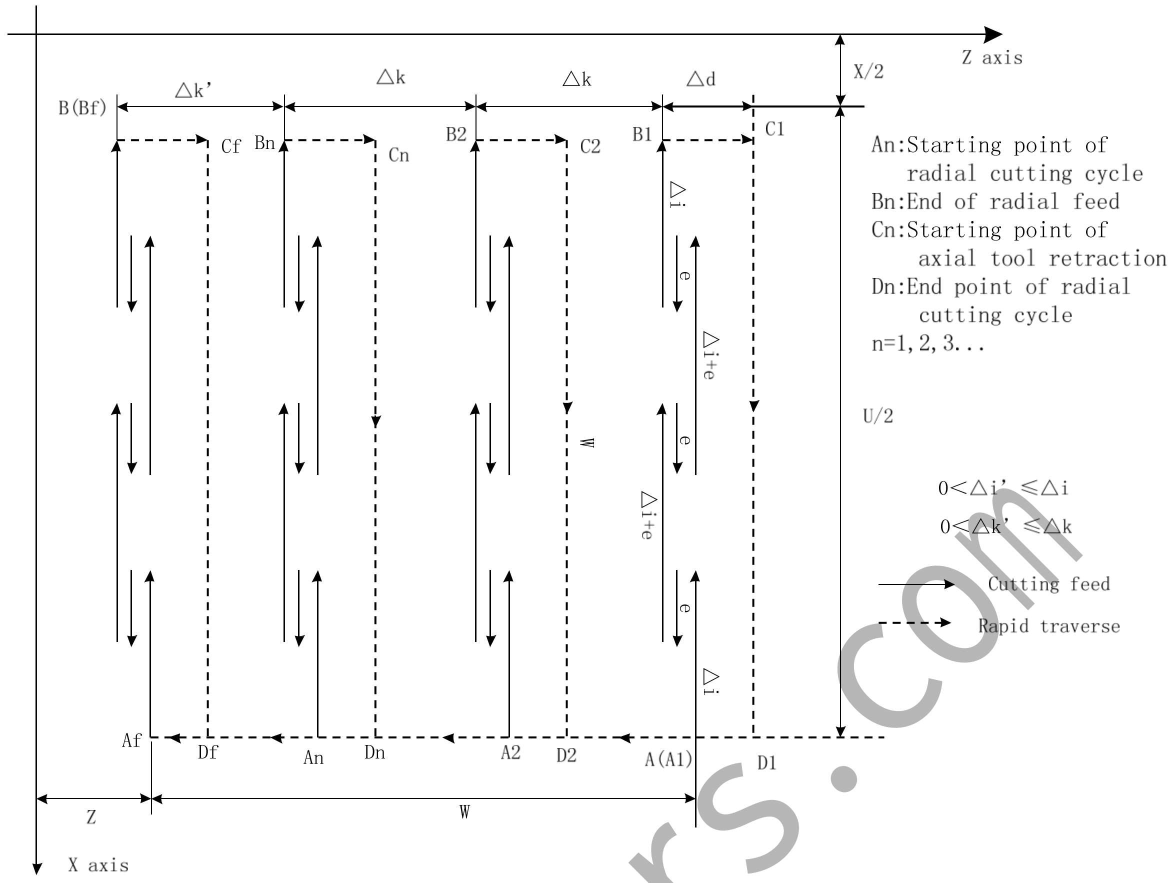


Fig. 3-36 G75 path

Execution process: (Fig. 3-36)

① Radial (X) cutting feed △i from the starting point of radial cutting cycle, feed in X negative direction when the coordinates of cutting end point is less than that of starting point in X direction, otherwise, feed in X positive direction;

② Radial(X) rapid tool retraction e and its direction is opposite to the feed direction of ①;

③ X executes the cutting feed (Δk+e) again, the end point of cutting feed is still in it between starting point A_n of radial cutting cycle and end point of radial tool infeed, X executes the cutting feed (Δi+e) again and executes ②; after X cutting feed (Δi+e) is executed again, the end point of X cutting feed is on B_n or is not on it between A_n and B_n cutting feed to B_n and then execute④;

④ Axial(Z) rapid tool retraction △d(radius value) to C_n , when Z coordinate of B_f (cutting end point) is less than that of A (starting point), retract tool in Z positive, otherwise, retract tool in Z negative direction;

⑤ Radial (Z) rapid retract tool to Dn, No. n radial cutting cycle is completed. The current radial cutting cycle is not the last one, execute ⑥; if it is the previous one before the last radial cutting cycle, execute ⑦;

⑥ Axial(X) rapid tool infeed, and it direction is opposite to ④ retract tool. If the end point of tool infeed is still on it between A and A_f (starting point of last radial cutting cycle) after Z tool infeed (△d+△k) (radius value), i.e. $D_n \rightarrow A_{n+1}$ and then execute ① (start the next radial cutting cycle); if the end point of tool infeed is not on it between D_n and A_f after Z tool infeed (△d+△k), rapidly traverse to A_f and execute ① to start the first radial cutting cycle;

⑦ Z rapidly traverses to A, and G75 is completed.

Explanation:

- The cycle movement is executed by X/W and P(Δi) blocks of G75, and the movement is not executed if only "G75 R(e) ; " block is executed;
- Δd and e are specified by the same address R and whether there are X(U) and P(Δi) words or not in blocks to distinguish them;
- When G75 is executed, the system can stop the automatic run and manual traverse
- When the system is executing the feed hold or single block, the program pauses after the system has executed end point of current path;
- R(Δd) must be omitted in grooving, and so there is no tool retraction clearance when the tool cuts to radial cutting end point.

Example：Fig.3-37

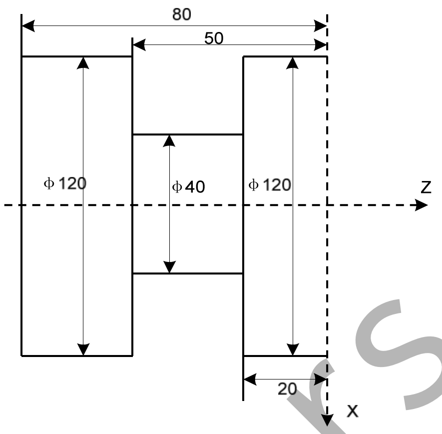


Fig. 3-37 G75 cutting

Program (suppose the grooving tool width is 4mm, the system least increment is 0.001mm):

```
O0008;
G00 X150 Z50 M3 S500;        (Start spindle with 500 r/min)
G0 X125 Z-20;                (Position to starting point of machining)
G75 R0.5 F150;               (Machining cycle)
G75 X40 Z-50 P6000 Q3000;    (X tool infeed 6mm every time, tool retraction 0.5mm,
                             rapid returning to starting point (X125) after infeeding
                             to end point (X40), Z tool infeed 3mm and cycle the
                             above-mentioned steps to continuously run programs)
G0 X150 Z50;                 (Return to starting point of machining)
M30;                         (End of program)
```

3.14 Thread cutting commands

C1000T CNC system can machine many kinds of thread cutting, including metric/inch single, multi threads, thread with variable lead and tapping cycle. Length and angle of thread run-out can be changed, multiple cycle thread is machined by single sided to protect tool and improve smooth finish of its surface. Thread cutting includes: continuous thread cutting G32, thread cutting with variable lead G34, Z thread cutting G33, Thread cutting cycle G92, Multiple thread cutting cycle G76.

The machine used for thread cutting must be installed with spindle encoder whose pulses are set by No.070m. Drive ratio between spindle and encoder is set by No.110 and No.111. X or Z traverses to start machine after the system receives spindle signal per rev in thread cutting, and so one thread is machined by multiple roughing, finishing without changing spindle speed.

The system can machine many kinds of thread cutting, such as thread cutting without tool retraction groove. There is a big error in the thread pitch because there are the acceleration and the deceleration at the starting and ending of X and Z thread cutting, and so there is length of thread lead-in and distance of tool retraction at the actual starting and ending of thread cutting.

X, Z traverse speeds are defined by spindle speed instead of cutting feedrate override in thread cutting when the pitch is defined. The spindle override control is valid in thread cutting. When the spindle speed is changed, there is error in pitch caused by X and Z acceleration/deceleration, and so the spindle speed cannot be changed and the spindle cannot be stopped in thread cutting, which will cause tool and workpiece to be damaged.

3.14.1 Thread cutting with constant lead G32

Command format: G32 X/U_ Z/W_ F(I)_ J_ K_ Q_

Command function: The path of tool traversing is a straight line from starting point to end point as Fig.3-84; the longer moving distance from starting point to end point(X in radius value) is called as the long axis and another is called as the short axis. In course of motion, the long axis traverses one lead when the spindle rotates one revolution, and the short axis and the long axis execute the linear interpolation. Form one spiral grooving with variable lead on the surface of workpiece to realize thread cutting with constant lead. Metric pitch and inch pitch are defined respectively by F, I. Metric or inch straight, taper, end face thread and continuous multi-section thread can by machined in G32.

Command specifications:

G32 is modal;

Pitch is defined to moving distance when the spindle rotates one rev(X in radius);

Execute the straight thread cutting when X coordinates of starting point and end point are the same one(not input X or U);

Execute the end face thread cutting when Z coordinates of starting point and end point are the same one(not input Z or W);

Execute the cutting taper thread when X and Z coordinates of starting point and end point are different;

Related definitions:

F: Metric pitch is moving distance of long axis when the spindle rotates one rev: 1 mm～500 mm. After F is executed, it is valid until F with specified pitch is executed again.

I: Teeth per inch. It is ones per inch(25.4 mm) in long axis, and also is circles of spindle rotation when the long axis traverses one inch(25.4 mm) :0.06tooth/inch～25400tooth/inch. After I is executed, it is valid until I with specified pitch is executed again. The metric, inch input both express the teeth per inch thread.

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

K: Length in the long axis in thread run-out. If the long axis is X, its value is in radius without direction; K is modal parameter.

Q: Initial angle(offset angle)between spindle rotation one rev and starting point of thread cutting: 0～360000 (unit: 0.001 degree). Q is non-modal parameter, must be defined every time, 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. Q unit : 0.001^0 . Q180000 is input in program if it offsets 180^0 with spindle one-turn; if Q180 or Q180.0, it is 0.18°.

Difference between long axis and short axis is shown in Fig. 3-38

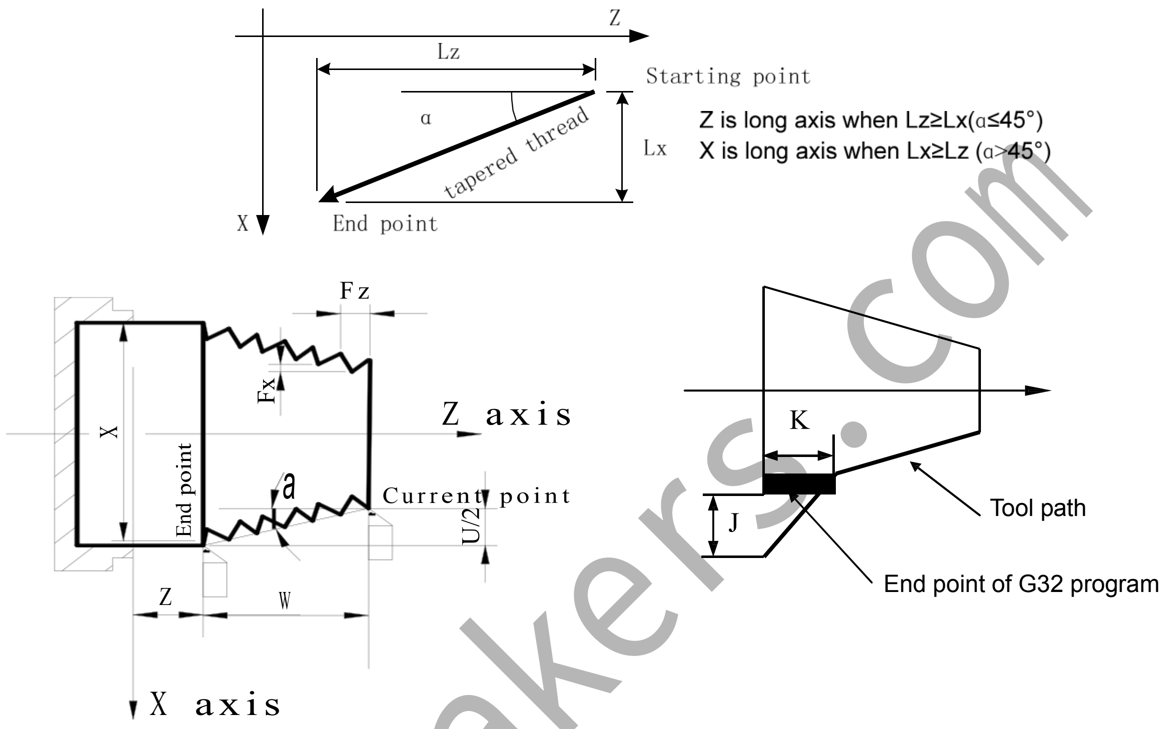


Fig. 3-38 G32 path

Notes:

- There is no thread run-out when J, or J, K are omitted; K=J is the thread run-out value when K is omitted;
- There is no thread run-out when J=0 or J=0, K=0;
- The thread run-out value J=K when J≠0,K=0;
- There is no thread run-out when J=0 or K≠0;
- If the current block is for thread and the next block is the same, the system does not test the spindle encoder signal per rev at starting the next block to execute the direct thread cutting, which function is called as continuous thread machining;
- After the feed hold is executed, the system displays “Pause” and the thread cutting continuously executes not to stop until the current block is executed completely; if the continuous thread cutting is executed, the program run pauses after thread cutting blocks are executed completely;
- In Single block, the program stops run after the current block is executed. The program stops running after all blocks for thread cutting are executed;

- The thread cutting decelerates to stop when the system resets, emergently stop or its drive unit alarms.

Example: Pitch: 2mm. δ1 = 3mm,δ2 = 2mm,total cutting depth 2mm divided into two times cut-in.

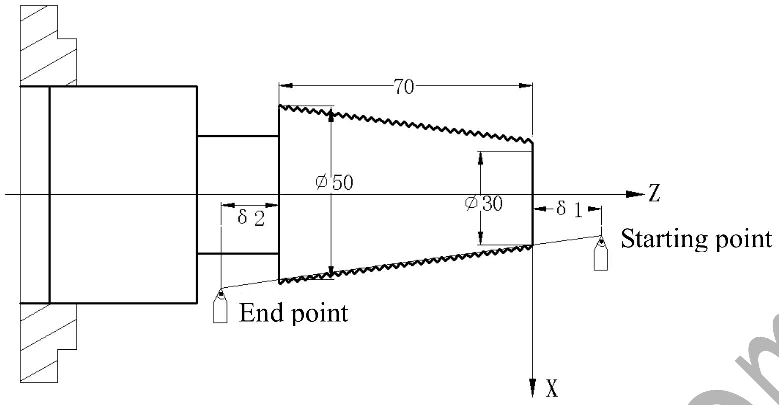


Fig.3-39

Program:

```
O0009;
G00 X28 Z3;           (First cut-in 1mm)
G32 X51 W-75 F2.0;    (First taper cutting)
G00 X55;              (Tool retraction)
W75;                  (Z returns to the starting point)
X27;                  (Second tool infeed 0.5mm)
G32 X50 W-75 F2.0;    (Second taper thread cutting )
G00 X55;              (Tool retraction)
W75 ;                 (Z returns to the starting point)
M30;
```

3.14.2 Thread cutting with variable lead G34

Command format：G34 X/U___ Z/W___ F(I)__ J__ K__ R__ ;

Command function: The motion path of tool is a straight line from starting point of X, Z to end point specified by the block, the longer moving distance from starting point to end point(X in radius value) is called as the long axis and another is called as the short axis. In course of motion, the long axis traverses one lead when the spindle rotates one rev, the pitch increases or decreases a specified value per rev and one spiral grooving with variable lead on the surface of workpiece to realize thread cutting with variable lead. Tool retraction can be set in thread cutting.

F, I are specified separately to metric, inch pitch. Executing G34 can machine metric or inch straight, taper, end face thread with variable pitch.

Command specifications:

G34 is modal;

Functions of X/U , Z/W , J, K are the same that of G32;

F: specifying lead, and its range: 0～500 mm;

I: Inch thread of first pitch from starting point: 0.06 tooth/inch～25400 tooth/inch;

R: Increment or decrement of pitch per rev, R=F1- F2, with direction; F1>F2, pitch decreases when R is negative; F1<F2, pitch increases when R is positive (as Fig. 3-87);

R: ±0.001~±500.000 mm/pitch (metric thread);
±0.060~±25400 tooth/inch (inch thread).

The system alarms when R exceeds the above-mentioned range or the pitch exceeds permissive value or is negative owing to R increases or decreases.

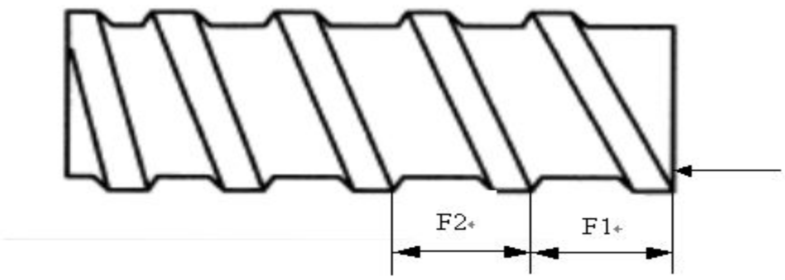


Fig. 3-40 Variable pitch thread

Note: It is the same as that of G32.

Example: First pitch of starting point**:** 4mm, increment 0.2mm per rev of spindle.

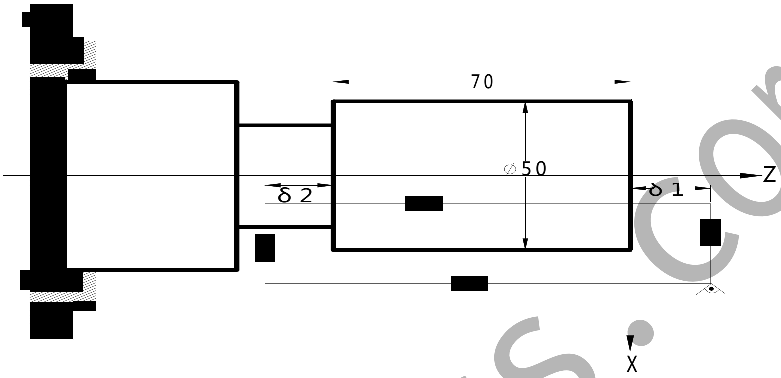


Fig. 3-42 Variable pitch thread machining

Program：O0010；

G00 X60 Z4 M03 S500;

G00 X48；

G34 W-78 F3.8 J5 K2 R0.2;

N30 M30；

3.14.3 Z thread cutting G33

Command format：G33 Z/W __ F(I)__ L__ ;

Command function: Tool path is from starting point to end point and then from end point to starting point. The tool traverses one pitch when the spindle rotates one rev, the pitch is consistent with pitch of tool and there is spiral grooving in internal hole of workpiece and the internal machining can be completed one time.

Command specification: G33 is modal command;

Z/W : When Z or W is not input and starting point and end point of Z axis are the same one, the thread cutting must not be executed;

F: Thread pitch, and its range is referred to Table 1-4;

I: Teeth per inch thread 0.06～25400 teeth/inch; its range is referred to Table 1-4.

L:The number of multi threads.Its range is 1~99. It is single thread when L is omitted.

Cycle process:

① Z tool infeed (start spindle before G33 is executed);

② M05 signal outputs after Z reaches the specified end point in programming;

③ Test spindle after completely stopping;

④ Spindle rotation (CW) signal outputs(reverse to the original rotation direction);

⑤ Z executes the tool retracts to starting point;

⑥ M05 signal outputs and the spindle stops;

⑦ Repeat the steps ①～⑤ if multi threads are machined.

Example: Fig. 3-43 thread M10×1.5

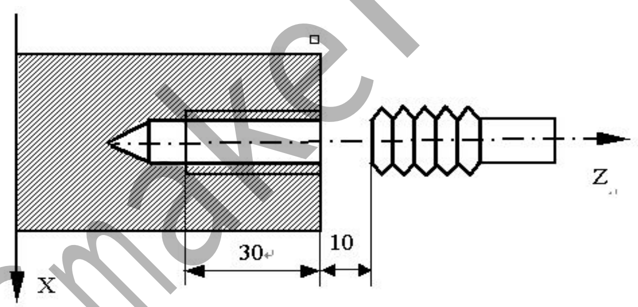


Fig. 3-43

Program:

```
O0011;
G00 Z90 X0 M03;        Start spindle
G33 Z50 F1.5;          Tap cycle
M03                    Start spindle again
G00 X60 Z100;          Machine continuously
M30
```

Note 1: Before tapping, define rotation direction of spindle according to tool rotating. The spindle stops rotation after the tapping is completed and the spindle is started again when machining thread continuously.

Note 2: G33 is for rigid tapping. The spindle decelerates to stop after its stop signal is valid, at the moment, Z executes continuously infeeds along with the spindle rotating, and so the actual cutting bottom hole is deeper than requirement and the length is defined by the spindle speed and its brake in tapping.

Note 3: Z rapid traverse speed in tapping is defined by spindle speed and pitch is not related to cutting feedrate override.

Note 4: In Single block to feed hold, the tapping cycle continuously executes not to stop until the tool returns to starting point when the system displays "Pause".

Note 5: The thread cutting decelerates to stop when the system resets, emergently stop or its driver alarms.

3.14.4 Thread cutting cycle G92

Command format: G92 X/U _ Z/W _ F_ J_ K_ L ; (Metric straight thread cutting cycle)
G92 X/U _ Z/W _ I_ J_ K_ L ; (Inch straight thread cutting cycle)
G92 X/U _ Z/W _ R_ F_ J_ K_ L ; (Metric taper thread cutting cycle)
G92 X/U _ Z/W _ R_ I_ J_ K_ L ; (Metric taper thread cutting cycle)

Command function: Tool infeeds in radial(X) direction and cuts in axial(Z or X, Z) direction from starting point of cutting to realize straight thread, taper thread cutting cycle with constant thread pitch. Thread run-out in G92: at the fixed distance from end point of thread cutting, Z executes thread interpolation and X retracts with exponential or linear acceleration, and X retracts at rapidly traverse speed after Z reaches to end point of cutting as Fig. 3-94.

Command specifications:

G92 is modal;

Starting point of cutting: starting position of thread interpolation;

End point of cutting: end position of thread interpolation;

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

U: different value of X absolute coordinate from end point to starting point of cutting, unit:mm;

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

W: Different value of X absolute coordinate from end point to starting point of cutting, unit:mm;

R: Different value(radius value) of X absolute coordinate from end point to starting point of cutting. When the sign of R is not the same that of U, $|R| \leq |U/2|$, unit:mm;

F: Thread lead, its range: $0 < F \leq 500$ mm. After F value is executed, it is reserved and can be omitted;

I: Thread teeth per inch, its range: 0.06tooth/inch ~ 25400tooth/inch , it is reserved and it can be omitted not to input after I specified value is executed;

J: Movement in the short axis in thread run-out, its range 0~99999.999× least input increment without direction (automatically define its direction according to starting position of program), and it is modal parameter. If the short axis is X, its value is specified by radius;

K: Movement in the long axis in thread run-out, its range: 0~99999.999× least input increment without direction (automatically define its direction according to starting position of program), and it is modal parameter. If the long axis is X, its value is specified by radius;

L: Multi threads: 1 ~ 99 and it is modal parameter. (The system defaults it is single thread when L is omitted).

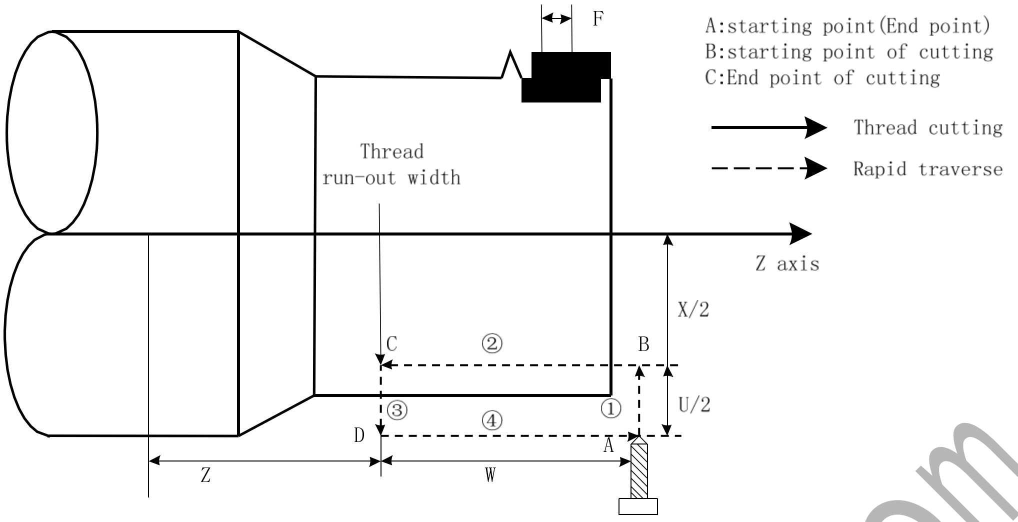


Fig. 3-44

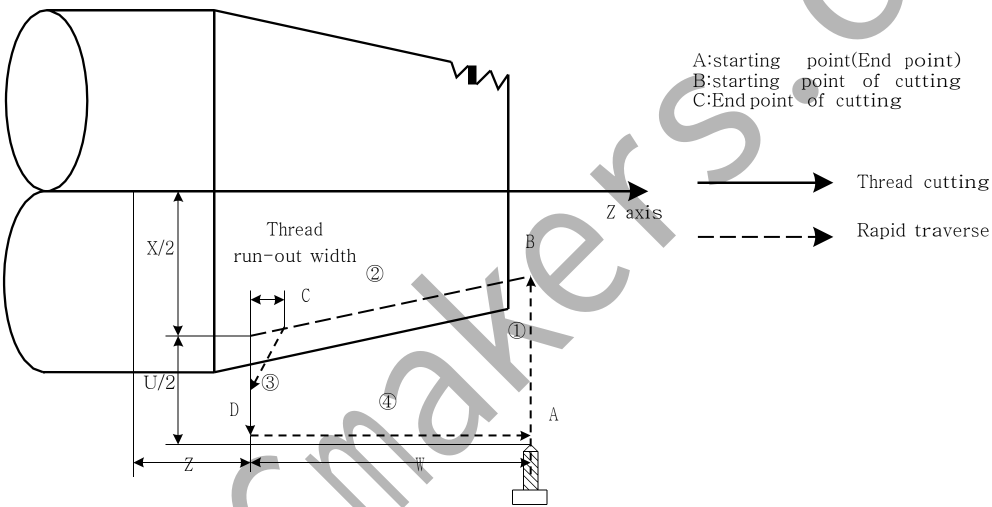


Fig. 3-45

The system can machine one thread with many tool infeed in G92, but cannot do continuous two thread and end face thread. Definition of thread pitch in G92 is the same that of G32, and a pitch is defined that it is a moving distance of long axis(X in radius) when the spindle rotates one rev.

Pitch of taper thread is defined that it is a moving distance of long axis(X in radius). When absolute value of Z coordinate difference between B point and C point is more than that of X (in radius), Z is long axis; and vice versa.

Cycle process: straight thread as Fig.3-44 and taper thread as Fig.3-45.

① X traverses from starting point to cutting starting point;

② Thread interpolates (linear interpolation) from the cutting starting point to cutting end point;

③ X retracts the tool at the cutting feedrate (opposite direction to the above-mentioned ①), and return to the position which X absolute coordinate and the starting point are the same;

④ Z rapidly traverses to return to the starting point and the cycle is completed.

Notes：

- Length of thread run-out is specified by №019 when J, K are omitted;
- Length of thread run-out is K in the long direction and is specified by №019 when J is omitted；
- Length of thread run-out is J=K when K is omitted;
- There is no thread run-out when J=0 or J=0, K=0;
- Length of thread run-out is J=K when J≠0,K=0;
- There is no thread run-out when J=0,K≠0;
- After executing the feed hold in thread cutting, the system does not stop cutting until the thread cutting is completed with ***Pause*** on screen;
- After executing single block in thread cutting, the program run stops after the system returns to starting point(one thread cutting cycle is completed);
- They are executed as the positive values when J, K negative values are input;
- Thread cutting decelerates to stop when the system resets, emergently stops or its driver alarms.

Command path: relative position between thread cutting end point and starting point with U, W, R and tool path and thread run-out direction with different U, W, R signs below:

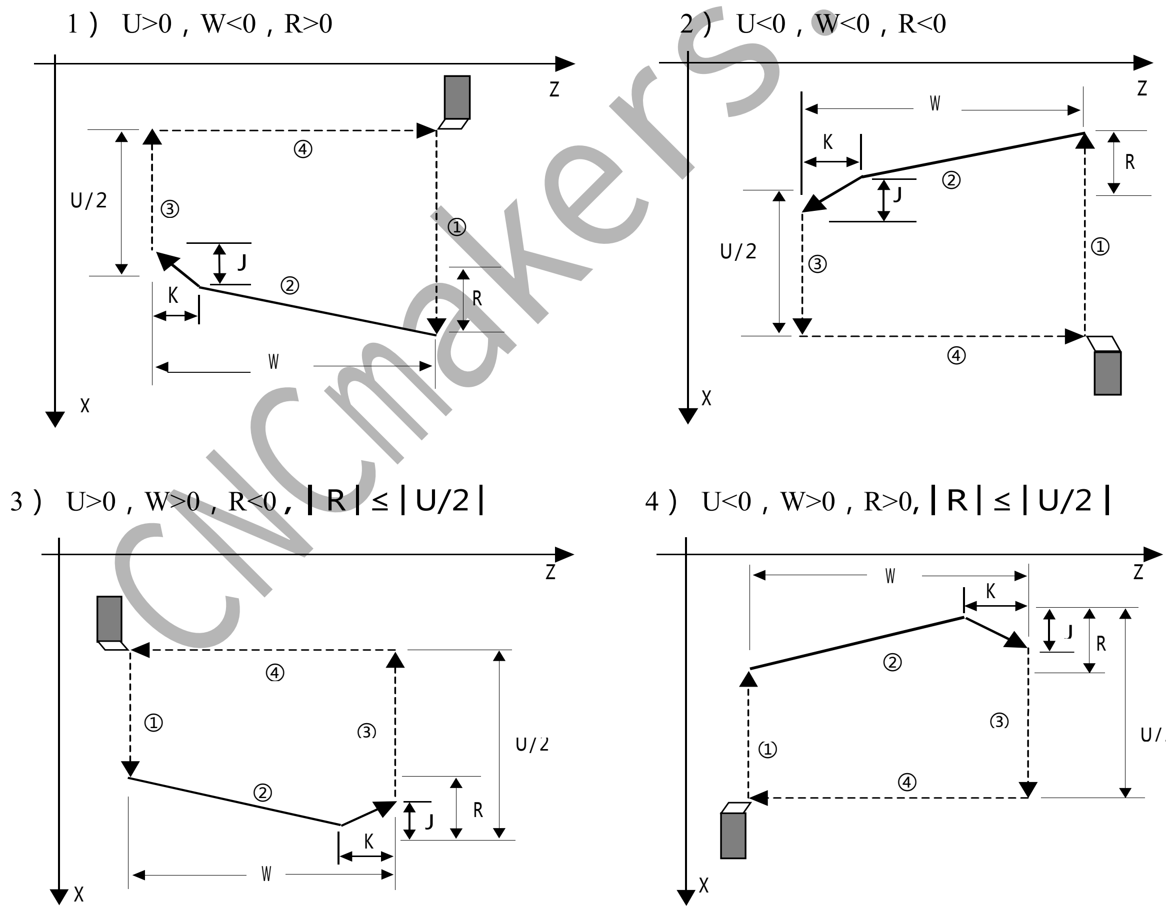


Fig.3-46

Example：Fig.3-47

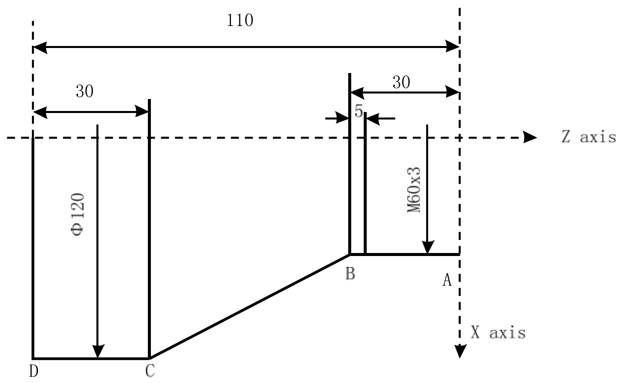


Fig.3-47

Program:

O0012;

M3 S300 G0 X150 Z50 T0101; (Thread tool)

G0 X65 Z5; (Rapid traverse)

G92 X58.7 Z-28 F3 J3 K1; (Machine thread with 4 times cutting, the first tool infeed 1.3mm)

X57.7 ; (The second tool infeed 1mm)

X57; (The third tool infeed 0.7mm)

X56.9; (The fourth tool infeed 0.1mm)

M30;

3.14.5 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) __;

Command function: Machining thread with specified depth of thread (total cutting depth)is completed by multiple roughing and finishing, if the defined angle of thread is not 0°, thread run-in path of roughing is from its top to bottom, and angle of neighboring thread teeth is the defined angle of thread. G76 can be used for machining the straight and taper thread with thread run-out path, which is contributed to thread cutting with single tool edge to reduce the wear of tool and to improve the precision of machining thread. But G76 cannot be used for machining the face thread. machining path is shown in Fig. 3-98:

Relevant definitions:

Starting point(end point): Position before block runs and behind blocks run, defined by A point;

End point of thread(D point): End point of thread cutting defined by X/U ___ Z/W ___.The tool will not reach the point in cutting if there is the thread run-out path;

Starting point of thread: Its absolute coordinates is the same that of A point and the different value of X absolute coordinates between C and D is i(thread taper with radius value). The tool cannot reach C point in cutting when the defined angle of thread is not 0°;

Reference point of thread cutting depth (B point) : Its absolute coordinates is the same that of A point and the different value of X absolute coordinate between B and C is k(thread taper with radius value).The cutting depth of thread at B point is 0 which is the reference point used for counting each thread cutting depth by the system;

Thread cutting depth: It is the cutting depth for each thread cutting cycle. It is the different value (radius value, without signs) of X absolute coordinate between B and intersection of reversal extension line for each thread cutting path and straight line BC. The cutting depth for each roughing is $\sqrt{n}\times\triangle d$, n is the current roughing cycle times, △d is the thread cutting depth of first roughing;

Thread cutting amount: Different value between the current thread current depth and the previous one:$(\sqrt{n}-\sqrt{n-1})\times\triangle d$;

End point of tool retraction: It is the end position of radial (X) tool retraction after the thread cutting in each thread roughing, finishing cycle is completed, defining with E point;

Thread cut-in point: B_n (n is the cutting cycle times) is the actual thread cutting starting point in each thread roughing cycle and finishing cycle, B_1 is the first thread roughing cutting-in point, B_f is the last thread roughing cut-in point, B_e is the thread finishing cutting-in point. B_n is X, Z replacement formula corresponding to B.

$$tg\frac{\alpha}{2}=\frac{Z\ \text{replacement}}{X\ \text{replacement}}$$ a : thread angle ;

X: X absolute coordinate (unit: mm) of thread end point;
U: Different value (unit: mm) of X absolute coordinate between thread end point and starting point;
Z: Z absolute coordinate (unit: mm) of thread end point;
W: Different value (unit: mm) of Z absolute coordinate between thread end point and starting point;
P(m): Times of thread finishing: 00～99 (unit: times). It is valid after m specified value is executed, and the system parameter №057 value is rewritten to m. The value of system parameter №057 is regarded as finishing times when m is not input. In thread finishing, every feed cutting amount is equal to the cutting amount d in thread finishing dividing the finishing times m;
P(r): Width of thread run-out 00～99(unit: 0.1×L,L is the thread pitch). It is valid after r specified value is executed and the system parameter №019 value is rewritten to r. The value of system parameter №019 is the width of thread run-out when r is not input. The thread run-out function can be applied to thread machining without tool retraction groove and the width of thread run-out defined by system parameter №019 is valid for G92, G76;
P(a): Angles at taper of neighboring two tooth，range: 00～99，unit：deg(°). It is valid after a specified value is executed and the system parameter №058 value is rewritten to a. The system parameter №058 value is regarded as angle of thread tooth. The actual angle of thread in defined by tool ones and so a should be the same as the tool angle;
△Q(△dmin): Minimum cutting travel of thread roughing (unit: 0.001mm(IS-B) or 0.0001 mm(IS-C), radius value without sign symbols). When $(\sqrt{n}-\sqrt{n-1})\times\triangle d<\triangle dmin$, △dmin is regarded as the cutting travel of current roughing, i.e. depth of current thread cutting is $(\sqrt{n-1}\times\triangle d+\triangle dmin)$. Setting △dmin is to avoid the too small of roughing amount and too many roughing times caused by the cutting amount deceleration in thread roughing. After Q(△dmin) is executed, the specified value △dmin is valid and the system data parameter NO. 059 value is rewritten to △dmin (unit: 0.001). when Q(△dmin) is not input, the system data parameter NO.059 value is taken as the least cutting amount;
R(d): It is the cutting amount in thread finishing, range: 00～99.999 (unit:mm, radius value without sign symbols), the radius value is equal to X absolute coordinates between cut-in point Be of thread finishing and Bf of thread roughing. After R(d) is executed, the specified value d is reserved and the system parameter №060 value is rewritten to d×1000(unit: 0.001 mm) . The value of system parameter №060 is regarded as the cutting travel of thread finishing when R(d) is not input;

R(i): It is thread taper and is the different value of X absolute coordinate between thread starting point and end point, range: ±99999.999× least input increment (radius value). The system defaults R(i)=0(straight thread) when R(i) is not input;

P(k): Depth of thread tooth, the total cutting depth of thread, range: 1 ~ 99999999× least input increment (radius value without sign symbols). The system alarms when P(k) is not input;

Q(△d): Depth of the 1st thread cutting, range: 1 ~ 99999999× least input increment (radius value without sign symbols). The system alarms when △d is not input;

F: metric thread lead, its range :0~500mm

I:thread teeth per inch for inch thread,its range:0.06~25400teeth/per inc

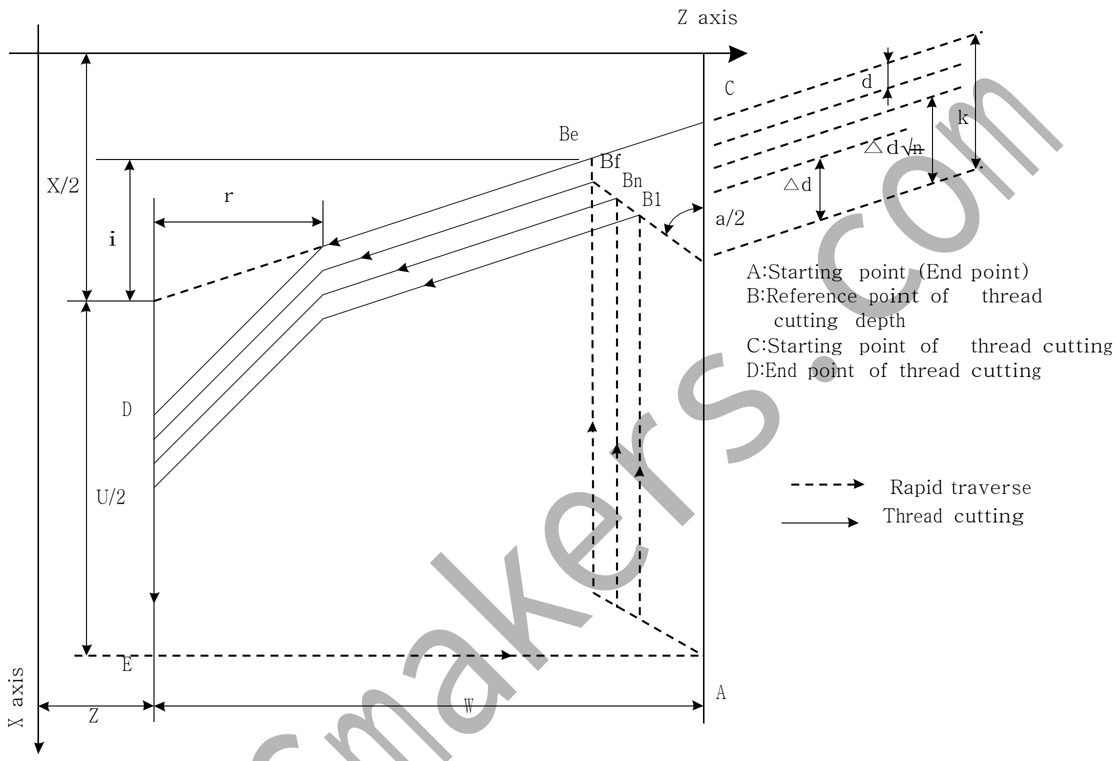


Fig. 3-48(a)

Cut-in method as follows: Fig. 3-48(b)：

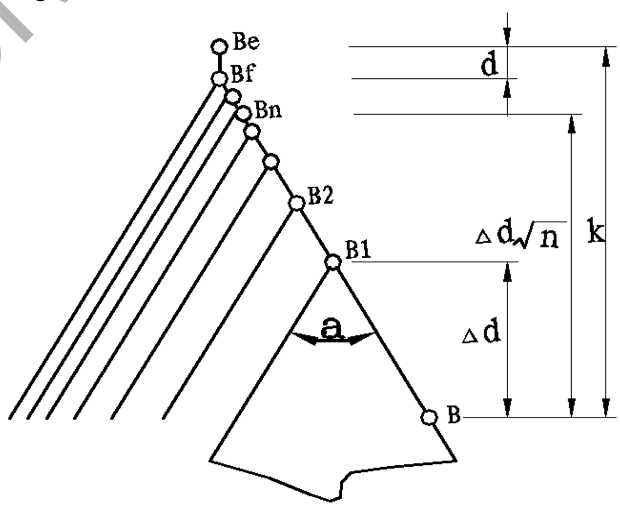


Fig. 3-48(b)

Pitch is defined to moving distance (X radius value) of long axis when the spindle rotates one rev. Z is long when absolute value of coordinate difference between C point and D point in Z direction is more than that of X direction (radius value, be equal to absolute value of i); and vice versa

Execution process:

① The tool rapidly traverses to B_1, and the thread cutting depth is △d. The tool only traverses in X direction when a=0; the tool traverses in X and Z direction and its direction is the same that of A→D when a≠0;

② The tool cuts threads paralleling with C→D to the intersection of D→E (r≠0: thread run-out);

③ The tool rapidly traverses to E point in X direction;

④ The tool rapidly traverses to A point in Z direction and the single roughing cycle is completed;

⑤ The tool rapidly traverses again to tool infeed to B_n (is the roughing times), the cutting depth is the bigger value of ($\sqrt{n}$ ×△d) ,($\sqrt{n-1}$ ×△d+△d_{min}) , and execute ② if the cutting depth is less than(k-d) ; if the cutting depth is more than or equal to(k-d) , the tool infeeds(k-d) to B_f, and then execute ⑥ to complete the last thread roughing;

⑥ The tool cuts threads paralleling with C→D to the intersection of D→E (r≠0: thread run-out);

⑦ X axis rapidly traverses to E point;

⑧ Z axis traverses to A point and the thread roughing cycle is completed to execute the finishing;

⑨ After the tool rapidly traverses to B(the cutting depth is k and the cutting travel is d), execute the thread finishing, at last the tool returns to A point and so the thread finishing cycle is completed;

⑩ If the finishing cycle time is less than m, execute ⑨ to perform the finishing cycle, the thread cutting depth is k and the cutting travel is 0; if the finishing cycle times are equal to m, G76 compound thread machining cycle is completed.

Notes:

- In thread cutting, execute the feed hold, the system displays ***Pause*** after the thread cutting is executed completely, and then the program run pauses;
- Execute single block in thread cutting, the program run stops after returning to starting point(one thread cutting cycle is completed);
- The thread cutting decelerates to stop when the system resets and emergently stop or the driver alarms;
- Omit all or some of G76 P(m) (r) (a) Q(△d_{min}) R(d) . The omitted address runs according to setting value of parameters;
- m, r, a used for one command address P are input one time. Program runs according to setting value of №57, 19, 58 when m, r, a are all omitted; Setting value is a when address P is input with 1 or 2 digits; setting values are r, a when address P is input with 3 or 4 digits;
- The direction of A→C→D→E is defined by signs of U,W , and the direction of C→D is defined by the sign of R(i) . There are four kinds of sign composition of U, W corresponding to four kinds of machining path as Fig. 3-100.

Example: Fig. 3-49, thread M68×6.

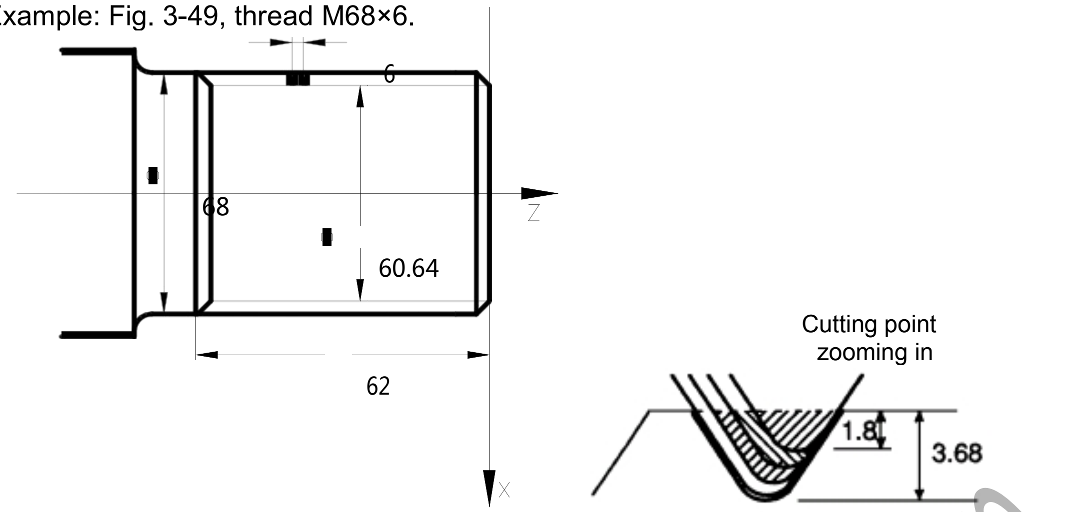


Fig.3-49

Program:

O0013;	
G50 X100 Z50 M3 S300;	(Set workpiece coordinate system, start spindle and specify spindle speed)
G00 X80 Z10;	(Rapid traverse to starting point of machining)
G76 P020560 Q150 R0.1;	(Finishing 2 times, chamfering width 0.5mm, tool angle 60°, min. cutting depth 0.15, finishing allowance 0.1)
G76 X60.64 Z-62 P3680 Q1800 F6;	(Tooth height 3.68, the first cutting depth 1.8)
G00 X100 Z50 ;	(Return to starting point of program)
M30;	(End of program)

3.15 Constant surface speed control G96, constant rotational speed control G97

The detailed is referred to Chapter 2.2.3.

3.16 Feedrate per minute G98, feedrate per rev G99

Command format: G98 F_; (the leading zero can be omitted, feed rate per minute is specified)

Command function: cutting feed rate is specified as mm/min, G98 is the modal G command. G98 cannot be input if the current command is G98 modal.

Command format: G99 F_; (its range is referred to Table 1-4, the leading zero can be omitted)

Command function: Cutting feed rate is specified as mm/min, G99 is the modal G command. G99 input may be omitted if current state is G99. The actual cutting feedrate is gotten by multiplying the F command value (mm/r) to the current spindle speed(r/min). If the spindle speed varies, the actual feedrate changes too. If the spindle cutting feed amount per rev is specified by G99 F__ , the even cutting texture on the surface of workpiece will be gotten. In G99 state, a spindle encoder should be fixed on the machine tool to machine the workpiece.

G98, G99 are the modal G commands in the same group and only one is valid. G98 is the initial state G command and the system defaults G98 is valid when the system turns on.

Reduction formula of feed between per rev and per min:

$F_m = F_r \times S$

F_m: feed per min (mm/min) ;

F_r: feed per rev (mm/r) ; S:

spindle speed (r/min) .

After the system turns on, the feedrate is ones set by №076 and F value is reserved after F is executed. The feed rate is 0 after F0 is executed. F value is reserved when the system resets and emergently stops. The feedrate override is reserved when the system is turned off.

Note: In G99 modal, there is the uneven cutting feed rate when the spindle speed is lower than 1 r/min; there is the follow error in the actual cutting feed rate when there is the swing in the spindle speed. To gain the high machining quality, it is recommended that the selected spindle speed should be not lower than min. speed of spindle servo or converter.

Related parameters:

System parameter No.027: the upper limit value of cutting feedrate(they are the same in X, Z direction, diameter/min in X direction);

System parameter No.029: exponential function for time constant of acceleration/deceleration when cutting feed and manual feed;

System parameter No.030: initial (ultimate) speed of acceleration/deceleration in exponential function when cutting feed and manual feed.

3.17 Macro commands

C1000T provides the macro command which is similar to the high language, and can realize the variable assignment, and subtract operation, logic decision and conditional jump by user macro command, contributed to compiling part program for special workpiece, reduce the fussy counting and simplify the user program.

3.17.1 MACRO variables

- **Presentation of macro variables**

  Present with "#" + macro variables number.;

  Format: # i(i=100,102,103,……) ;

  Example: #105, #109, #125.

- **Variable Type**

  The variable is divided into four types according to the variable number:

Number NO.	Variable type	Function
#0	Null variable	The variable is null and is not valued.
#1~#50	Local variable	The local variable is used to store data in the macro program, such as result. When the system is turned off, the local variable is initialized to be null. When the macro program is called, the argument values to the local.
#100~#199 #500~#999	Share variable	The share variable has the same meaning in the different macro program. When the system is turned off, the variable #100~#199 is initialized to be null, #500~#999 is saved and is not lost.
#1000~#5235	System variable	System variable

- **Macro variables reference**

1. Macro variables can replace command values

Format: ‹ Address › +"# i" or ‹ Address › +" - # I". It shows the system takes variable value or negative value of variable value as address value.

Example: F#103…when #103=15, its function is the same that of F15;

Z-#110…when #110=250, its function is the same that of Z-250;

Note 1: The address O, G and N cannot refer macro variables. For example, O#100 ,G#101 ,N#120 are illegal;

Note 2: If macro variables values exceed the maximum rang of command values, they cannot be used. For example: #130 = 120, M#130 exceeds the maximum command value.

- **Null variable**

When the variable value is not defined, it is null, the variable #0 is always null and only is read instead of writing.

a. Reference

When an undefined variable (null variable) is referred, the address is ignored.

#1=<null>	#1=0
G00 X100 Z#1 is equal to G00 X100	G00 X100 Z#1 is equal to G00 X100 Z0

- **Variable display**

MACRO O0099 N0000

NO.	DATA	NO.	DATA	NO.	DATA
100	123.123	110		120	
101	*********	111	2.001	121	120
102		112		122	
103		113		123	
104	0	114	4.002	124	
105		115		125	
106		116		126	
107		117		127	
108		118	1000	128	
109	1	119		129	

NO. 100

MDI S0000 T0101

(1) In macro window, the variable being displayed to the null means it is null, i.e. it is not defined.

(2) The share variable (#100~#199, #500~#999) values are displayed in the macro variable window, and is also displayed the window, the data is input directly to value the share variable.

(3) The local variable (#1~#50) and the system variable values are not displayed. Some local variable or system variable value is displayed by assigned with the share variable.

● **System variable**

(1)Interface signal: CNC only executes G and F signals. Whether there are I/O to correspond to it is defined by PLC.

Variable No.	Function
#1000~#1015	Correspond G54.0~G54.7, G55.0~G55.7 signal states
#1032	Correspond G54, G55 signal states
#1100~#1115	Correspond F54.0~G54.7, F55.0~F55.7 signal states
#1132	Correspond F54, F55 signal states
#1133	Correspond F56, F57, F58, F59 signal states

(2) Tool compensation system variable:

Compensation No.	Offset compensation value				Wear compensation value			
	X	Z	Y	radius	X	Z	Y	radius
1 … 32	#1500 … #1531	#1600 … #1631	#1700 … #1731	#1800 … #1831	#1900 … #1931	#2000 … #2031	#2100 … #2131	#2200 … #2231

(3) System modal information variable

Variable No.	Function	
#4001	G00, G01, G02, G03, G32, G33, G34, G80, G84, G88, G90, G92, G94	No. 1 group
#4002	G96, G97	No. 2 group
#4003	G98, G99	No. 3 group
#4005	G54,G55,G56,G57,G58,G59	No. 5 group
#4006	G20, G21	No. 6 group
#407	G40,G41,G42	No. 7 group
#4016	G17,G18,G19	No. 16 group
#4120	F command	
#4121	M command	
#4122	Serial No.	
#4123	Program No.	
#4119	S command	
#4120	T command	

(4) system variable of coordinate position information:

Variable No.	Position signal	Coordinate system	Tool compensation value	Read in running
#5001~#5005	End point of block	Workpiece coordinate system	Not including	Possible
#5006~#5010	Current position (Machine coor.)	Machine coordinate system	Including	impossible
#5011~#5015	Current position (Abs. coordinate)	Workpiece coordinate system		

Note: The position listed in the above table separately corresponds orderly to X, Y, Z, 4th, 5th axis. For example: #5001 meanings to be X position information, #5002 meanings to be Y position information, #5003 meanings to be Z position information and #5004 meanings to 4th position information and #5005 meanings to 5th position information.

(5) Workpiece zero offset value and Workpiece coordinate system:
Basic offset value:

G54：#5201 ～ #5205
G54：#5206 ～ #5210
G55：#5211 ～ #5215
G56：#5216 ～ #5220
G57：#5221 ～ #5225
G58：#5226 ～ #5230
G59：#5231 ～ #5235

Local variable

- The relation of adress and local variable:

Variable adress	Local variable	Variable adress	Local variable	Variable adress	Local variable
A	#1	E	#8	U	#21
B	#2	F	#9	V	#22
C	#3	M	#13	W	#23
I	#4	Q	#17	X	#24
J	#5	R	#18	Y	#25
K	#6	S	#19	Z	#26
D	#7	T	#20		

3.17.2 Operation and jump command G65

Command format:

G65 Hm P# i Q# j R# k;

m: operation or jump command

I: macro variables name for storing values.

j: macro variables name 1 for operation, can be constant.

k: macro variables name 2 for operation, can be constant.

Command significance: # i = #j O # k

Operation sign specified by Hm

Example: P#100 Q#101 R#102…..#100 = #101 O #102;

P#100 Q#101 R15….#100 = #101 O 15;

P#100 Q-100 R#102…..#100 = -100 O #102;

Note: Macro variable name has no "#" when it is presented directly with constant.

Macro command list

Command format	Functions	Definitions
G65 H01 P#i Q#j	Assignment	# i = # j assign value of j to i
G65 H02 P#i Q#j R#k;	Decimal add operation	# i = # j + # k
G65 H03 P#i Q#j R#k;	Decimal subtract operation	# i = # j - # k
G65 H04 P#i Q#j R#k;	Decimal multiplication operation	# i = # j×# k
G65 H05 P#i Q#j R#k;	Decimal division operation	# i = # j÷# k
G65 H11 P#i Q#j R#k;	Binary addition	# i = # j OR # k
G65 H12 P#i Q#j R#k;	Binary multiplication(operation)	# i = # j AND # k
G65 H13 P#i Q#j R#k;	Binary exclusive or	# i = # j XOR # k
G65 H21 P#i Q#j;	Decimal square root	# i = $\sqrt{\# j}$
G65 H22 P#i Q#j;	Decimal absolute value	# i = |# j |
G65 H23 P#i Q#j R#k;	Decimal remainder	Remainder of # i = (#j÷# k)
G65 H24 P#i Q#j;	Decimal into binary	# i = BIN(# j)
G65 H25 P#i Q#j;	Binary into decimal	# i = DEC(# j)
G65 H26 P#i Q#j R#k;	Decimal multiplication/division operation	# i = # i×# j÷# k
G65 H27 P#i Q#j R#k;	Compound square root	# i = $\sqrt{\# j^2 + \# k^2}$
G65 H31 P#i Q#j R#k;	Sine	# i = # j×sin(# k)
G65 H32 P#i Q#j R#k;	Cosine	# i = # j×cos(# k)
G65 H33 P#i Q#j R#k;	Tangent	# i = # j×tan(# k)
G65 H34 P#i Q#j R#k;	Arc tangent	# i = ATAN(# j / # k)
G65 H80 Pn;	Unconditional jump	Jump to block n
G65 H81 Pn Q#j R#k;	Conditional jump 1	Jump to block n if # j = # k,otherwise the system executes in order
G65 H82 Pn Q#j R#k;	Conditional jump 2	Jump to block n if # j ≠ # k, otherwise the system executes in order
G65 H83 Pn Q#j R#k;	Conditional jump 3	Jump to block n if # j ＞ # k, otherwise the system executes in order

Command format	Functions	Definitions
G65 H84 Pn Q#j R#k;	Conditional jump 4	Jump to block n if # j < # k, otherwise the system executes in order
G65 H85 Pn Q#j R#k;	Conditional jump 5	Jump to block n if # j ≥ # k, otherwise the system executes in order
G65 H86 Pn Q#j R#k;	Conditional jump 6	Jump to block n if # j ≤# k, otherwise the system executes in order
G65 H99 Pn;	P/S alarm	(500+n) alarms

1 Operation commands

1) Assignment of macro variables: # I = # J

G65 H01 P#I Q#J

(Example) G65 H01 P# 101 Q1005; (#101 = 1005)

G65 H01 P#101 Q#110; (#101 = #110)

G65 H01 P#101 Q-#102; (#101 = -#102)

2) Decimal add operation: # I = # J+# K

G65 H02 P#I Q#J R#K

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

3) Decimal subtract 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) Decimal 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) Decimal 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) Binary 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) Binary logic multiply(and) : # I = # J.AND. # K

G65 H12 P#I Q#J R#K

(Example) G65 H12 P# 201 Q#102 R#103; (#101 = #102.AND.#103)

8) Binary executive 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) Decimal square root: # I = $\sqrt{\#J}$

G65 H21 P#I Q#J

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

10) Decimal absolute value: # I = | # J |

G65 H22 P#I Q#J

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

11) Decimal remainder: # I = # J - TRUNC(#J/#K)×# K,TRUNC: omit decimal fraction

G65 H23 P#I Q#J R#K

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

12) Decimal converting into binary: # I = BIN (# J)

G65 H24 P#I Q#J

(Example) G65 H24 P#101 Q#102 ; (#101 = BIN(#102))

13) Binary converting into decimal: # I = BCD (# J)

G65 H25 P#I Q#J

(Example) G65 H25 P#101 Q#102 ; (#101 = BCD(#102))

14) Decimal multiplication/division operation: # I =(# I×# J) ÷# K

G65 H26 P#I Q#J R# k

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

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

G65 H27 P#I Q#J R#K

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

16) 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))

17) Cosine: # I = # J•COS(# K) (Unit: ‰)

G65 H32 P#I Q#J R# k

(Example) G65 H32 P#1Q#102 R#103; (#101 =#102•COS(#103))

18) Tangent: # I = # J•TAM(# K) (Unit: ‰)

G65 H33 P#I Q#J R# K

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

19) Cosine: # 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))

2 Jump commands

1) Unconditional jump

G65 H80 Pn; n: Block number

(Example) G65 H80 P120; (jump to N120)

2) Conditional jump 1 #J.EQ.# K (=)

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

(Example) G65 H81 P1000 Q#101 R#102;

The program jumps N1000 when # 101= #102 and executes in order when #101 ≠#102.

3) Conditional jump 2 #J.NE.# K (≠)

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

(Example) G65 H82 P1000 Q#101 R#102;

The program jumps N1000 when # 101 ≠ #102 and executes in order when #101 = #102.

4) Conditional jump 3 #J.GT.# K (>)

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

(Example) G65 H83 P1000 Q#101 R#102;

The program jumps N1000 when # 101 > #202 and executes in order when #101 ≤ #102.

5) Conditional jump 4 #J.LT.# K (< =)

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

(Example) G65 H84 P1000 Q#101 R#102;

The program jumps N1000 when # 101 < #102 and executes in order when #101≥#102.

6) Conditional jump 5 #J.GE.# K (≥)

G65 H85 Pn Q#J R# K; n: Block number

(Example) G65 H85 P1000 Q#101 R#102;

The program jumps N1000 when # 101 ≤ #1 and executes in order when #101 < #102.

7) Conditional jump 6 #J.LE.# K (≤)

G65 H86 Pn Q#J R# K; n: Block number

(Example) G65 H86 P1000 Q#101 R#102;

8) P/S alarm

G65 H99 Pi; i: alarm number +500

(Example) G65 H99 P15;

P/S alarm 515.

Note: Block number can be specified by variables. Such as: G65 H81 P#100 Q#101 R#102; The program jumps to block that its block number is specified by #100.

3.17.3 Program example with macro command

Differences between user macro program call (G65, G66) and subprogram call (M98) are as follows:

1. G65 can specify the argument data and send them to macro program and M98 has no such function.
2. G65 can change the level of local variable and M98 has no such function.
3. G65 only follows N and only P or H follows them.

z **Non-modal call (G65)**

Command format: G65 P_ L_ <argument>_;

Macro program specified by P is called, the argument(data) is send to the user macro program body.

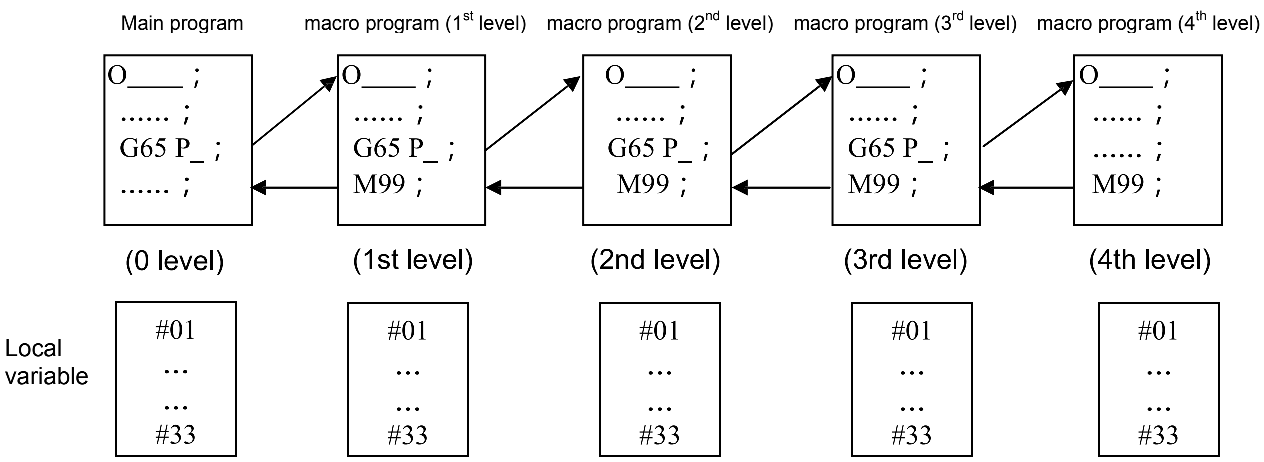
Command explanation:

P —— called macro program number

L —— called times (it is 1 when it is omitted, it can be the repetitive times from 1 to 9999)

<argument> —— data sent to macro program is valued with the corresponding local variable.

Nest call: G65 call has four-level nest.



Specifying argument:

use the letter besides G, L, O, N, P, and each is only specified one time, and the last which is specified many times is valid.

Argument address and corresponding variable No. table in method 1

Address	Variable No.		Address	Variable No.		Address	Variable No.
A	#1		I	#4		T	#20
B	#2		J	#5		U	#21
C	#3		K	#6		V	#22
D	#7		M	#13		W	#23
E	#8		Q	#17		X	#24
F	#9		R	#18		Y	#25
H	#11		S	#19		Z	#26

Note: The addresses which are not needed to specify can be omitted, the corresponding local variable of the omitted address is valued by <null>.

3.18 Metric/Inch Switch

3.18.1 Functional summary

CNC input, output separately has two kinds of unit: metric unit , mm; inch unit, inch.

The corresponding state parameter related to metric, inch in C1000T CNC: No001＃0(INI)： input incremental unit selection

0：metric input (G21)　　　　1：inch input (G20)

The parameter completely corresponds to G20/G21. i.e. the parameter changes along when G20/G21 is being executed; G20/G21 modal correspondingly changes when the parameter is changed.

No003＃0(OIM)：when the metric/inch input mode is switched, whether the tool compensation value and wear value is automatically switched:

0: do not automatically switch(only move one-bit decimal point)

1：automatically switch

No004＃0(SCW)：metric machine, inch machine selection (least output increment selection)

0：metric machine output (0.001mm)

1：inch machine output (0.0001inch)

3.18.2 Function command G20/G21

Command format: G20; (inch input)

G21; (mm input)

G command must be in the beginning of the program, and is specified by the single block.

Warning: must not switch G20/G21 in program being executed; the system is turned on again after G20/G21 is executed.

3.18.3 Notes

(1) **No.001＃0(INI) input increment unit change**

①. After the input increment unit is changed (inch/metric input), the following unit system is changed: (i.e.: mm<>inch; mm/min<>inch/min):

- F specifies the feedrate (mm/min<>inch/min), thread lead (mm <>inch)

—position command (mm<>inch)

—tool compensation value (mm<>inch)

—MPG graduation unit (mm<>inch)

—movement distance in incremental feed (mm<>inch)

—some data parameters, including NO.45~NO.48, NO.56, NO.59, NO.60, NO.114~ NO.116, NO.120~ NO.131, NO.139, No.140, No.154; the unit is 0.001mm(IS-B) in the metric input system, is 0.0001inch(IS-B) in the inch input system. For example, the same parameter NO.45 setting value is 100m, it means to be 0.1mm in the metric input system (G21), and it means 0.1inch in the inch input system (G20).

②. The machine coordinates will automatically switch after the input increment unit change is switched:

(2) **No.004＃0(SCW) output command unit change**

SCW=0: the system minimal command increment uses the metric output (0.001mm)

SCW=1: the system minimal command increment uses the inch output (0.0001inch)

Some data parameter meanings will be changed when the output control bit parameter SCW is changed:

①. Speed parameter:

Metric machine: mm/min

Inch machine: 0.1 inch/min

Example: when the speed is set to 3800, the metric machine is 3800 mm/min and the inch machine is 380 inch/min.

Speed parameters: No.22, No.23, No.27, No.28~No.31, No.32, No.33, No.41, No.107, No.113,No.134;

②. Position(length) parameter

metric machine: 0.001 mm

inch machine: 0.0001 inch

When the setting is 100, the metric machine is 0.1mm and the inch machine is 0.01 inch.

Position parameters: No.34, No.35, No.37~No.40, No.45~No.48, No.102~No.104,No.136~No.138 and all pitch error compensation parameter;

Note 1: When the minimal input increment unit and the minimal command unit are different, the maximal error is the half of minimal command unit. The error cannot be accumulated.

Note 2: The current system increment is IS-B in the above explanation.

CHAPTER 4 TOOL NOSE RADIUS COMPENSATION (G41, G42)

4.1 Application

4.1.1 Overview

Part program is compiled generally for one point of tool according to a workpiece contour. The point is generally regarded as the tool nose A point in an imaginary state (there is no imaginary tool nose point in fact and the tool nose radius can be omitted when using the imaginary tool nose point to program) or as the center point of tool nose arc (as Fig. 4-1). Its nose of turning tool is not the imaginary point but one arc owing to the processing and other requirement in the practical machining. There is an error between the actual cutting point and the desired cutting point, which will cause the over- or under-cutting affecting the part precision. So a tool nose radius compensation is needed in machining to improve the part precision.

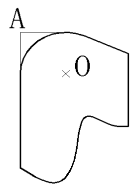


Fig. 4-1 Tool

B tool compensation is defined that a workpiece contour path is offset one tool nose radius, which cause there is excessive cutting at an intersection of two programs because of executing motion path of next after completing the previous block.

To avoid the above-mentioned ones, the system uses C tool compensation method (namely, tool nose radius compensation). The system will read the next block instead of executing it immediately after reading a block in C tool compensation method, and count corresponding motion path according to intersection of blocks. Contour can be compensated precisely because reading two blocks are pretreated as Fig.4-2.

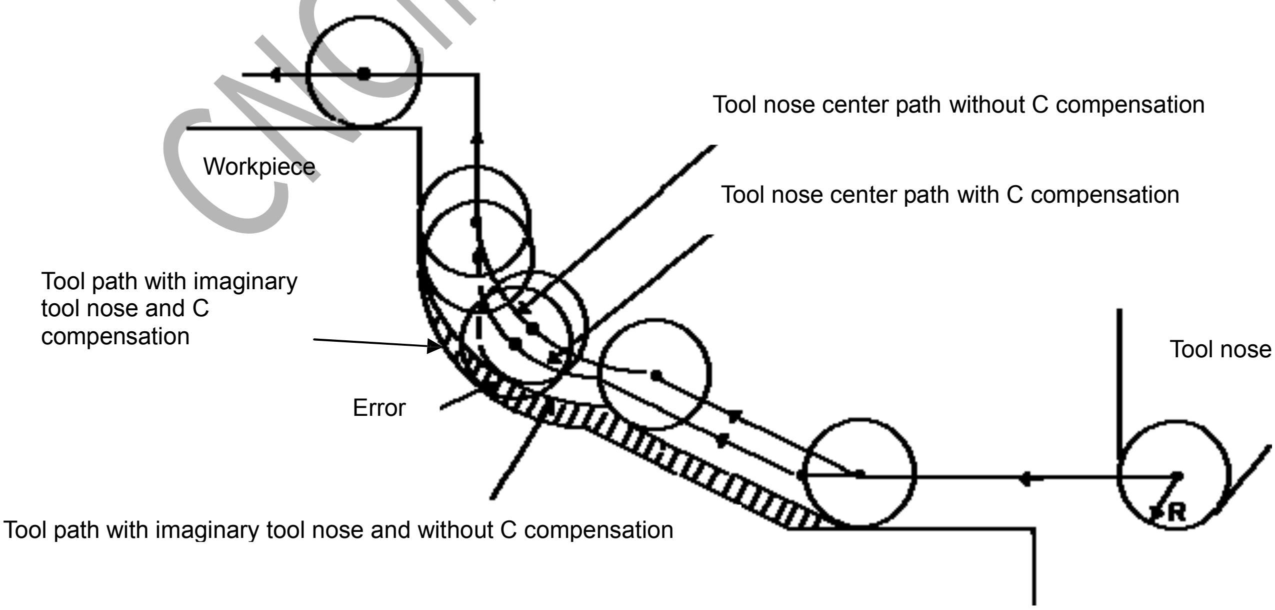


Fig. 4-2 Tool nose center path

4.1.2 Imaginary tool nose direction

Suppose that it is generally difficult to set the tool nose radius center on the initial position as Fig. 4-3; suppose that it is easily set the tool nose on it as Fig. 4-4; The tool nose radius can be omitted in programming. Fig. 4-5 and Fig.4-6 correspond separately to the tool paths of tool nose center programming and imaginary tool nose programming when tool nose radius is executed or not.

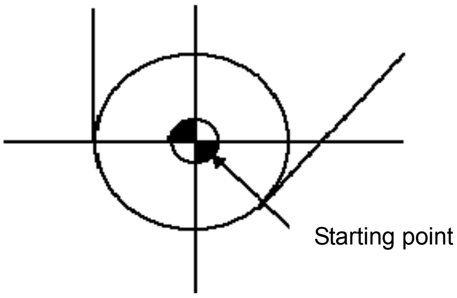


Fig. 4-3 Programming with tool nose

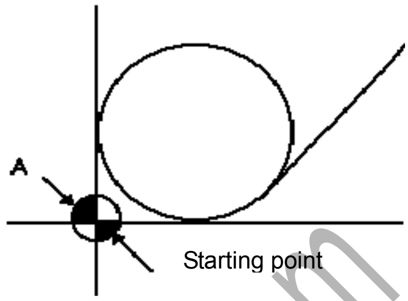
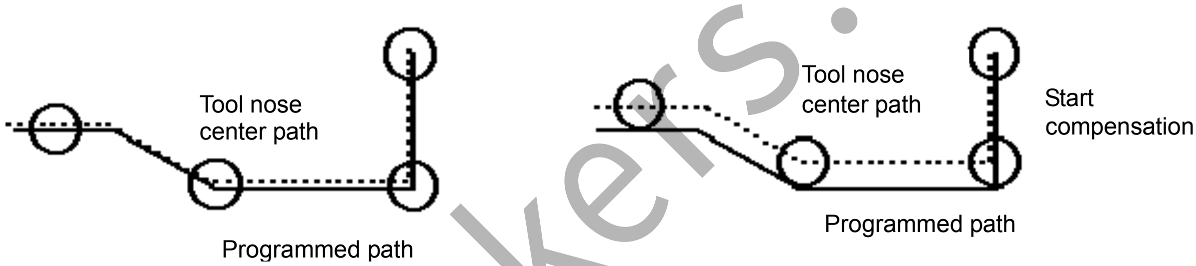


Fig. 4-4 Programming with imaginary tool nose

Tool nose path is the same as programming path without using tool nose radius compensation

Finishing when using tool nose radius compensation



Tool nose path is the same as programming path without using tool nose radius compensation

Finishing when using tool nose radius compensation

Fig. 4-5 Tool path in tool nose center programming

Fig. 4-6 Tool path in imaginary tool nose programming

The tool is supposed to one point in programming but the actual cutting blade is not one ideal point owing to machining technology. Because the cutting blade is not one point but one circular, machining error is caused which can be deleted by tool nose circular radius compensation. In actual machining, suppose that there are different position relationship between tool nose point and tool nose circular center point, and so it must create correct its direction of imaginary tool nose.

From tool nose center to imaginary tool nose, set imaginary tool nose numbers according to tool direction in cutting. Suppose there are 10 kinds of tool nose setting and 9 directions for position relationship. The tool nose directions are different in different coordinate system (rear tool post coordinate system and front tool post coordinate system) even if they are the same tool nose direction numbers as the following figures. In figures, it represents relationships between tool nose and starting point, and end point of arrowhead is the imaginary tool nose; T1～T8 in rear tool post coordinate system is as Fig. 4-7; T1～T8 in front tool post coordinate system is as Fig. 4-8. The tool nose center and starting point for T0 and T9 are shown in Fig. 4-9.

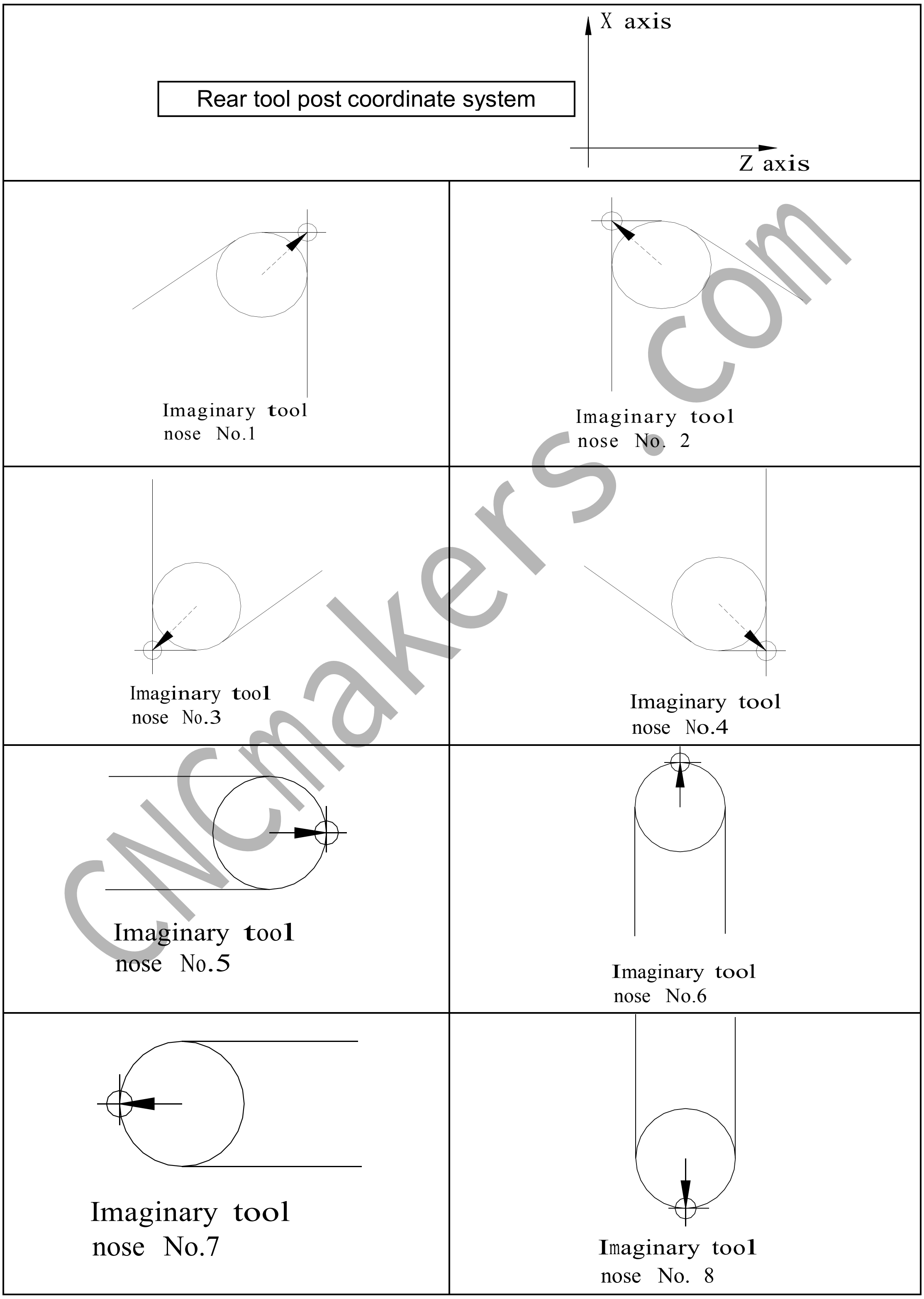


Fig. 4-7 Imaginary tool nose number in rear tool post coordinate system

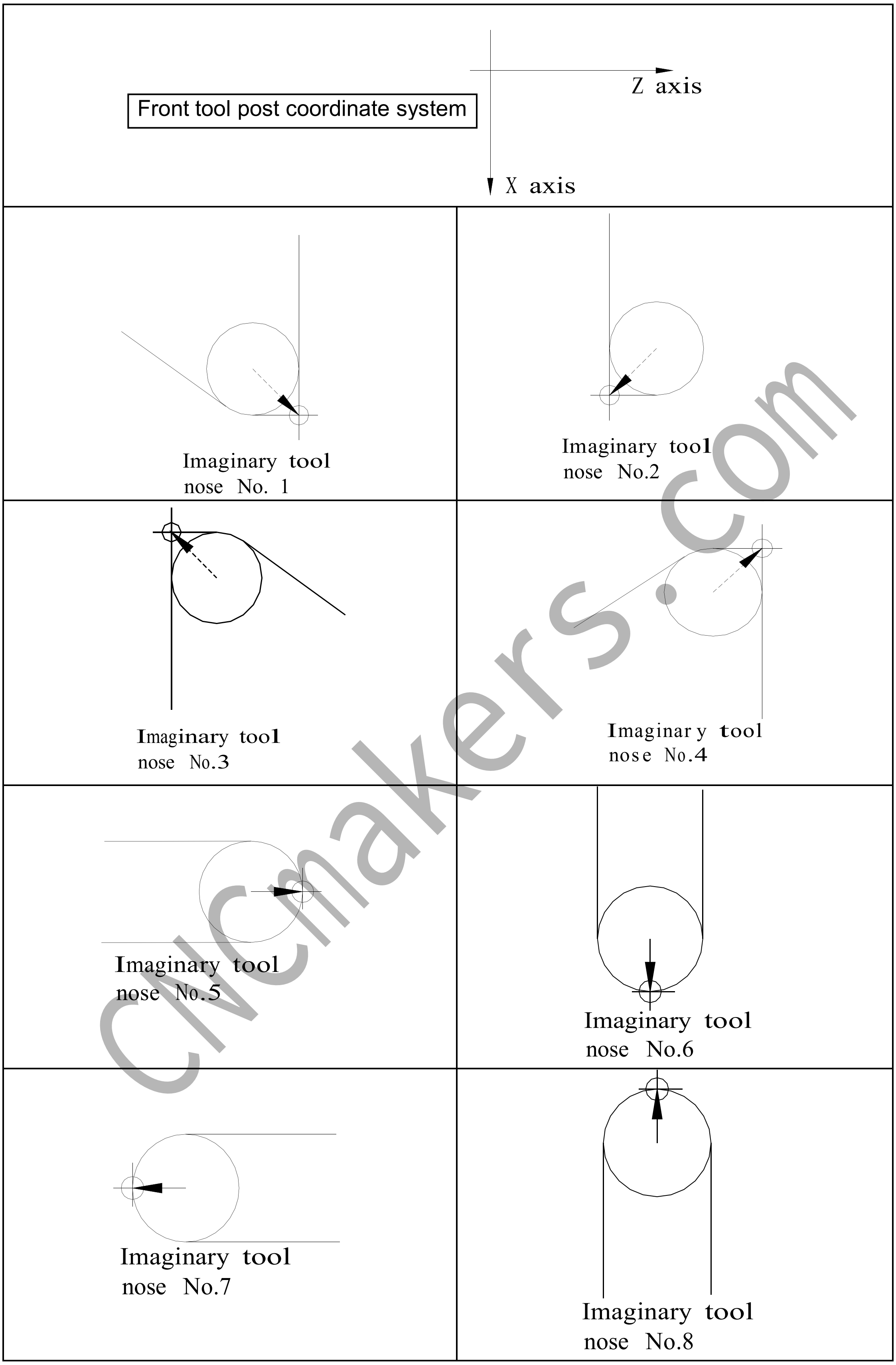


Fig. 4-8 Imaginary tool nose number in front tool post coordinate system

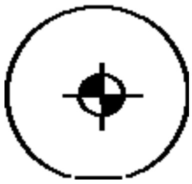


Fig. 4-9 Tool nose center on starting point

4.1.3 Compensation value setting

Preset imaginary tool nose number and tool nose radius value for each tool before executing tool nose radius compensation. Set the tool nose radius compensation value in ***OFFSET*** window (as Fig. 4-1), R is tool nose radius compensation value and T is imaginary tool nose number.

Table 4-1 CNC tool nose radius compensation value display window

number	X	Z	R	T
000	0.000	0.000	0.000	0
001	0.020	0.030	0.020	2
002	1.020	20.123	0.180	3
…	…	…	…	…
032	0.050	0.038	0.300	6

Note: X tool offset value can be specified in diameter or radius, set by No.004 Bit4 ORC, offset value is in radius when ORC=1 and is in diameter when ORC=0.

In toolsetting, the tool nose is also imaginary tool nose point of Tn (n=0~9) when taking Tn(n=0~9) as imaginary tool nose. For the same tool, offset value from standard point to tool nose radius center (imaginary tool nose is T3) is different with that of ones from standard point to imaginary tool nose(imaginary tool nose is T3) when T0 and T3 tool nose points are selected to toolsetting in rear tool post coordinate system, taking tool post center as standard point. It is easier to measure distances from the standard point to the tool nose radius center than from the standard point to the imaginary tool nose, and so set the tool offset value by measuring distance from the standard point to the imaginary tool nose(tool nose direction of T3).

Volume Ⅰ Programming

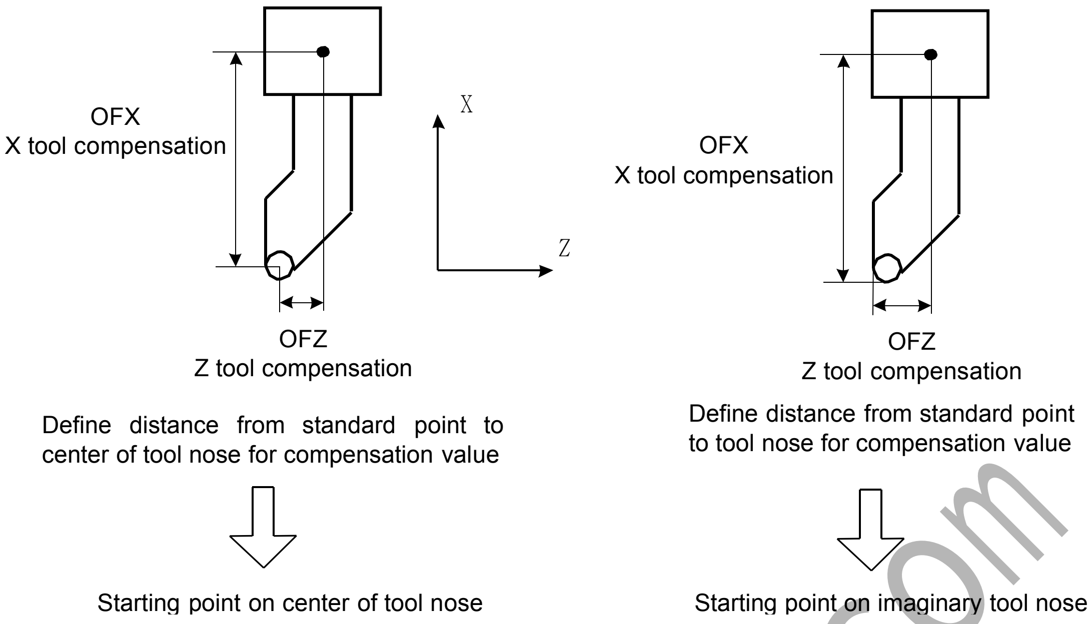


Fig. 4-10 Tool offset value of tool post center as benchmark

4.1.4 Command format

$$\begin{Bmatrix} G40 \\ G41 \\ G42 \end{Bmatrix} \begin{Bmatrix} G00 \\ G01 \end{Bmatrix} \quad X\_\_ \quad Z\_\_ \quad T\_\_ \quad ;$$

Commands	Function specifications	Remark
G40	Cancel the tool nose radius compensation	
G41	Tool nose radius left compensation is specified by G41 in rear tool post coordinate system and tool nose radius right compensation is specified by G41 in front tool post coordinate system	See Fig.4-11 and 4-12
G42	Tool nose radius right compensation is specified by G42 in rear tool post coordinate system and tool nose radius left compensation is specified by G42 in front tool post coordinate system	

4.1.5 Compensation direction

Specify its direction according to relative position between tool nose and workpiece when executing tool nose radius compensation is shown in Fig. 4-11 and Fig.4-12.

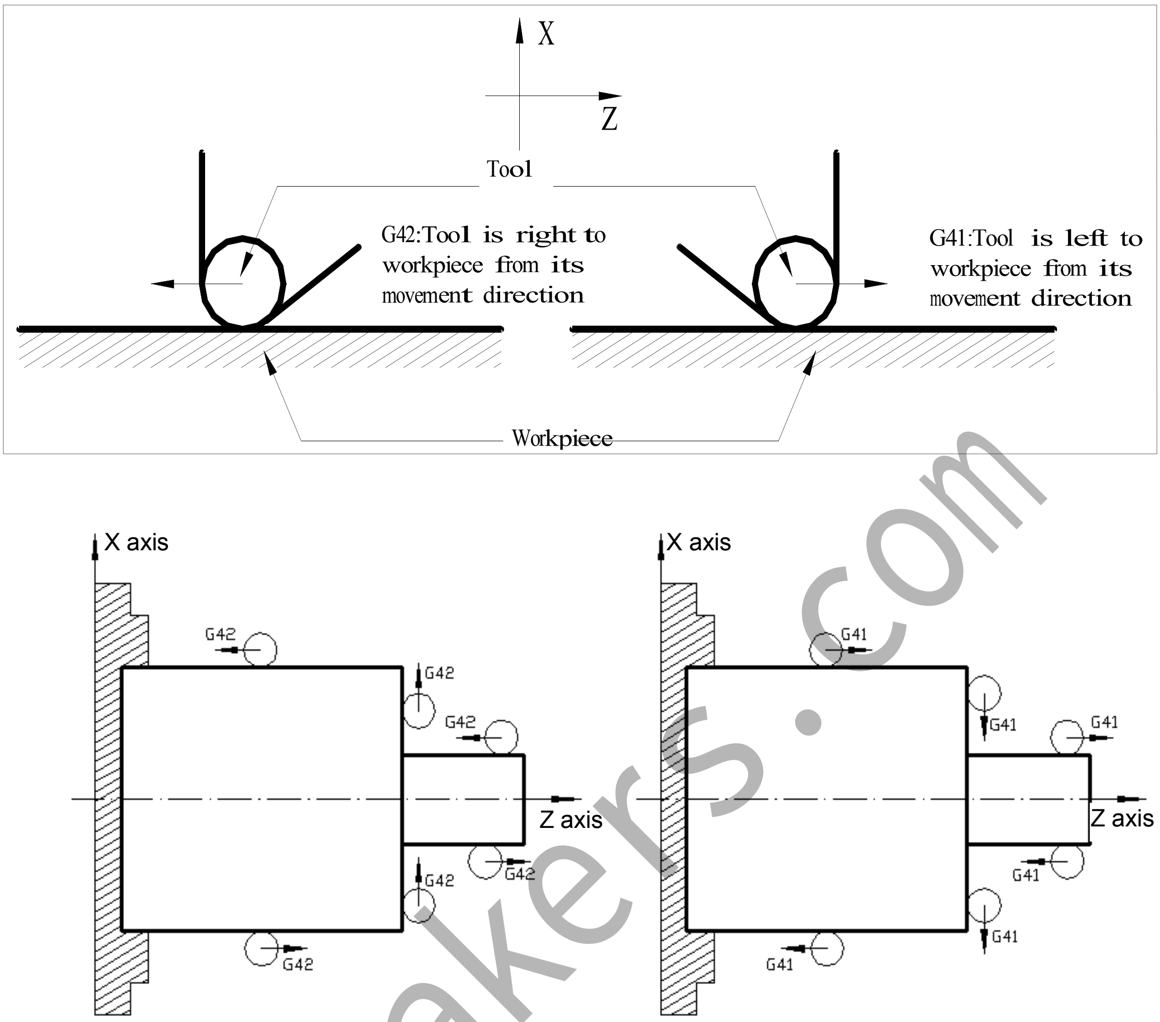
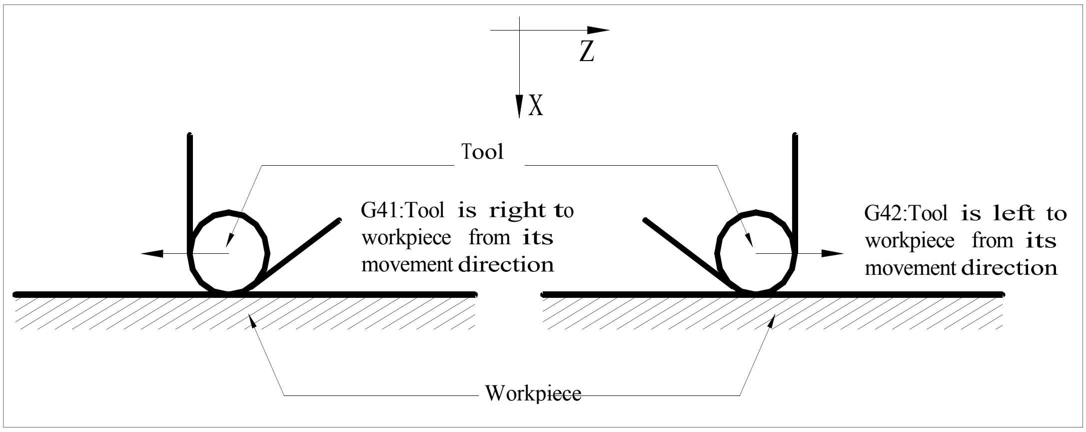


Fig. 4-11 Compensation direction of rear coordinate system



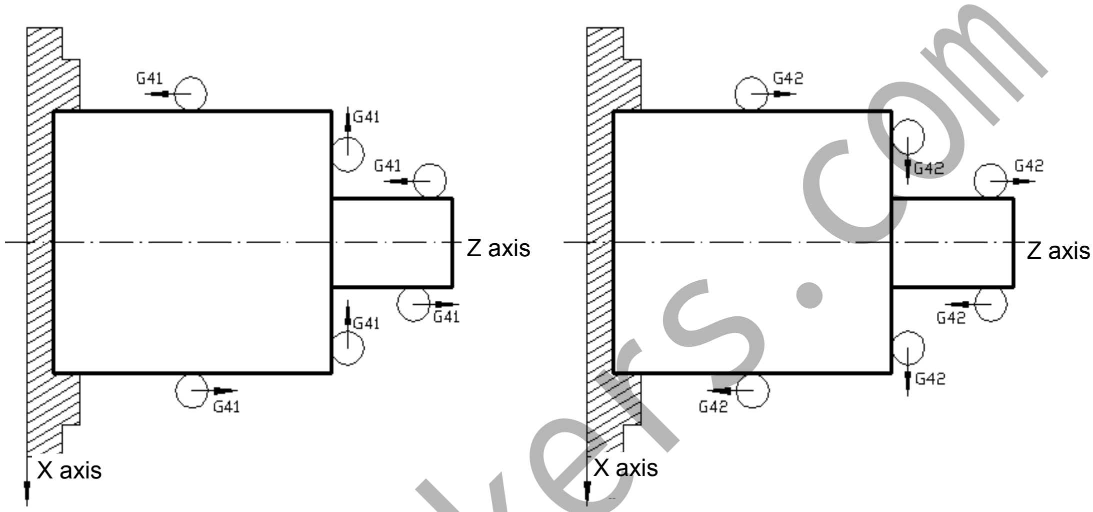


Fig. 4-12 Compensation direction of front coordinate system

4.1.6 Notes

- The system is in tool nose radius compensation mode at initial state, and starts to create tool nose radius compensation offset mode when executing G41 or G42. When the system starts to execute compensation, it pre-read two blocks, and the next block is saved to storage for tool nose radius compensation when executing one of them. The system reads two blocks in ***"Single"*** mode and stops after executing end point of the first block.
- In tool nose radius compensation mode, the tool nose center moves to end point of previous block and is vertical to its path when the system executes two block or more than blocks without motion Command.
- The system cannot create and cancel tool nose radius compensation.
- Tool nose radius R is without negative value, otherwise there is a mistake running path.
- Tool nose radius compensation is created and cancelled in G00 or G01 instead of G02 or G03, otherwise, the system alarms.
- The system cancels the tool nose radius compensation mode when pressing RESET key.
- G40 must be specified to cancel offset mode before the program is ended, otherwise the tool path offsets one tool nose radius.
- The system executes the tool nose radius compensation in main program and subprogram but

must cancel it before calling subprogram and then create it again in the subprogram.

- The system does not execute the tool nose radius compensation in G71, G72, G73, G74, G75, G76 and cancel it temporarily.
- The system executes the tool nose radius compensation in G90, G94, it offsets one tool nose radius for G41 or G42.

4.1.7 Application

Machine a workpiece in the front tool post coordinate system as Fig. 4-13. Tool number: T0101, tool nose radius R=2, imaginary tool nose number T=3.

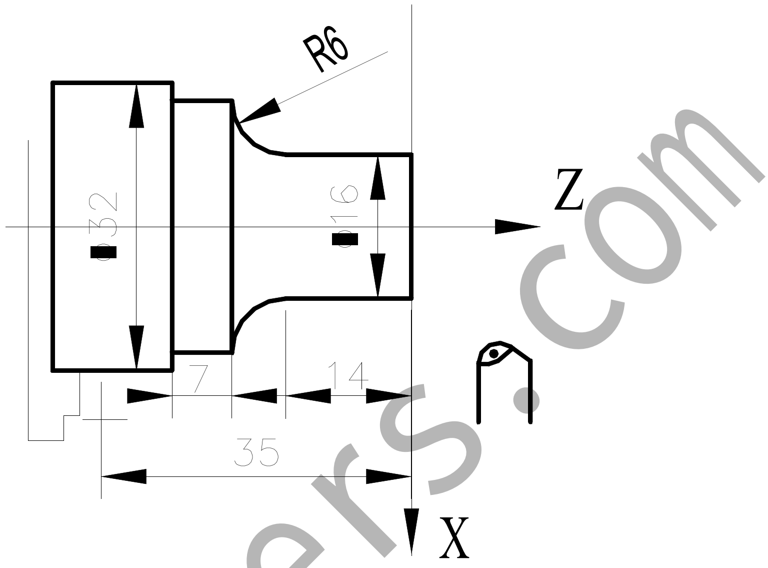


Fig. 4-13

For toolsetting in Offset Cancel mode, after toolsetting, Z axis offsets one tool nose radius and its direction is relative to that of imaginary tool nose and toolsetting point, otherwise the system excessively cuts tool nose radius when it starts to cut.

Set the tool nose radius R and imaginary tool nose direction in "***TOOL OFFSET&WEAR***" window as following:

Table 4-3

No.	X	Z	R	T
001			2.000	3
002	…	…	…	…
…	…	…	…	…
007	…	…	…	…
008	…	…	…	…

Program:

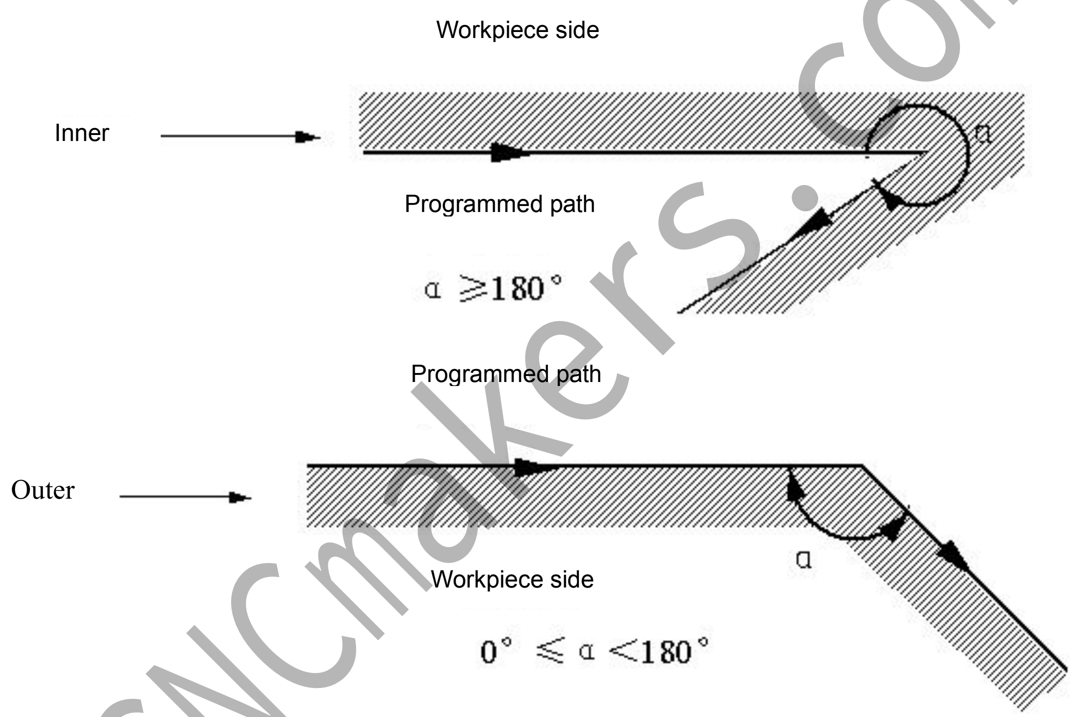
```
G00 X100 Z50 M3 T0101 S600;   (Position, start spindle, tool change and execute
                               tool compensation)
G42 G00 X0 Z3;                (Set tool nose radius compensation)
G01 Z0 F300;                  (Start cutting)
X16;
```

```
Z-14 F200;
G02 X28 W-6 R6;
G01 W-7;
X32;
Z-35;
G40 G00 X90 Z40;          (Cancel tool nose radius compensation)
G00 X100 Z50 T0100;
M30;
```

4.2 Tool nose radius compensation offset path

4.2.1 Inner and outer side

Inside is defined that an angle at intersection of two motion blocks is more than or equal to 180°; **Outside** is 0~180°.



4.2.2 Tool traversing when starting tool

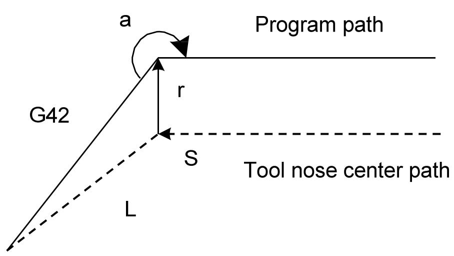
3 steps to execute tool nose radius compensation: tool compensation creation, tool compensation execution and tool compensation canceling.

Tool traverse is called tool compensation creation (starting tool) from offset canceling to G41 or G42 execution.

Note: Meanings of S, L, C in the following figures are as follows:
S——Stop point of single block; L——linear; C——circular.

(a) Tool traversing inside along corner(α≥180°)

1) linear —linear



2) Linear —circular

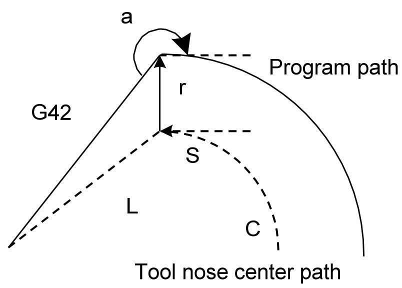
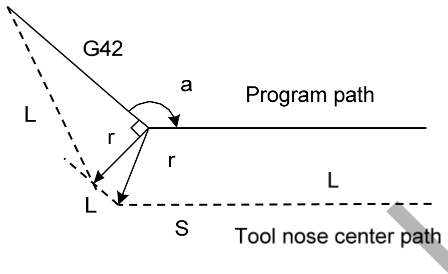


Fig.4-14a Linear —linear(starting tool inside) Fig. 4-14b Linear —circular (starting tool inside)

(b) Tool traversing inside along corner(180° > α≥90°)

1) linear —linear



2) linear —circular

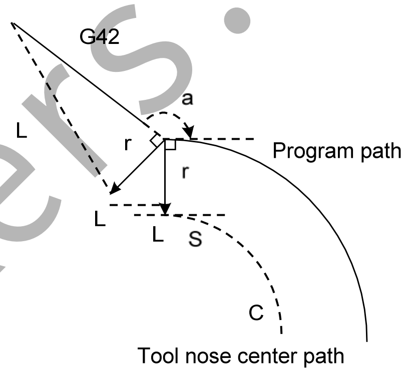
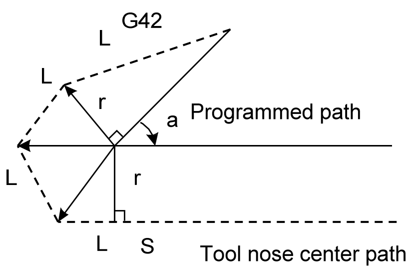


Fig.4-15a Linear —linear(starting tool outside) Fig.4-15b Linear—circular(starting tool outside)

(c) Tool traversing inside along corner (α < 90°)

1) linear —linear



2) linear —circular

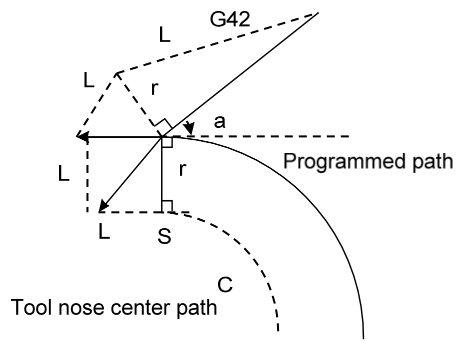


Fig.4-16a Linear —linear(starting tool outside) Fig. 4-16b Linear—circular (starting tool outside)

(d) Tool traversing inside along corner(α≤1°) , linear →linear

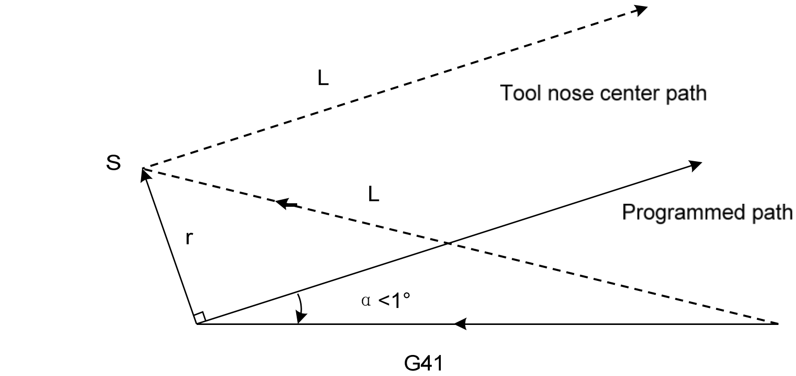


Fig. 4-17 Linear—linear (α < 1°, starting tool outside)

4.2.3 Tool traversing in Offset mode

Offset mode is called to ones after creating tool nose radius compensation and before canceling it.

z **Offset path without changing compensation direction in compensation mode**

(a) Tool traversing inside along corner(α≥180°)

1) linear —linear

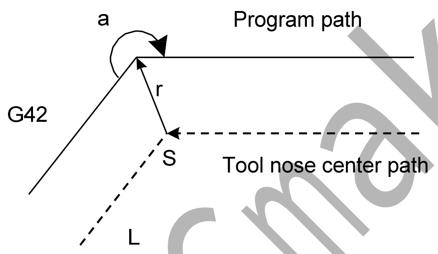


Fig. 4-18a Linear —linear (moving inside)

2) linear —circular

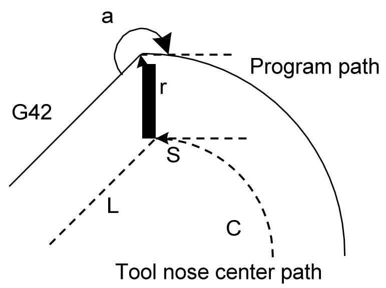


Fig. 4-18b Linear—circular (moving inside)

3) circular—linear

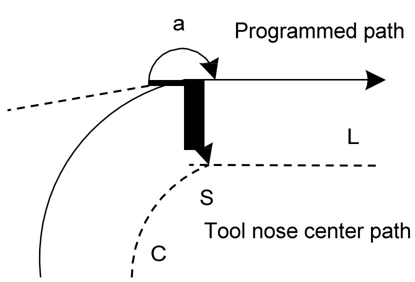


Fig. 4-18c Circular—linear(moving inside)

4) circular —circular

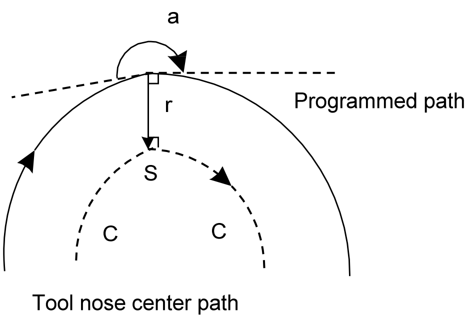


Fig. 4-18d Circular—circular(moving inside)

(b) Tool traversing outside along corner(180° > α≥90°)

1) linear —linear

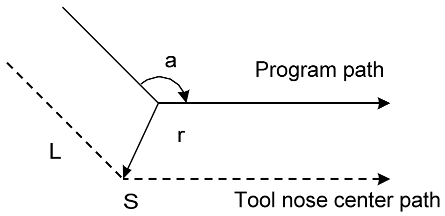


Fig. 4-19a Linear —linear
(180° > α≥90°, obtuse angle, moving outside)

2) linear —circular

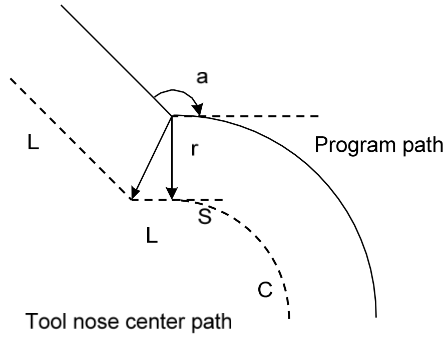


Fig. 4-19b Linear—circular
(180° > α≥90°, obtuse angle, moving outside)

3) circular—linear

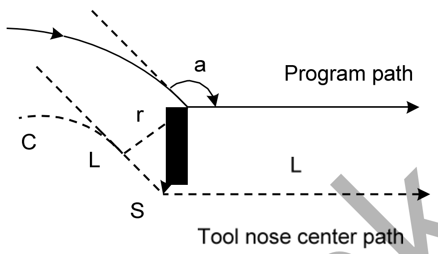


Fig. 4-19c cirular —linear
(180° > α≥90°, obtuse angle, moving outside)

4)circular —circular

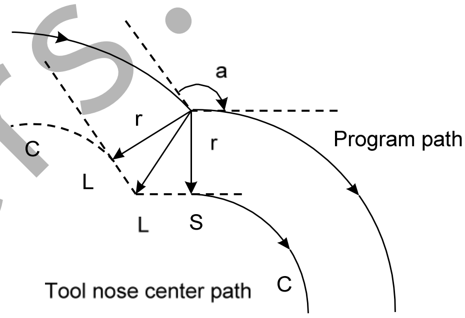
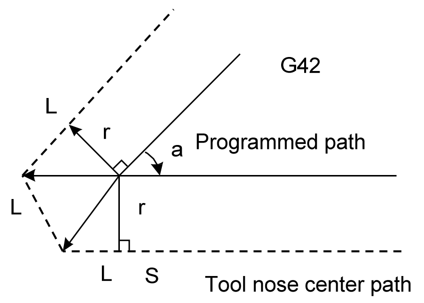


Fig. 4-19d cirular —cirular
(180° > α≥90°, obtuse angle, moving outside)

(c) Tool traversing outside along corner(α < 90°)

1) linear —linear



2) linear —circular

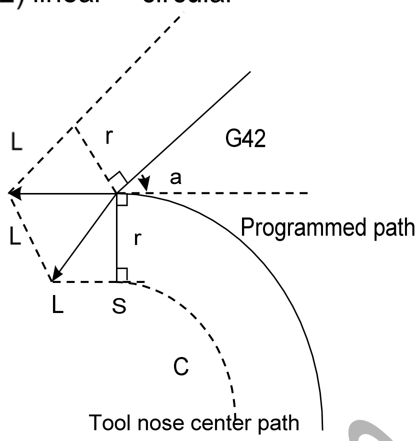
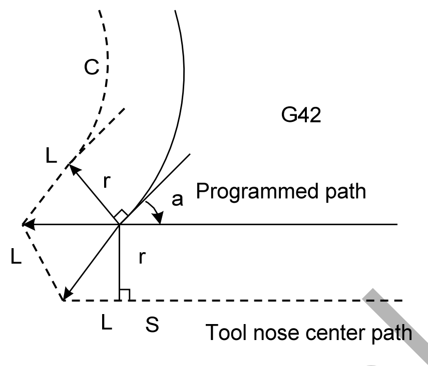


Fig. 4-20a Linear—Linea (moving outside) Fig. 4-20b Linear—circular (moving outside)

3)circular—linear



4) circular—circular

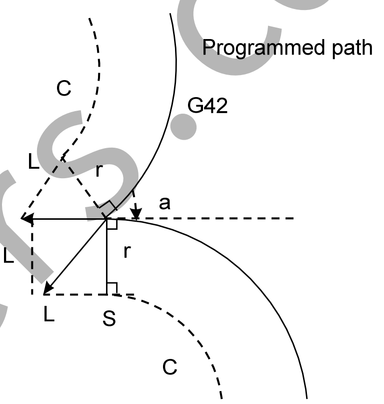


Fig.4-20c Circular—linear(moving outside) Fig.4-20d Circular—circular(moving outside)

5) Machining inside (α < 1°) and zoom in the compensation vector

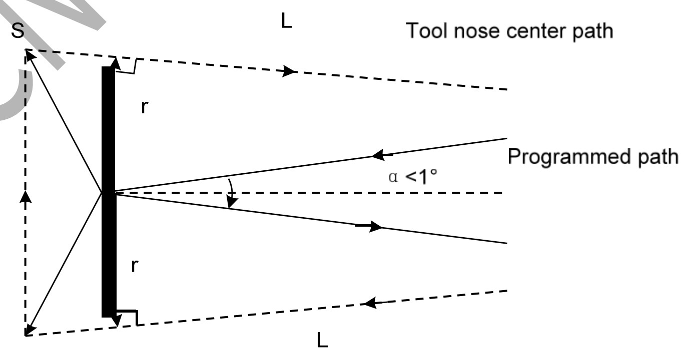


Fig. 4-20e Linear —linear (α < 1°, moving inside)

(d) Special cutting

1） Without intersection

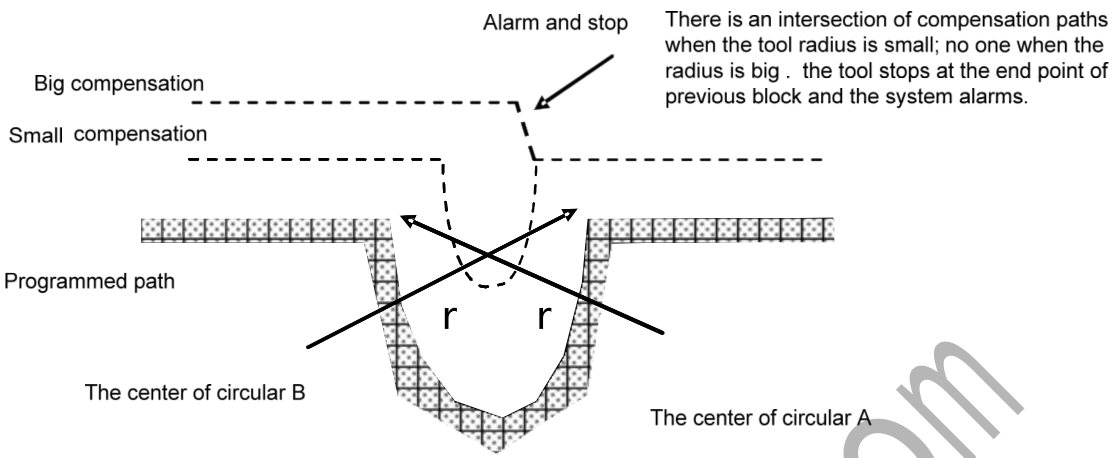


Fig. 4-21 Paths without intersection after offset

2）Center point and starting point of circular being the same one

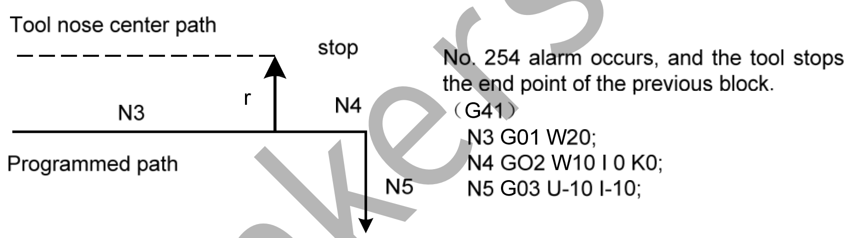


Fig. 4-22 Center point and starting point of circular being the same one

z **Offset path of compensation direction in compensation mode**

The compensation direction of tool nose radius is specified by G41 and G42 and the sign symbol is as follows:

Table 4-3

Comp. sign	Sign symbol of compensation value	
G Command	**+**	**-**
G41	Left compensation	Right compensation
G42	Right compensation	Left compensation

The compensation direction can be changed in compensation mode in special cutting, it cannot be changed at starting block and its following one. There is no inside and outside cutting when the system changes the compensation direction. The following compensation value is supposed to be positive.

1)linear—linear

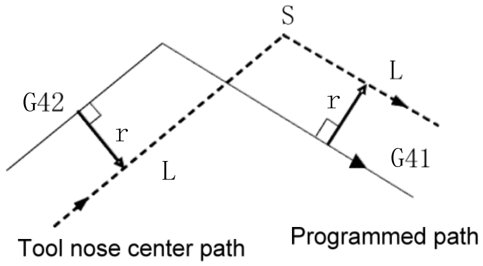


Fig. 4-23 Linear—linear
(changing compensation direction)

2) linear—circular

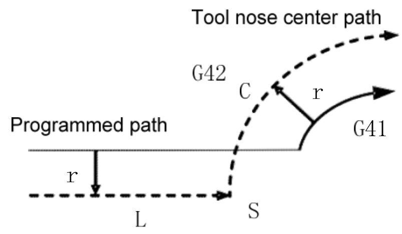


Fig. 4-24 Linear—circular
(changing compensation direction)

3)circular—linear

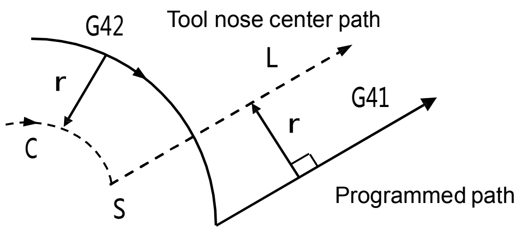


Fig. 4-25 circular—linear
(changing compensation direction)

4) circular—circular

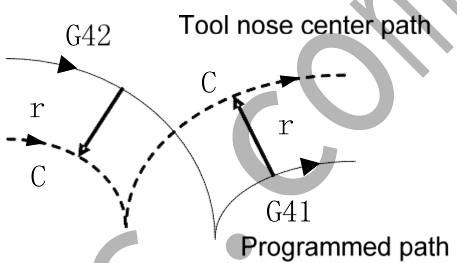


Fig. 4-26 circular—circular
(changing compensation direction)

5) No intersection when compensation is executed normally

When the system executes G41 and G42 to change the offset direction between block A and B, a vector perpendicular to block B is created from its starting point.

i) Linear----Linear

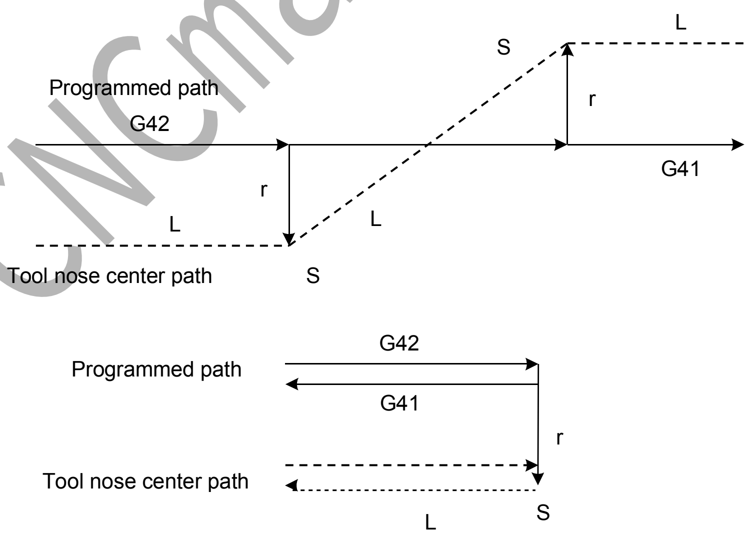


Fig. 4-27a Linear—linear, no intersection (changing compensation direction)

ii) Linear ---circular

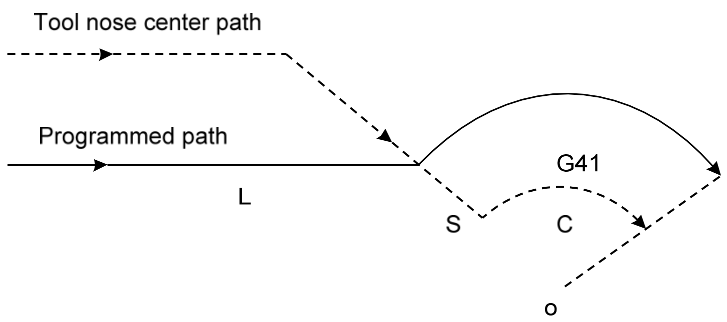


Fig. 4-27b Linear—circular without intersection (changing compensation direction)

iii) Circular-----circular

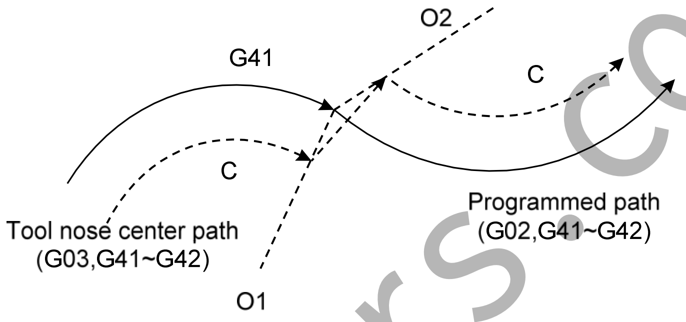


Fig. 4-27c Circular—circular without intersection (changing compensation direction)

4.2.4 Tool traversing in Offset canceling mode

In compensation mode, when the system executes a block with one of the followings, it enters compensation canceling mode, which is defined to compensation canceling of block.

1. Execute G40 in a program;
2. Execute M30.

The system cannot execute G02 and G03 when canceling C tool compensation (tool nose radius compensation), otherwise the system alarms and stops run.

In compensation canceling mode, the system executes the block and ones in the register for tool nose radius compensation. At the moment, the run stops after one block is executed when single block is ON. The system executes the next one but does not read its following one when pressing **CYCLE START** button again.

(a) Tool traversing inside along corner(α≥180°)

1) linear —linear

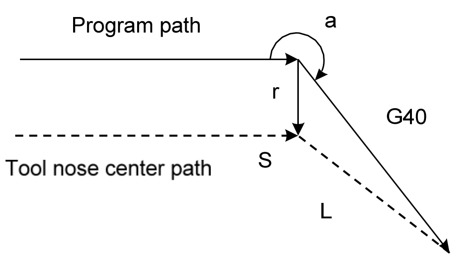


Fig. 4-28a linear-linear
(moving inner and canceling offset)

2) circular—linear

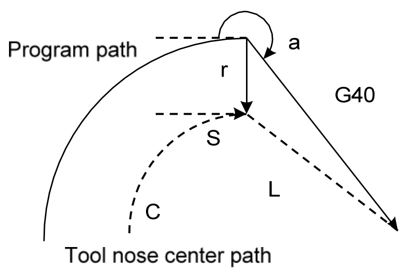


Fig. 4-28b Circular-linear
(moving inner and canceling offset)

(b) Tool traversing outside along corner(180° > α≥90°)

1) linear —linear

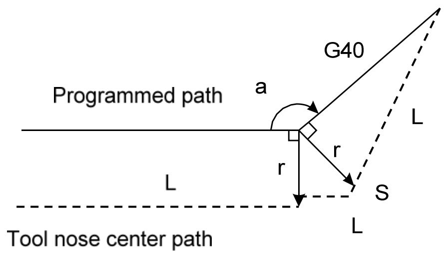


Fig. 4-29a linear—linear (α≥90° moving outside and canceling offset)

2) circular—linear

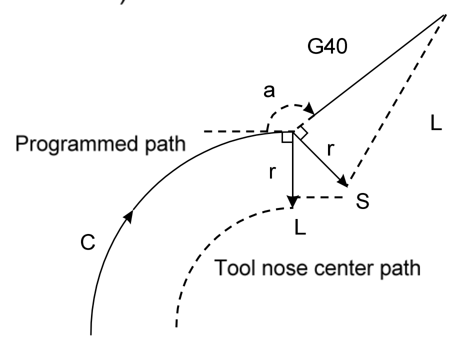


Fig. 4-29b Circular—linear (α≥90°moving outside and canceling offset)

(c) Tool traversing outside along corner(α < 90°)

1) linear —linear

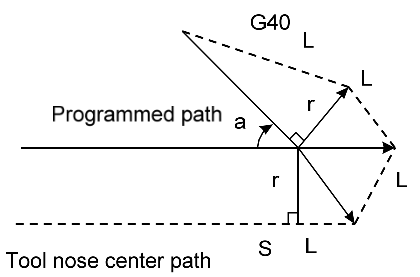


Fig. 4-30a Linear—linear (α < 90°cutting outside and canceling offset)

2) circular —linear

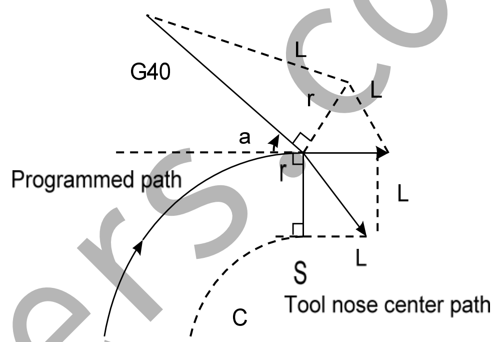


Fig. 4-30b Circular—linear (α < 90°cutting outside and canceling offset)

(d) Tool traversing outside along corner(α < 1°) ; linear→ linear

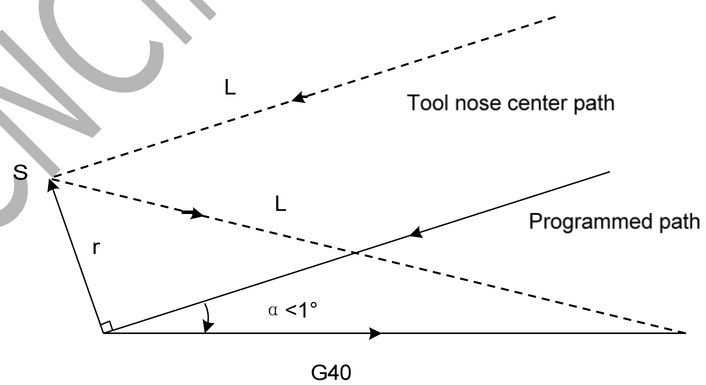


Fig. 4-31 Linear—linear (α < 1°cutting outside and canceling offset)

4.2.5 Tool interference check

"Interference" is defined that the tool cuts workpiece excessively and it can find out excessive cutting in advance, the interference check is executed even if the excessive cutting is not created, but the system cannot find out all tool interferences.

(1) Fundamental conditions

1) The tool path direction is different that of program path (angle is 90°~270°).

2) There is a big difference (α > 180°) for two angles between starting point and end point of tool nose center path, and between starting point and end point of program path.

Example: linear machining

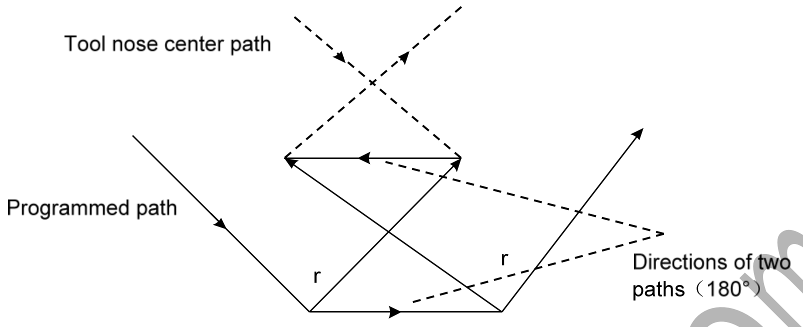


Fig. 4-32a Machining interference (1)

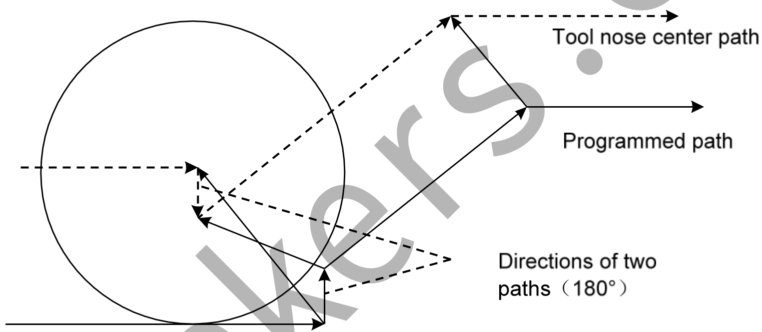


Fig. 4-32b Machining interference (2)

(2) Executing it without actual interference

1) Concave groove less than compensation value

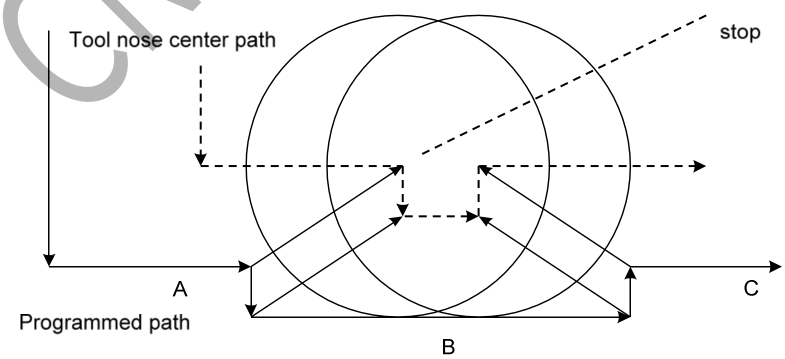


Fig. 4-33 Executing interference (1)

Directions of block B and tool nose radius compensation path are opposite without interference, the tools stops and the system alarms.

2) Concave channel less than compensation value

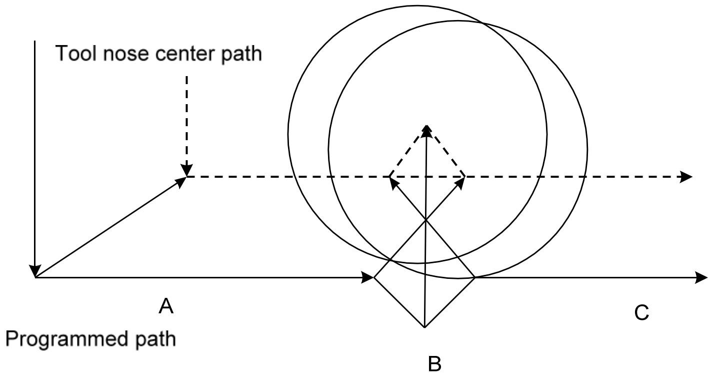


Fig. 4-34 Executing interference (2)

Directions of block B and tool nose radius compensation path are opposite without interference, the tools stops and the system alarms.

4.2.6 Commands for canceling compensation vector temporarily

In compensation mode, the compensation vector is cancelled temporarily in G50, G71~G76 and is automatically resumed after executing the commands. At the moment, the compensation is cancelled temporarily and the tool directly moves from intersection to a point for canceling compensation vector. The tool directly moves again to the intersection after the compensation mode is resumed.

- **Setting coordinate system in G50**

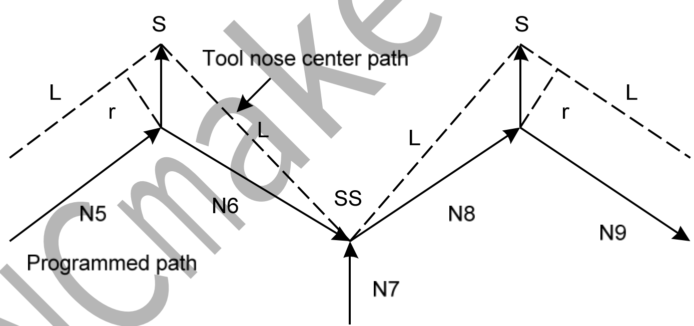


Fig. 4-35 Temporary compensation vector in G50

Note: SS indicates a point at which the tool stops twice in Single mode.

- **Reference point automatic return G28**

In compensation mode, the compensation is cancelled in a middle point and is automatically resumed after executing the reference point return in G28.

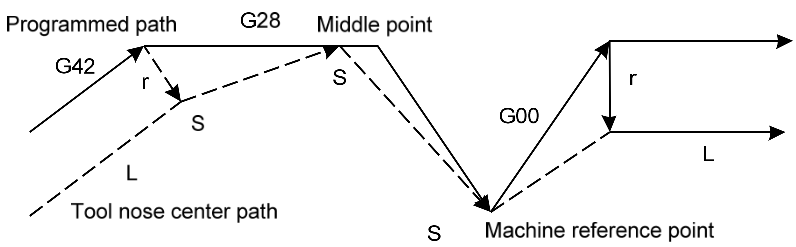


Fig. 4-36 Cancel compensation vector temporarily in G28

- **G71～G75 compound cycle; G76, G92 thread cutting**

  When executing G71～G76 , G96 thread cutting, the system does not execute the tool nose radius compensation and cancel it temporarily, and there is G00, G01,in the following blocks, and the system automatically recovers the compensation mode.

- **G32, G33,G34 thread cutting**

  They cannot run in the tool nose radius compensation mode, otherwise, No.131 alarm occurs "……CANNOT USED TO C COMPENSATION".

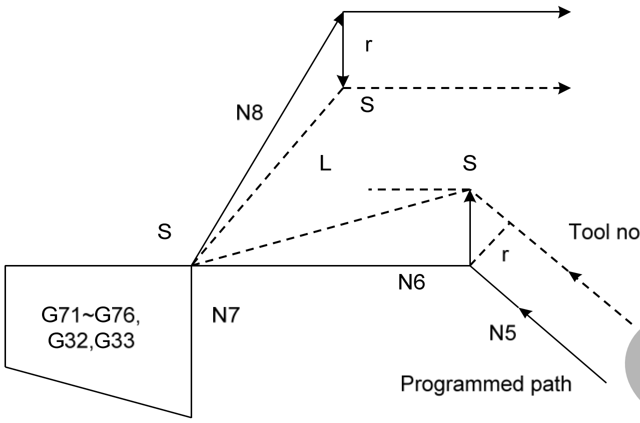


Fig. 4-37 Cancel compensation vector temporarily in G71～G76

- **G90, G94**

  Compensation method of tool nose radius compensation in G90 or G94:

  A. Cancel the previous tool nose radius compensation;

  B. Create the previous C compensation before cutting, and the path ① in the following figure creates the previous radius compensation mode;

  C. The paths ②,③ in the following figure are the radius compensation cutting;

  D. The path ④ in the following figure can cancel the radius compensation, and the tool returns to the cycle starting point; there is G00,G01 in the following block, and the CNC automatically recovers the compensation mode.

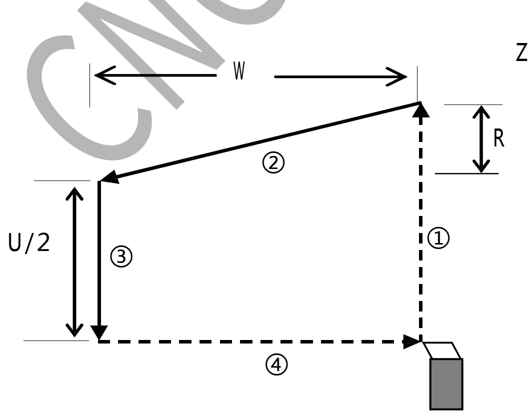


Fig. 4-38 Offset direction of tool nose radius compensation in G90

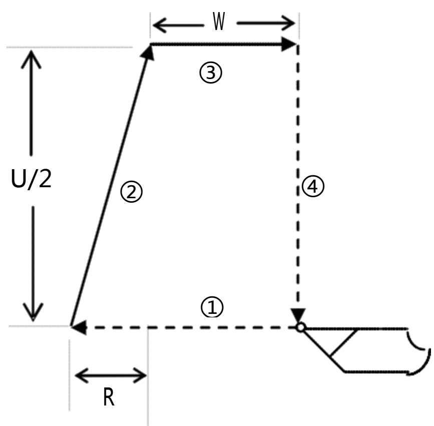


Fig. 4-39 Offset direction of tool nose radius compensation in G94

4.2.7 Particulars

- **Inside chamfer machining less than tool nose radius**

At the moment, the tool inside offset causes an excessive cutting. The tool stops and the system alarms (P/S41) when starting the previous block or chamfer moving. But the tool stops the end point of previous block when **Single** is ON.

- **Machining concave less than tool nose diameter**

There is an excessive cutting when the tool nose center path is opposite to program path caused by tool nose radius compensation. At the moment, the tool stops and the system alarms when starting the previous block or chamfer moving.

- **Machining sidestep less than tool nose radius**

The tool center path can be opposite to program path when the sidestep is less than tool nose radius and is an circular in program. At the moment, the system automatically ignores the first vector and directly moves end point of second vector linearly. The program stops at the end point in single block and otherwise the cycle machining is continuously executed. If the sidestep is a linear, compensation is executed correctly and the system does not alarm (but the not-cutting is still reserved).

- **Subprograms in G Commands**

The system must be in canceling compensation mode before calling subprograms. After calling subprograms, the offset is executed and the system must be in canceling compensation mode before returning to main programs, otherwise the system alarms.

- **Changing compensation value**

(a) Change compensation value in canceling tool change mode. New compensation value is valid after tool change when the compensation value is changed in compensation mode.

(b) Compensation value sign symbol and tool nose center path

G41 and G42 are exchanged each other if the compensation value is negative (-). The tool moves along inside when its center moves along outside of workpiece, and vice versa.

Generally, the compensation value is positive (+) in programming. The compensation value is negative (-) when the tool path is as the above-mentioned (a), and vice versa.

Besides, direction of tool nose offset changes when offset value sign symbol is changed, but we suppose the direction of tool nose is not changed. Generally, the offset value sign symbol is not changed.

- **End point of programming circular out of circular**

The tool stops and the system alarms and displays “End point of circular is not on circular” when the end point of circular is not on circular in programs.

Volume Ⅱ Operation

CHAPTER 1 OPERATION MODE AND DISPLAY INTERFACE

1.1 Panel division

C1000T CNC system uses an integrated panel, which is divided as follows:

C1000T panel division

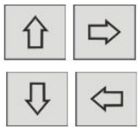
1.1.1 State indication

X Y Z 4th C	Axis zero return completion indictor
ALM READY RUN	Three color indictor

1.1.2 Edit keypad

Press key	Name	Function
RESET	RESET	CNC reset, feed, output stop etc.
X Z U W	Address key	Address input
O␣ N# G* PQ X Z U W IA JB KC RV M[S] TY H= LD FE		Double address key, switching them by pressing it repetitively
. / < - + >	symbol	Three address key, switching them by pressing it repetitively
7 8 9 4 5 6 1 2 3 0	Number key	Number input
. < >	Decimal point	Decimal point input
DATA INPUT	Input key	Parameter, compensation value and other data input
DATA OUTPUT	Output key	Communication output
CHANGE	Change key	Switching message, display

D INSERT ALTER DELETE	Edit key	Inserting, altering, deleting programs, fields in EDIT working mode(INSERT ALTER compound key, switching them by pressing it repetitively)
EOB	EOB key	Inputting the end character of block

Press key	Name	Function
	Cursor move keys	controlling cursor move
	Window key	Switch the window in the same display window

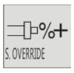
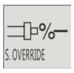
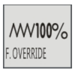
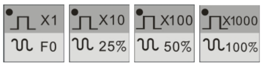
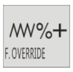
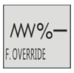
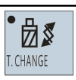
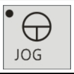
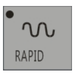
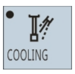
1.1.3 Menu display

Menu key	Remark
POSITION	To enter POS interface. There are RELATIVE POS, ABSOLUTE POS, INTEGRATED POS, POS&PRG windows in this interface.
PROGRAM	To enter PRG interface. There are PRG CONTENT, PRG LIST, PRG STATE windows in this interface.
OFFSET	To enter TOOL OFFSET, MACRO interface (switching between interfaces by pressing it repeatedly). OFFSET interface displays offset values; MACRO for CNC macro variables.
ALARM	To enter ALARM interface. There are ALARM, WARN LOG windows in this interface.
SETTING	To enter Setting, Graphic interface (switching between interfaces by pressing it repeatedly). There are SWITCH,PARM OPERATION,PASSWORD SETTING, In Setting interface.GRAPH window can display the movement path of feed axis
PARAVETER	To enter BIT PARAMETER, DATA PARAMETER, SCREW-PITCH COMP interfaces (switching between each interface by pressing it repeatedly).
DAGNOSS	To enter CNC DIAGNOSIS, PLC STATE, PLC VALUE, TOOL PANEL, VERSION MESSAGE interfaces (switching between each interfaces by pressing this key repeatedly). CNC DIAGNOSIS, PLC STATE, PLC VALUE interfaces display CNC internal signal state, PLC addresses, data message; TOOL PANEL is used for machine soft keypad operation; the VERSION MESSAGE interface displays CNC software, hardware and PLC version No.
GRAPH	Enter ladder interface. There are PLC version, PLC state,PLC data,ladder interface in this interface.(switching between interfaces by pressing it repeatedly)

1.1.4 Machine panel

The key functions on C1000T machine panel are defined by PLC program (ladder), the detailed function meanings are referred to machine manufacturer manual.

The functions of this C1000T machine panel keys defined by standard PLC program are as follows:

Key	Name	Function explanation	Operation mode
	Feed hold key	Dwell commanded by program, MDI code	Auto, MDI
	Cycle Start key	Cycle start commanded by program, MDI code	Auto, MDI
 	Feedrate Override keys	Adjusting feedrate	Auto, MDI, Edit , Machine zero return, MPG, Step, Manual, Program zero return
	Feedrate override 100% press key	Adjust the feedrate rate	Auto, MDI, Edit , Machine zero return, MPG, Step, Manual, Program zero return
	Rapid override keys	Adjusting rapid traverse	Auto, MDI, Machine zero return, Manual, Program zero return
 	Spindle override keys	spindle speed adjustment (spindle analog control active)	Auto, Edit, MDI, Machine zero return, Manual, Step, MPG, Program zero return
	Manual tool change key	manual tool change	Machine zero return, Manual, Step, MPG, Program zero return
	JOG key	spindle jog on/off	Machine zero return, Manual, Step, MPG, Program zero return
	Rapid Switch	switch rapid speed/ Feed speed	
	Lubricating key	For lubricating ON/OFF	
	Cooling key	For cooling ON/OFF	Auto, Edit, MDI, Machine zero return, Manual, Step, MPG, Program zero return

Key	Name	Function explanation	Operation mode
S. CW S. STOP S. CCW	Spindle control keys	For spindle CCW For spindle stop For spindle CW	Machine zero return, Manual, Step, MPG, Program zero return
RAPID	Rapid traverse key	For rapid traverse /feedrate switching	Auto, MDI, Manual
X	X feed key	Positive/negative movement of each axis in Manual, Step mode	Machine zero return, Step, Manual, Program zero return mode
Z	Z feed key		
Y	Y feed key		
4th	The 4th feed key		
C/S	Cs feed key		
MPG	MPG axis selection key	Each axis selection in MPG mode	MPG mode
X1 F0, X10 25%, X100 50%, X1000 100%	MPG/Step increment and Rapid override selection key	Move amount per MPG scale 1/10/100/1000 mm Move amount per step 1/10/100/1000 mm Rapid override F0, F25%, F50%,F100%	Auto, MDI, Machine zero return, Manual, Step, MPG, Program zero return
SINGLE	Single Block switch	For switching of block/blocks execution, Single block indicator lights up if Single mode is active	Auto, MDI
SKIP	Block Skip switch	For skipping of block headed with"/"sign, if its switch is set for ON, the Block Skip indicator lights up	Auto, MDI
MACHINE LOCK	Machine Lock key	If the machine is locked, its indicator lights up, and X, Z axis output is inactive.	Auto, MDI, Edit, Machine zero return, Manual, Step, MPG, Program zero return
MST M.S.T LOCK	M.S.T. Lock key	If the miscellaneous function is locked, its indicator lights up and M, S, T function output is inactive.	Auto, MDI

Key	Name	Function explanation	Operation mode
	Dry Run key	If dry run is active, the Dry run indicator lights up. Dry run for program/MDI codes	Auto, MDI
	Edit mode key	To enter Edit mode	Auto, MDI, Machine zero return, Manual, Step, MPG, Program zero return
	Auto mode key	To enter Auto mode	MDI, Edit, Machine zero return, Manual, Step, MPG, Program zero return
	MDI mode key	To enter MDI mode	Auto, Edit, Machine zero return, Manual, Step, MPG, Program zero return
	Machine zero return mode key	To enter Machine zero return mode	Auto, MDI, Edit, Manual, Step, MPG, Program zero return
	Step/MPG mode key	To enter Step or MPG mode (one mode by parameter)	Auto, MDI, Edit, Machine zero return, Manual, Program zero return
	Manual mode key	To enter Manual mode	Auto, MDI, Edit, Machine zero return, Step, MPG, Program zero return
	Program zero return mode key	To enter Program zero return mode	Auto, MDI, Edit, Machine zero return, Step, MPG, Manual

1.2 Summary of operation mode

There are 7 modes in C1000T, which are Edit, Auto, MDI, Machine zero, Step/MPG, Manual, Program Zero modes.

- **Edit mode**

  In this mode, the operation of part program setup, deletion and alteration can be performed.

- **Auto mode**

  In this mode, the program is executed automatically.

- **MDI mode**

  In this mode, the operation of parameter input, command blocks input and execution can be performed.

- **Machine zero mode**

  In this mode, the operation of X, Z machine zero return can be performed separately.

- **MPG / Step mode**

  In the Step/MPG feed mode, the moving is performed by an increment selected by CNC system.

- **Manual mode**

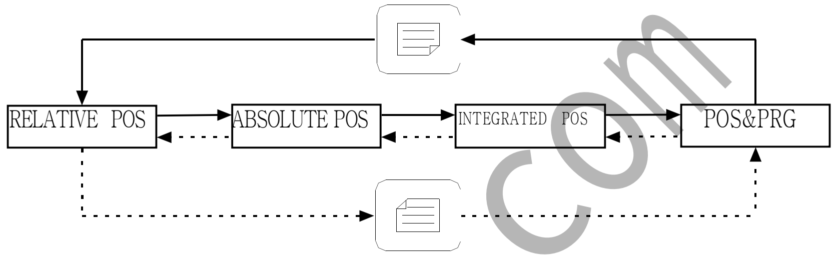
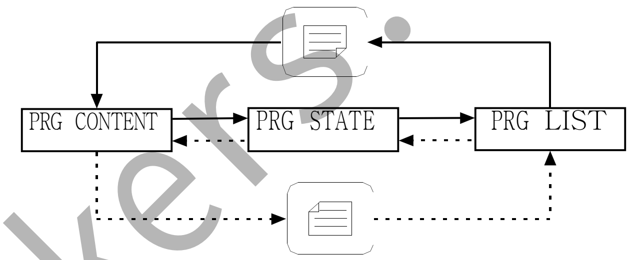
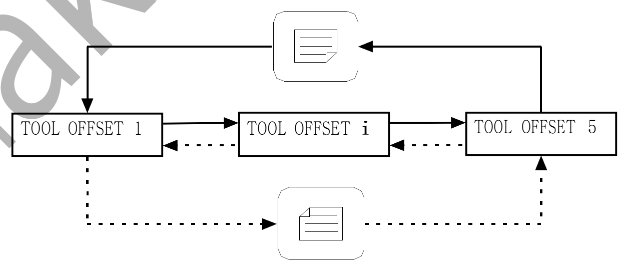
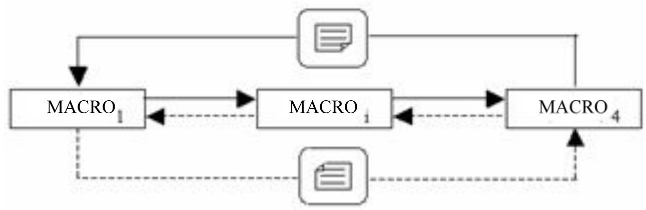
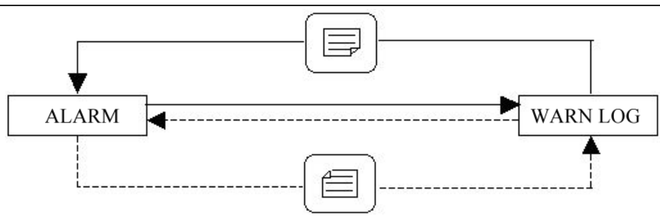
  In this mode, the operation of Manual feed, Manual Rapid, feedrate override adjustment, Rapid override adjustment and spindle ON/OFF, cooling ON/OFF, Lubricating ON/OFF, spindle jog, manual tool change can be performed.

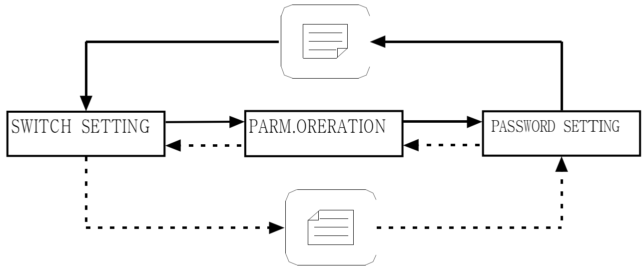
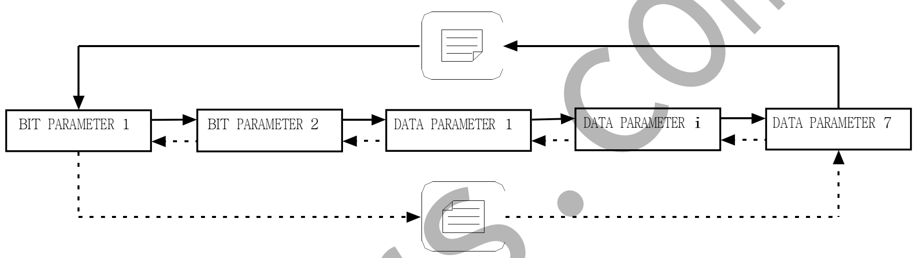
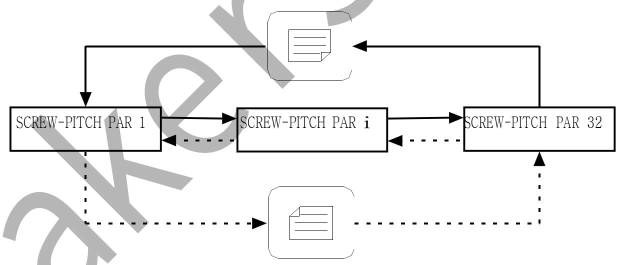
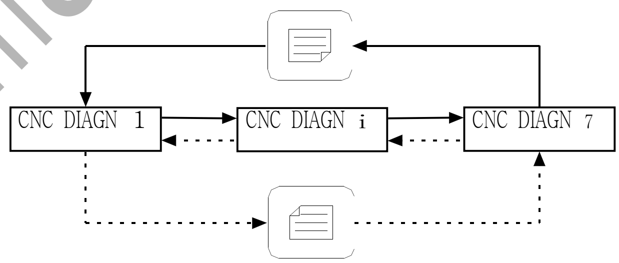
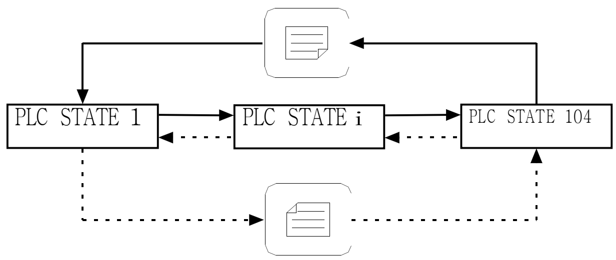
- **Program zero return mode**

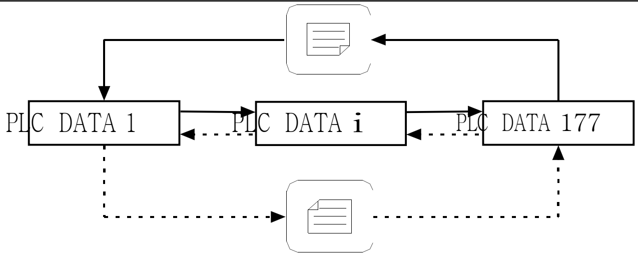
  In this mode, the operation of X, Z program zero return can be performed separately.

1.3 Display interface

C1000T has 9 interfaces such as POS, PRG etc., and there are multiple windows in each interface. Each interface (window) is separated with the operation mode. See the following figures for the display menu, display interface and window layers:

Menu key	Display interface	Display window
POSITION	POS interface	
PROGRAM	PRG interface	
OFFSET	TOOL OFFSET interface	
	MACRO interface	
ALARM	ALARM interface	

Menu key	Display interface	Display window
SETTING	SETTING interface	
	GRAPH interface	GRAPH
PARAMETER	BIT PARAMETER	
	DATA PARAMETER	
	SCREW-PITCH COMP	
DIAGNOSIS	CNC DIAGNOSIS	
	PLC SIGNAL	

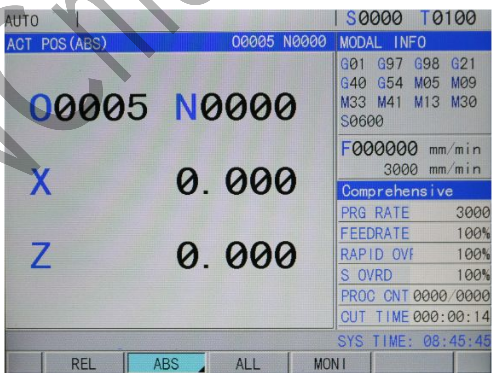
Menu key	Display interface	Display window
	PLC VALUE	
	TOOL PANEL	TOOL PANEL
	VERSION MESSAGE	VERSION MESSAGE
GRAPH	GRAPH	GRAPH

1.3.1 POS interface

Press POSITION to enter POS interface, which has four interfaces such as ABSOLUTE POS, RELATIVE POS, INTEGRATED POS and POS&PRG, and they can be viewed by  or  keys.

1) ABSOLUTE POS display interface

The X, Z coordinates displayed are the absolute position of the tool in current workpiece coordinate system, X,Z coordinates are memorized as power is down and the workpiece coordinate system is specified by G50,G54~G59



Note: It displays "PRG. F" In Edit, Auto, MDI;"MANUAL.F" in Machine zero, Program zero, Manual mode ; "HNDL INC" in MPG mode; "STEP INC" in Step mode.

ACT. F: Actual speed after feedrate override in a machining. FED OVRI: An override by feedrate override switch

G CODE: Modal value of 01 group G code and 03 group G code

PART CNT: Part number plus 1 when M30 (or M99 in the main program) is executed

CUT TIME: Time counting starts if Auto run starts, time units are hour, minute and second

RAP OVRI: Current rapid rate

SPI OVRI: Spindle override display as the BIT4 of the parameter No.001 is set to 1.

S0000: Feedback spindle speed of spindle encoder, and spindle encoder is necessary.

T0100: Current tool No. and tool offset No.

The parts counting and the cut time are memorized at power-down, the clearing ways for them are as follows:

PART CNT clearing: Press CANCEL key then press N # key.

CUT TIME clearing: Press CANCEL key then press T Y key.

2) **RELATIVE POS display interface**

The U, W coordinates displayed are the current position relative to the relative reference point, and they are held on at CNC power on. They can be cleared at any time. If U, W coordinates are cleared, the current position is the relative reference point. When CNC parameter No.005 Bit1=1, as the absolute coordinates are set by G50 code, U, W coordinates are identical with the set absolute coordinates.

The clearing steps of U, W relative coordinates:

In RELATIVE POS window, press and hold U key till the "U" in the window blinks, press CANCEL key to clear U coordinate;

In RELATIVE POS window, press and hold W key till the "W" in the window blinks, press CANCEL key to clear W coordinate.

Note: When Y, the 4th, the 5th axis are valid, their zero clearing method are the same those of the above.

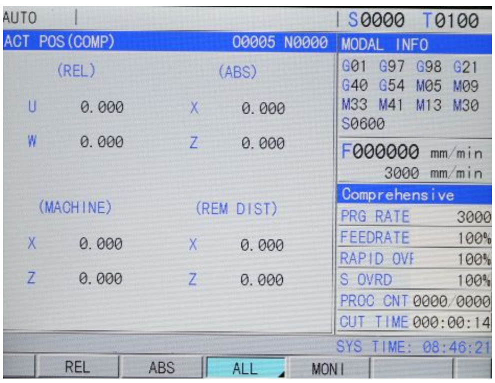
3) INTEGRATED POS display interface

In INTEGRATED POS window, the RELATIVE, ABSOLUTE, MACHINE, DIST TO GO (only in Auto and MDI mode) are displayed together.

The displayed value of machine coordinate is the current position in the machine coordinate system which is set up according to the machine zero.

The DIST TO GO is the difference of the target position by block or MDI command to the current position.

The display window is as follows:



4) POS&PRG display interface

In this window, it displays ABSOLUTE, RELATIVE coordinate of the current position (ABSOLUTE, DIST TO GO of current position will be displayed if BIT0 of bit parameter No.180 is set to 1) as well as 6 blocks of current program together. During the program execution, the displayed blocks are refreshed dynamically and the cursor is located in the block being executed.

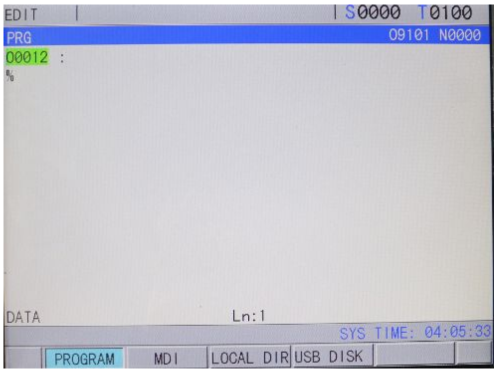
Note: Press CHANGE in POSITION window and switch the cutting time, and the system time at the bottom right corner below:

1.3.2 PRG interface

Press PROGRAM to enter PRG interface, which has 4 windows such as PRG CONTENT, PRG STATE, PRG LIST,FILE LIST in non-Edit modes, and they can be viewed by or key. There is only PRG CONTENT window in Edit mode, all the blocks of the current program can be shown by pressing or key.

1) PRG CONTENT window

In this window, the program content including current block can be displayed. In Edit mode, the program content can be viewed forward or backward by pressing or key.



2) PRG STATE window

In this window, press  to enter PRG STATE window.

3) PRG LIST window

In program content window, press  to enter the program list window. The window displays all machine programs, and the first 3 lines of the current program are displayed in the below of the window.

In this window, it displays:

(a) PART-PRG NO.: Number of the programs that can be saved and programs saved by CNC (including subprogram)

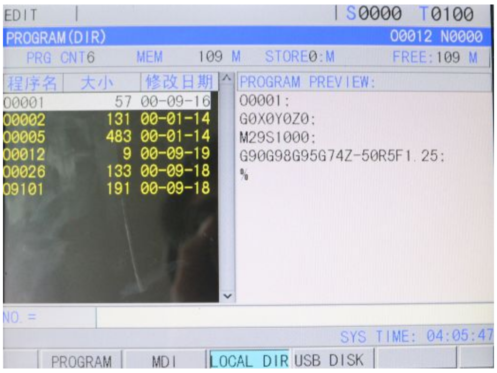
(b) MEMORY SIZE: The maximum capacity (MB) for the programs that can be saved and the capacity that has been taken up by programs.

(c) PRG LIST: Number of the programs saved by name size order

(d) USED: Display the memory capacity of part programs saved in the CNC

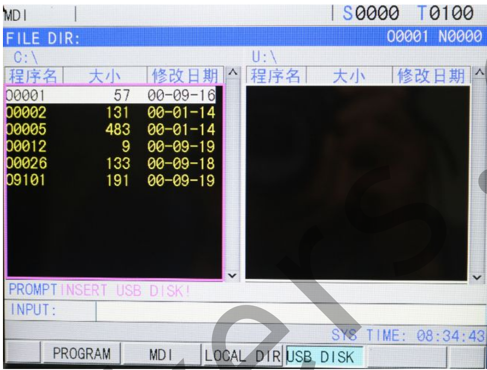
(e) PRG SIZE: Display the memory size of the program where the current cursor is

(f) PROGRAM LIST: Display orderly the saved program No. based on the program name



4) FILE DIRECTORY window

In program list window, press to enter the file directory window.



1.3.3 TOOL OFFSET&WEAR, MACRO, TOOL-LIFE MANAGEMENT interfaces

OFFSET is a compound key, press OFFSET key once in other window, it enters the TOOL OFFSET window, press OFFSET key again, it enters the MACRO interface.

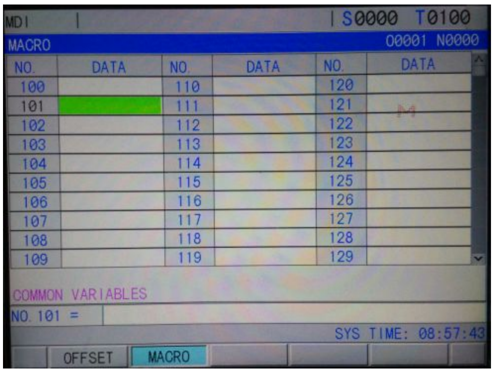
If Bit0 of bit parameter No.002 is 1, OFFSET key is pressed again, it enters the TOOL-LIFE MANAGEMENT interface.

1. TOOL OFFSET&WEAR interface

There are 7 windows and 33 offset & wear No. (No.000 ~ No.032) available for user in this interface, which can be shown as follows by pressing or key.

2. MACRO interface

There are 20 windows in this interface, which can be shown by pressing  or  key. In Macro window there are 600 (No.100 ~ No.199 and No.500 ~ No.999) macro variables which can be specified by macro command or set by keypad.

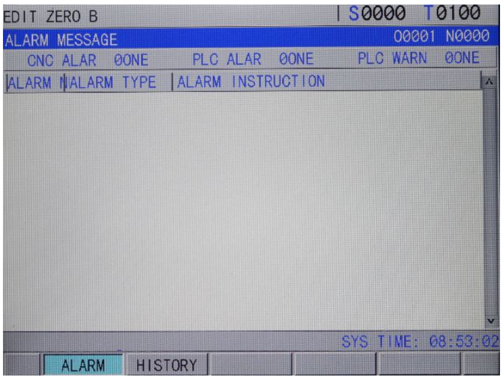


Volume II Operation

1.3.4 ALARM interface

1) Alarm:

Press ALARM key to enter alarm interface, which can be viewed by or key, the window is as follows:

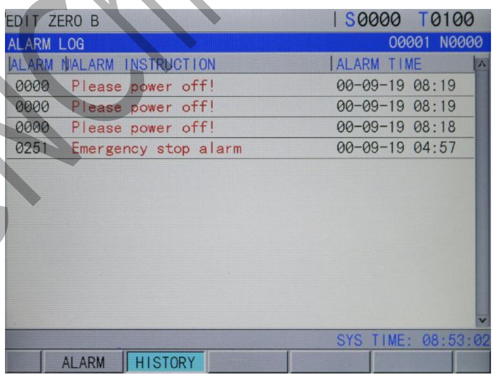


Note: Alarm clearing: It may clear alarms by pressing RESET (it should press RESET and CANCEL keys together to clear No.100 alarm). If the current alarm window is as follows:

2) Alarm log:

Press ALARM key again to enter WARN LOG interface, 200 messages can be viewed by pressing or key.

Sequence of warn log: The latest log message is shown on the forefront of the 1st window, and the others queue in sequence. If the messages are over 200, the last one will be cleared.



Note: Manual clear alarm log: press CANCEL + J B in 2-level password to clear all log message.

1.3.5 Setting interface

SETTING is a compound key, press SETTING key in other window, it enters the SETTING interface, press it again, it enters the GRAPHIC interface. Press SETTING key repeatedly, it switches between

SETTING and GRAPHIC interfaces.

1. SETTING interface

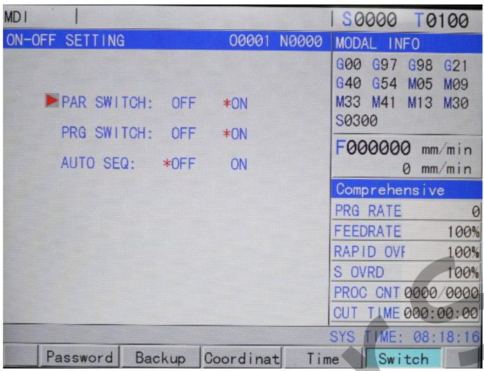
There are 5 windows in this interface , which can be viewed by  or  key.

SWITCH SETTING: It is used for the parameter, program, auto sequence No. on-off state.

PARM SWT: when it is turned for ON, the parameters are allowed to be altered; it is turned for OFF, the parameters are forbidden to be altered.

PROG SWT: when it is turned for ON, the programs are allowed to be edited; it is turned for OFF, the programs are forbidden to be edited.

AUTO SEG: when it is turned for ON, the block No. is created automatically; it is turned for OFF, the block No. is not created automatically, but manually if needed.



DATA OPERATION: In this window, the CNC data (such as bit parameter, data parameter, screw-pitch parameter, tool offset) can be backup and restored.

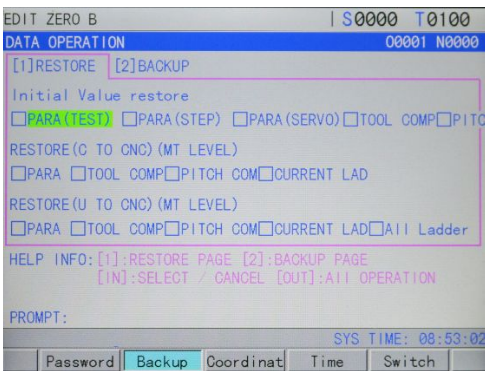
Restore original value:Restore parameters,tool compensation,screw-pitch to system default value.

C disk data restore to CNC:Restore data files which are backup to system disk to system

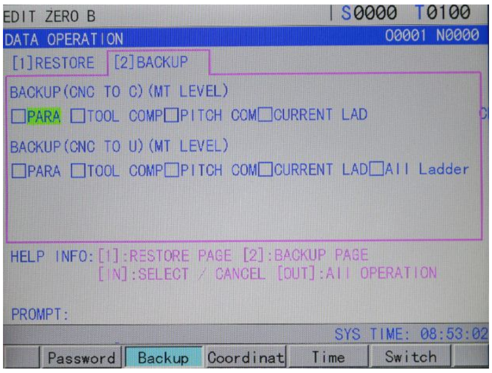
U disk data restore to CNC:Restore data files which are backup to U disk to system

CNC data backup to C disk:Backup current parameters,tool compensation,screw- pitch and ladder to system disk

CNC data backup to U disk :Backup current parameters,tool compensation,screw- pitch and ladder to U disk



User window of 3, 4, 5 level



User window of 2 level

PASSWORD SETTING: For user operation level display and setting

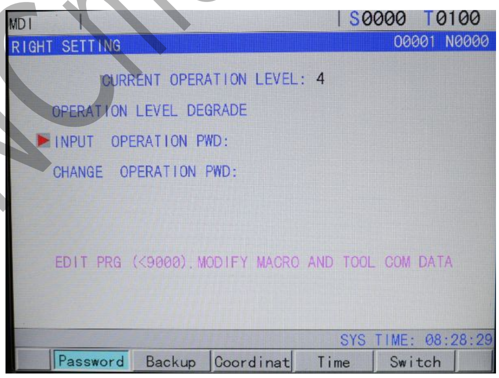
By descending sequence the password of C1000T is classified for 4 levels, which are machine builder (2) level, equipment management (3) level, technician (4) level, machining operation (5) level.

Machine builder level: The CNC bit parameter, data parameter, screw-pitch parameter, tool offset data, part program edit(including macro), PLC ladder editing and alteration, ladder upload and download operations are allowed;

Equipment management level: Initial password is 12345, the CNC bit parameter, data parameter, tool offset data, part program edit operations are allowed;

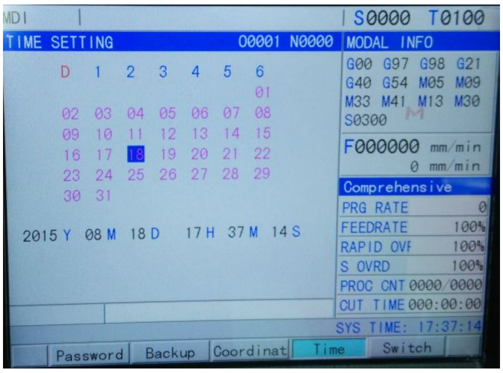
Technician level: Initial password is 1234, tool offset data (for toolsetting), macro variables, part program edit operations are allowed; but the CNC bit parameter, data parameter, screw-pitch parameter operations are forbidden.

Operation level: No password. Only the machine panel operation is allowed, the operations of part program edit and selection, the alteration operations of CNC bit parameter, data parameter, screw-pitch parameter, tool offset data are forbidden.



Time, data: display current time and date.

Press CHANGE to enter the change mode, press ⇦ , ⇨ to change Year/ Month/ Day/ Minute/ Second, press ⇧ , ⇩ to change the value. If the system escapes the mode, CHANGE is pressed again.



2) Graph interface

In GRAPH window, it may perform the graphic scaling and clearing operation.

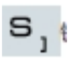
1.3.6 BIT PARAMETER, DATA PARAMETER, SCREW-PITCH COMP interfaces

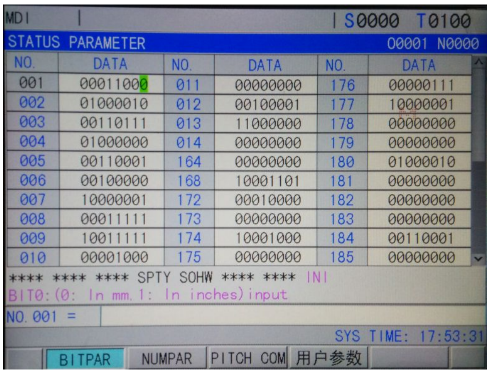
is a compound key, it enters BIT PARAMETER, DATA PARAMETER and SCREW-PITCH COMP interfaces by pressing this key repeatedly.

1) BIT PARAMETER interface

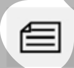
Press 

key, it enters BIT PARAMETER interface, there are 48 bit parameters which are displayed by 2 windows in this interface, and they can be viewed or altered by pressing  or  key to enter the corresponding window. It is shown as follows:

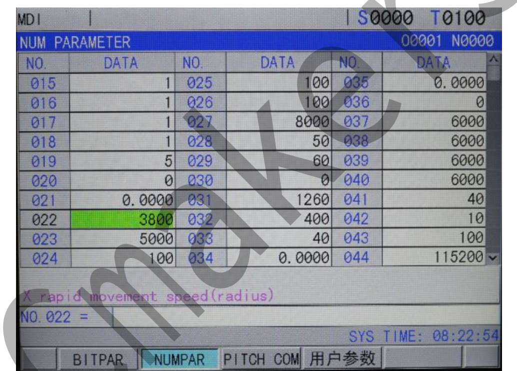
As is shown in this window, there are 2 parameter rows at the window bottom, the 1st row shows the meaning of a bit of a parameter where the cursor locates, the bit to be displayed can be positioned by pressing  or  key. The 2nd row shows the abbreviation of all the bits of a parameter where the cursor locates.



2) DATA PARAMETER interface

Press  key repeatedly ( key if in BIT PARAMETER interface), it enters DATA PARAMETER interface, and they can be viewed or altered by pressing  or  key to enter the corresponding window. It is shown below:

As is shown in this window, there is a cue line at the window bottom, it displays the meaning of the parameter where the cursor locates.



3) SCREW-PITCH COMP interface

Press  key repeatedly, it enters SCREW-PITCH COMP interface, there are 256 screw-pitch parameters which are displayed by 11 windows in this interface, and they can be viewed by pressing  or  key.

Volume II Operation

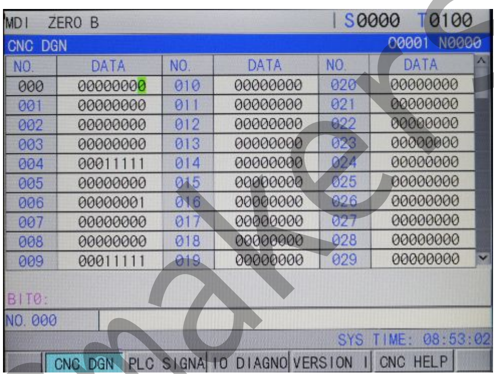
1.3.7 CNC DIAGNOSIS, PLC STATE, MACHINE SOFT PANEL, VERSION MESSAGE,HELP MESSAGE interfaces

DIAGNOSIS is a compound key, it enters CNC DIAGNOSIS, PLC STATE, PLC VALUE, TOOL PANEL, VERSION MESSAGE interfaces by pressing this key repeatedly.

1) CNC DIAGNOSIS interface

The input/output signal state between CNC and machine, the transmission signal state between CNC and PLC, PLC internal data and CNC internal state can all be displayed via diagnosis. Press DIAGNOSIS key it enters CNC DIAGNOSIS interface, the keypad diagnosis, state diagnosis and miscellaneous function parameters etc. can be shown in this interface, which can be viewed by pressing or key.

In CNC DIAGNOSIS window, there are two diagnosis rows at the window bottom, the 2nd row shows the meaning of a bit diagnosis No. where the cursor locates, the bit to be displayed can be positioned by pressing or key. The 1st row shows the abbreviation of the bit diagnosis number where the cursor locates.

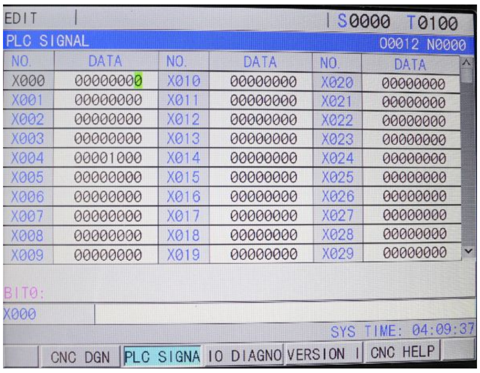


2) PLC STATE interface

In the window of this interface, it orderly displays the state of address X0000~X0063, Y0000~Y0047, F0000~F063, G0000~G063 etc.. And it enters PLC STATE interface by pressing DIAGNOSIS key repeatedly. The signal state of PLC addresses can be viewed by pressing or key.

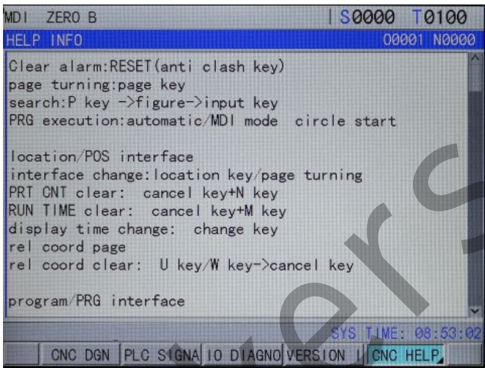
In PLC STATE window, there are 2 rows at the window bottom, the 2nd row shows the meaning of a bit of an address where the cursor locates, the bit to be displayed can be positioned by pressing or key. The 1st row shows the abbreviation of the bit address number where the cursor locates.

Volume Ⅱ Operation



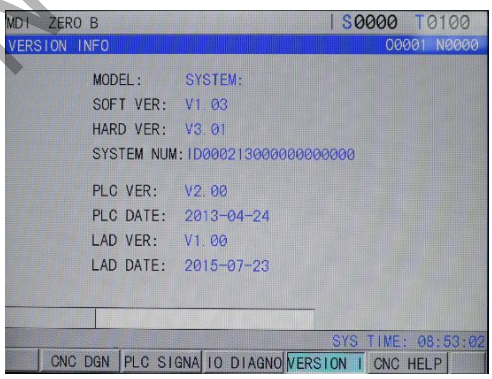
3)HELP MESSAGE interface

It enters HELP MESSAGE interface by pressing key repeatedly. The operation list, alarm list, G command list ,and macro command message can be shown in this interface. As is shown in the following figure:



4) VERSION MESSAGE interface

It enters VERSION MESSAGE interface by pressing  key repeatedly. The software, hardware, and PLC version message can be shown in this interface. As is shown in the following figure:



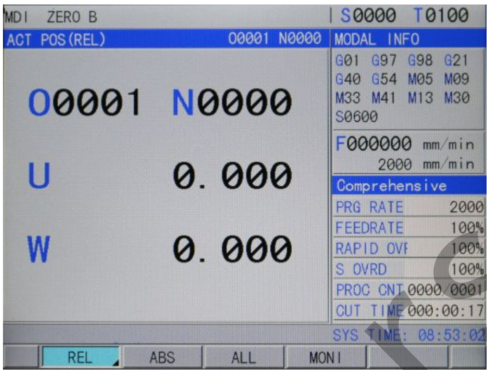
CHAPTER 2 POWER ON/OFF AND PROTECTION

2.1 System power on

Before C1000T power on, the following items should be confirmed:

1. The machine is in a normal state.
2. The power voltage conforms to the requirement of the machine.
3. The connection is correct and secure.

The current position (RELATIVE POS) window is displayed after C1000T automatic detection and initiation are finished.



2.2 System power off

Before power is off, ensure that:

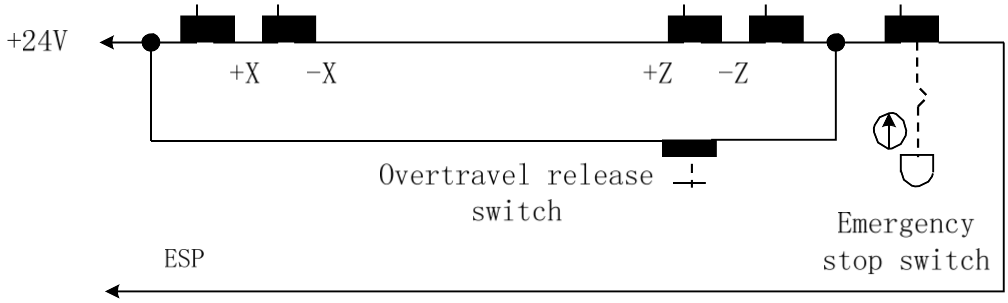
1. The feed axes of the CNC is at stop;
2. Miscellaneous functions (spindle, cooling etc.) are OFF;
3. Cut off CNC power prior to machine power cutting off.

Note: Please refer to the machine manufacturer manual about cut-off the machine power.

2.3 Overtravel protection

2.3.1 Hardware overtravel protection

The stroke switches are fixed at the positive and negative maximum travel of the machine X, Z axis respectively, they are connected by the following figure. And the BIT3(ESP) of bit parameter No.215 must be set to 0. If the overtravel occurs, the stroke switch acts to make C1000T stop, and the emergency alarm is issued.



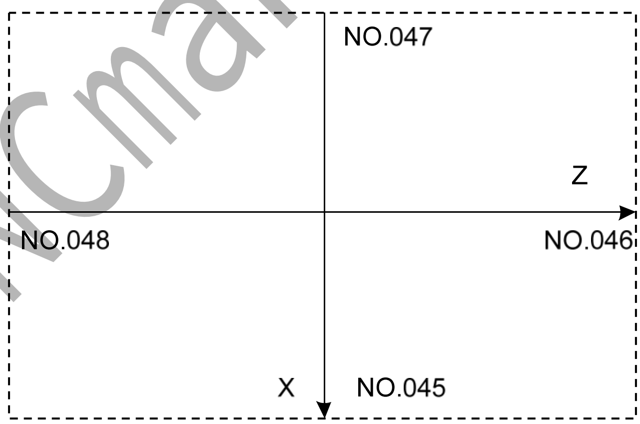
When the hardware overtravel occurs, there will be an "emergency stop" alarm in C1000T. The steps to eliminate this alarm are: press the OVERTRAVEL button to switch to the ALARM window, view the alarm message, and reset the alarm and move the table reversely to detach the stroke switch (for positive overtravel, move negatively; vice versa) .

2.3.2 Software Overtravel Protection

When the Bit4 of bit parameter No.172 is set to 0, the software limit is active.

X, Z axis

The software strokes are set by data parameter No.045, No.046, No.047, No.048, they refer to the machine coordinates. As follows figure shows, X, Z are the machine coordinate system axes;No.045, No.047 are for X axis positive and negative strokes, No.046, No.048 are for Z axis positive and negative strokes, within the broken line is the software stroke scope.



If the machine position (coordinate) exceeds the area within broken line, overtravel alarm will be issued. The steps to eliminate this alarm are: press RESET key to clear the alarm, then moves reversely (for positive overtravel, move out negatively; vice versa).

Additional axis

The software stroke is set by data parameter No.192, No.195, which is referred to the machine coordinates. No.192 is for Y positive stroke, No.195 is for Y negative stroke.

The software stroke is set by data parameter No.193, No.196, which is referred to the machine coordinates. No.193 is for 4^{th} axis positive stroke, No.196 is for 4^{th} axis negative stroke.

The software stroke is set by data parameter No.194, No.197, which is referred to the machine coordinates. No.194 is for 5th axis positive stroke, No.197 is for 5th axis negative stroke.

2.4 Emergency operation

During the machining, some unexpected incidents may occur because of the user programming, operation and product fault etc. So this C1000T should be stopped immediately for these incidents. This section mainly describes the resolutions that the system is capable of under the emergency situation. Please see the relative explanation on these resolutions under the emergency by machine builder.

2.4.1 Reset

Press 

key to reset C1000T system when there are abnormal output and axis actions:

1 All axes motion stops;

2 M, S function output is inactive (which can be set by parameter whether automatically cut off signals such as spindle CCW/CW, lubricating, cooling by pressing 

key, defined by PLC ladder);

3 Automatic run ends, modal function and state are held on.

2.4.2 Emergency stop

During machine running, if the emergency button is pressed under the dangerous or emergent situation (external SP signal active), the CNC system enters into emergency status and the machine movement is stopped immediately. All the outputs such as the spindle running, cooling are cut off. If the emergency button is released, the emergency alarm is cancelled and the CNC resets. Its circuit wiring is shown in **Section 2.3.1** of this Chapter.

Note 1: Ensure the fault is eliminated before the emergency alarm is cancelled.

Note 2: Pressing down the Emergency button prior to power on or off may alleviate the electric shock to the machine system.

Note 3: Re-perform the machine zero return to get the correct position coordinate after the emergency alarm is cancelled (machine zero return is forbidden if there is no machine zero on the machine.).

Note 4: Only Bit2(ELAM) of the bit parameter No.215 is set to 0, is the external emergency stop active.

2.4.3 Feed hold

key can be pressed during the machine running to make the running to pause. But in threading, cycle running, this function can not stop the running immediately.

2.4.4 Power-off

Under the dangerous or emergency situations during the machine running, the machine power should be cut off immediately to avoid the accidents. But it should be noted that there may be a large error between the CNC coordinates displayed and the actual position. So the toolsetting operation should be performed again.

CHAPTER 3 MANUAL OPERATION

Note!

The key functions of this C1000T machine panel are defined by PLC program (ladders), please refer to the manual from the machine builder for their function significance.

Please note that the following functions for the machine panel keys are described based on theCNCmakers Limited standard PLC programs!

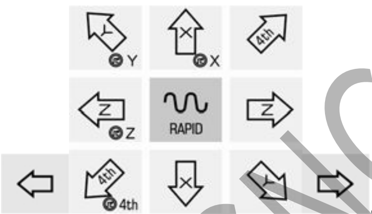
Press MANUAL key, it enters Manual mode. In this mode, the manual feed, spindle control, override adjustment, tool change etc. operations can be performed.

3.1 Coordinate axis move

In Manual mode, 2 coordinate axes can be moved manually for feeding and rapid traverse.

3.1.1 Manual feed

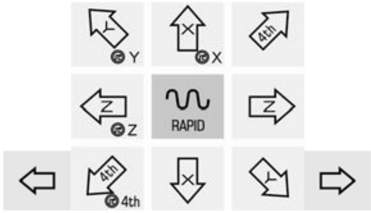
Press and hold X axis feed and direction key or in the selection area



, X feeds negatively or positively, and its feeding stops if the key is released; press and hold Z axis feed and direction key or , Z feeds negatively or positively, and its feeding stops if the key is released; press and hold Y axis feed and direction key or , Y feeds negatively or positively, and its feeding stops if the key is released; press and hold the 4th axis feed and direction key or , the 4th axis feeds negatively or positively, and its feeding stops if the key is released.

In Manual mode, press RAPID key to make the indicator in the panel state area to light up, and it enters the manual rapid traverse mode.

3.1.2 Manual rapid traverse

Press and hold the RAPID key in the direction selection area  till the rapid traverse indicator in the panel state area lights up. X can rapidly move negatively or positively by pressing the axis direction key X or, and the axis moving stops if the key is released; press and hold Z axis feed and direction key Z or, Z can be rapidly moved negatively or positively, and Z moving stops if the key is released; press and hold Y axis feed and direction key Y or, Y can rapidly move negatively or positively, and Y moving stops if the key is released; press and hold the 4th axis feed and direction key 4th or, the 4th axis can rapidly move negatively or positively, and the 4th axis moving stops if the key is released.

In Manual rapid mode, press RAPID key to make the indicator to go out, and the rapid traverse will be inactive, it enters the manual feed mode.

Note 1: If no reference point return is performed after power on, as the rapid traverse switch is turned on (rapid indicator lighting up), the manual feedrate or rapid rate for the traverse is defined by the Bit0(ISOT) of the bit parameter No.012 of this C1000T system.

Note 2: In Edit/MPG mode, RAPID key is inactive.

3.1.3 Speed tune

In Manual mode, 

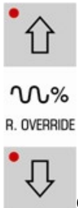
or can be pressed to alter the manual feedrate override that has 16 steps. The relation of the override and the feedrate is as follows table if data parameter No.031 is set to 1260:

Feedrate override (%)	Feedrate (mm/min)
0	0
10	126
20	252
30	378
40	504
50	630
60	756

Volume II Operation

Feedrate override (%)	Feedrate (mm/min)
70	882
80	1008
90	1134
100	1260
110	1386
120	1512
130	1638
140	1764
150	1890

Note : There is about 2% error for the data in the above table.

In the manual rapid traverse, it can press 

or 

key to modify the rapid override, and there are 4 steps of F0, 25%, 50%,100% for the override.(F0 set by data parameter No.032)

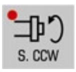
The rapid override is active under the following conditions:

(1) G00 rapid traverse (2) Rapid traverse in canned cycle

(3) Rapid traverse in G28 (4) Manual rapid traverse

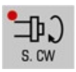
3.2 Other manual operations

3.2.1 Spindle CCW, CW, stop control



: In Manual mode, the spindle rotates counterclockwise if pressing this key;

: In Manual mode, the spindle stops if pressing this key;



: In Manual mode, the spindle rotates clockwise if pressing this key.

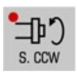
3.2.2 Spindle jog

: At the moment, the spindle is in JOG state.

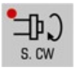
Functional description:

Press 

to enter JOG mode, and the spindle JOG function ON/OFF is executed only when the spindle is in the state of stop.

In spindle JOG mode, by pressing 

key, the spindle rotates counterclockwise for jogging;

by pressing 

key, the spindle rotates clockwise for jogging. The jog time and speed are set by data parameter No.108 and No.109 respectively.

When the spindle JOG rotates, 

is pressed to stop the spindle JOG rotation, the spindle brake signal is not output when the JOG rotation stops.

K10.4 is set to 1, the spindle JOG is valid in any mode. In Auto or MDI mode, the spindle is in the JOG rotation state, the program closes the spindle JOG rotation and the JOG function.

Parameter setting:

PLC parameter K104 1/0: the spindle JOG is valid in any mode/Manual, MPG, Zero return mode.

Data parameter No.108: spindle JOG time

Data parameter No.109: rotary speed in spindle JOG.

3.2.3 Cooling control

: In Manual mode, press this key, the cooling is switched on/off.

Parameter setting: PLC parameter K10.1 1/0: the spindle lubricating and cooling output remains/closes in reset.

3.2.4 Lubricating control

Function description:

1. Non-automatic lubricating

DT17 =0: For non-automatic lubricating

When data parameter No.112 is 0, it is lubricating turn output, by pressing the 

key, the lubricating is output. And the lubricating is cancelled by pressing it again. M32 is for lubricating output, and M33 is for lubricating output cancellation.

When data parameter No.112 ＞1, it is timing lubricating output, by pressing the 

key, the lubricating is output. And it is cancelled after a setting time by data parameter No.112; by executing M32, the lubricating is output, and it is cancelled after a setting time by data parameter No.112. If the setting time is not yet up, M33 is executed to cancel the lubricating output.

2. Automatic lubricating

DT17>0: For automatic lubricating, the lubricating time DT17 and lubricating interval time DT16 may be set.

After C1000T system is switched on, it is lubricating for a time set by DT17, then the lubricating output stops. After an interval set by DT16, the lubricating is output again, so it cycles by sequence.

In the automatic lubricating, M32, M33 codes as well as the 

key are all inactive.

Parameter setting:

PLC parameter: K10.1 1/0: the spindle lubricating/cooling output remains/closes in reset.

PLC parameter:K16.2 1/0: whether the lubricating outputs in power-on when the automatic lubricating is valid.

PLC data: DT16 automatic lubricating interval time (ms)

PLC data: DT17: automatic lubricating output time (ms)

Data parameter: No.80: M execution duration(ms)

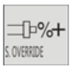
Data parameter: No.112: lubricating start time (0-60000ms)(0:lubricating time is not limited)

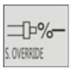
3.2.5 Manual tool change

: In Manual mode, by pressing this key, the tools are changed manually by sequence (if current tool is No.1, by pressing this key, it is changed for No.2 tool; if current tool is No.4, by pressing this key, it is changed for No.1 tool.

3.2.6 Spindle override

In Manual mode, if the spindle speed is controlled by analog voltage output, the spindle speed may be overrided.



By pressing 

in Spindle Override keys, the spindle speed can be changed by real-time adjusting of the spindle override that has 8 steps of 50％～120％.

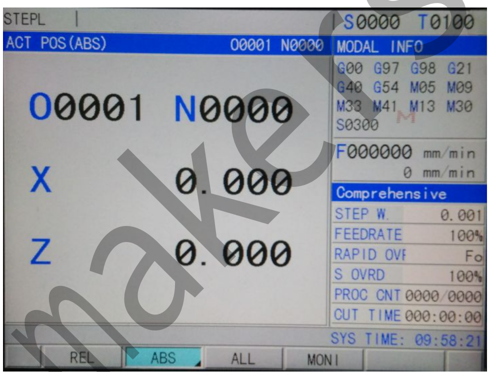
CHAPTER 4 MPG/STEP OPERATION

In MPG/Step mode, the machine moves by a specified increment.

> **Note!**
>
> The key functions of this C1000T machine panel are defined by PLC program (ladders), please refer to the manuals by the machine builder for their significance.
>
> Please note that the following description for the key functions in this chapter is based on the C1000T standard PLC program!

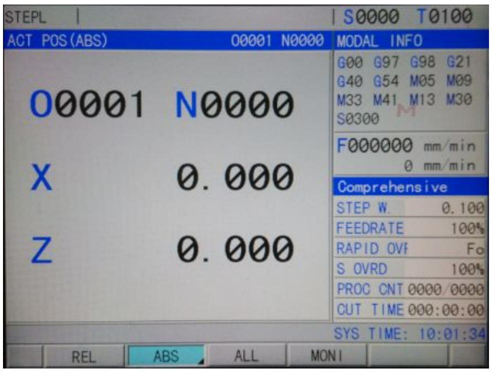
4.1 Step feed

Set the system parameter No.001 Bit3 to 0, and press MPG key to enter the STEP working mode, it displays as follows:



4.1.1 Increment selection

Press X1 F0, X10 25%, X100 50%, X1000 100% to select the move increment, the increment will be shown in the window. When the BIT7(SINC) of the bit parameter K016 is 1, step X1000 100% is inactive; when the BIT7 is 0, X1 F0, X10 25%, X100 50%, X1000 100% are all active. For example, to press X100 50% key, the window is shown as follows:

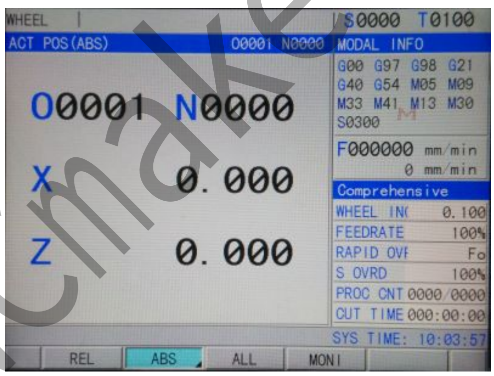


4.1.2 Moving direction selection

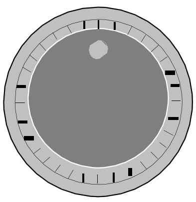
Pressing or key once can move X negatively or positively by a step increment; pressing or key once can move Z negatively or positively by a step increment; pressing or key once can move Y negatively or positively by a step increment.

4.2 MPG(handwheel) feed

Set the BIT3 of the system parameter No.001 to 1, and press key to enter the MPG mode, it displays as follows:



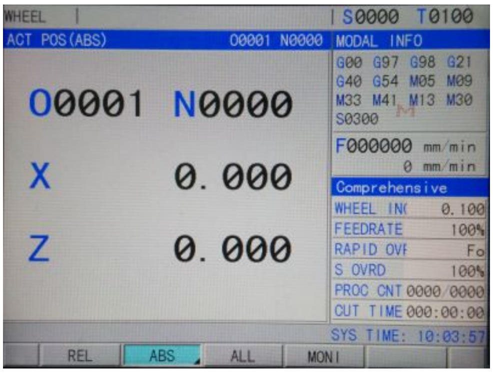
The handwheel figure is as follows:



The handwheel (MPG) figure

4.2.1 Increment selection

Press key to select the move increment, the increment will be shown in the window. When the BIT7(SINC) of PLC K016 is 1, step is inactive; when BIT7 is 0, are all active. For example, to press key, the window is shown as follows:



4.2.2 Moving axis and direction selection

In MPG mode, press X, Y, Z, 4th key, the corresponding axis will be selected.

The MPG feed direction is defined by its rotation direction. Generally, the handwheel CW is for positive feed, and CCW for negative feed. In case of that handwheel CW is for negative feed, CCW for positive feed, it may exchange the A, B signals of the handwheel terminals. No. 013 Bit0 ~ Bit4 selects the feed direction of MPG rotation.

4.2.3 Other operations

1） Spindle CCW, CW, stop control

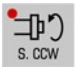
S. CCW: In Manual/Step mode, the spindle rotates counterclockwise if pressing this key;

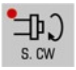
S. STOP: In Manual/Step mode, the spindle stops if pressing this key;

S. CW: In Manual/Step mode, the spindle rotates clockwise if pressing this key.

2） Spindle Jog

: at the moment, the spindle is in JOG working mode.

In spindle Jog mode, by pressing 

key, the spindle rotates counterclockwise for jogging; by pressing 

key, the spindle rotates clockwise for jog. The jogging time and speed are set by data parameter No.108 and No.109 respectively. The concrete is referred to Chapter 3.2.2.

3） Cooling control

Refer to OPERATION, Chapter 3.2.3

4） Lubricating control

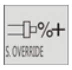
Refer to OPERATION, Chapter 3.2.4

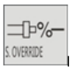
5) Manual tool change

: In MPG/Step mode, press it to execute the tool change orderly.

6) Spindle override tune

In MPG/Step mode, if the spindle speed is controlled by analog voltage output, the spindle speed may be overrided.

By pressing 



in Spindle Override keys, the spindle speed can be changed by real-time adjusting of the spindle override that has 8 steps of 50％ ~ 120％.

4.2.4 Explanation items

The correspondence of the handwheel scale to the machine moving amount is as follows table:

	Moving amount of each MPG scale			
MPG increment	0.001	0.01	0.1	1
Specified coordinate value	0.001mm	0.01mm	0.1mm	1mm

（Taking example of the least input increment 0.001mm）

Note 1: The MPG increment is related to the system's current metric/inch input state and the system's least input increment.

Note 2: The MPG speed cannot be more than 5r/s, otherwise, the scale value is inconsistent with the movement amount.

CHAPTER 5 MDI OPERATION

In MDI mode, the operations of parameter setting, code words input and execution can be performed.

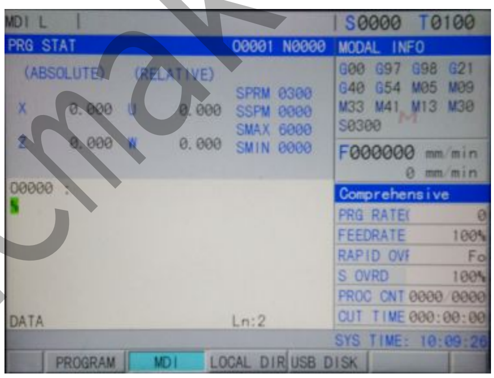
> **Note!**
>
> The key functions of this C1000T machine panel are defined by PLC program (ladders), please refer to the manuals by the machine builder for their significance.
>
> Please note that the following description for the key functions in this chapter is based on the C1000T standard PLC program!

5.1 Code words input

Select MDI mode to enter the PRG STATE window, to input an block “G50 X50 Z100” , the steps are as follows:

1) Press MDI key to enter MDI mode;

2) Press PROGRAM key (or or press PROGRAM many times if needed) to enter PRG STATE window:



3) press address key G, number key 5, 0 ;

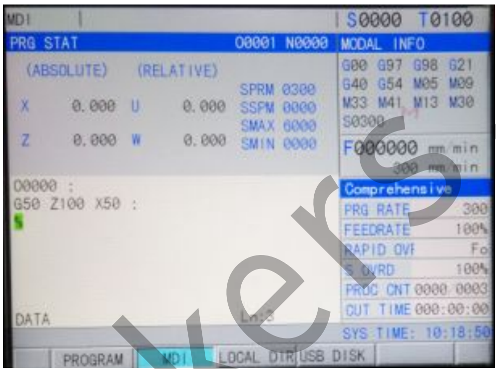
4) press address key Z, number key 1, 0, 0;

5) press address key X, number key 5, 0;

The window is shown as follows after above operations are completed(can input 4 block and display 6 block):

5.2 Code words execution

After the word is input, press DATA INPUT and the display is as follows:



After the code words are input and DATA INPUT is pressed to display the block, CYCLE START key is pressed to execute the input block. During the execution, FEED HOLD, RESET and Emergency Stop button may be pressed to terminate these code words execution.

Note: The subprogram call codes (M98 P_ ; etc.), compound cutting cycle codes(G70, G71, G72, G73, G74, G75, G76 and so on) is inactive in MDI mode.

5.3 Parameter setting

In MDI mode, the parameter value can be altered after entering the parameter interface. The detailed is referred to **OPERATION, Chapter 10**.

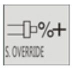
5.4 Data alteration

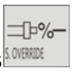
In the PRG STATE window of MDI mode, if there is an error during words inputting, CANCEL is pressed to cancel the display or RESET is pressed to clear all the input, then re-input the correct ones.

5.5 Other operations

`. Spindle override available

In MPG/Step mode, when the spindle speed is controlled by analog voltage output, the spindle speed may be tuned.



press 

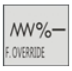
, the spindle speed can be changed by real-time adjusting of the spindle override that has 8 levels of 50％ ~ 120％.

2. Rapid override is available.

Press 

to tune the rapid traverse rate which has 4 levels.

3. Feedrate override is available.

In MDI mode, by pressing 

, the actual speed real-time adjusting of 0 ~ 150％ feedrate by F code can be done by the override that has 16 levels from 0％ to 150％.

CHAPTER 6 PROGRAM EDIT AND MANAGEMENT

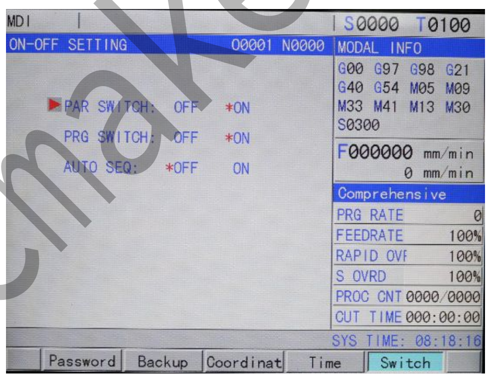
In Edit mode, a program can be created, selected, altered, copied and deleted, and the bidirectional communication of CNC to CNC, or CNC to PC can also be done. To prevent the program to be altered or deleted accidentally, a program switch is set up for this C1000T system. And it must be turned on before program editing. Please see details in Section 10.1.1 of this part. Also 3 level user passwords are set in this C1000T system to facilitate the management. Only the operation level above 4 is authorized (4th or 3rd level etc.) can the program switch be opened for program editing. See **OPERATION, Section 10.3 .**

6.1 Program creation

6.1.1 Creating a block number

In the program, the block number can be added or not, the program is executed by the sequence. When the “AUTO SEG” switch in SWITCH SETTING window is OFF, CNC doesn’t generate the block number automatically, but the blocks may be numbered manually.

When the “AUTO SEG” switch in SWITCH SETTING window is ON, CNC generates the block number automatically, it automatically generates the next block number by pressing EOB key in editing. The block number increment is set by the CNC data parameter No.042. (See details in **Section 10.1.1** of this part.)

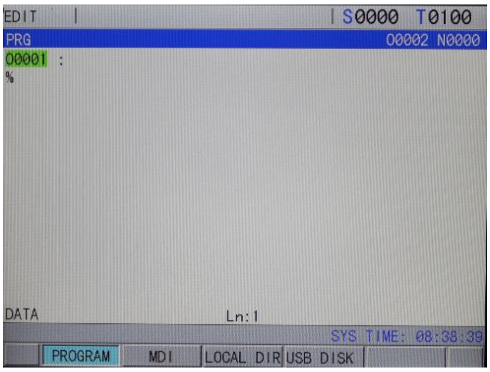


6.1.2 Inputting a program

1. Press EDIT key to enter the Edit mode;

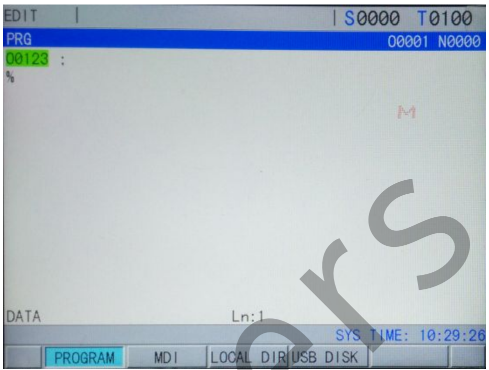
Press PROGRAM key to enter the Program interface, select the PRG CONTENT window by pressing  or  key;

Volume Ⅱ Operation



2. Press address key O, number key O, O, O and 1 key by sequence (e.g. program O0001 creation);

3. Press EOB key to create the new program;



4. Input the edited part program one by one, the character will be displayed on the screen immediately as it is input(as for compound key, press this key repeatedly for alternate input), after a block is finished, press EOB key to terminate it.

5. Other blocks input may be finished by step 4 above.

Note: The unexpected power-off when the program is input, the program being edited cannot be saved.

6.1.3 Movement of cursor

1) Press EDIT key to enter the Edit mode, then press PROGRAM key to enter the PRG CONTENT window;

2） Press ⇧ key, the cursor shifts a row upward; if the number of the column where the cursor locates is over the total columns of the previous row, the cursor moves to the previous block end (at “;” sign) after ⇧ key is pressed;

3) Press ⇩ key, the cursor shifts a row downward; if the number of the column where the cursor locates is over the total columns of the next row, the cursor moves to the next block end (at “;” sign) after the ⇩ key is pressed;

4) Press ⇨ key, the cursor shifts a column to the right; if the cursor locates at the row end, it moves to the head of the next block;

5) Press key, the cursor shifts a column to the left; if the cursor locates at the row head, it moves to the end of the next block;

6) Press key to window upward

7) Press key to window downward,

6.14.Searching a character and line No.

Searching a character: To search for the specified character upward or downward from the cursor current location

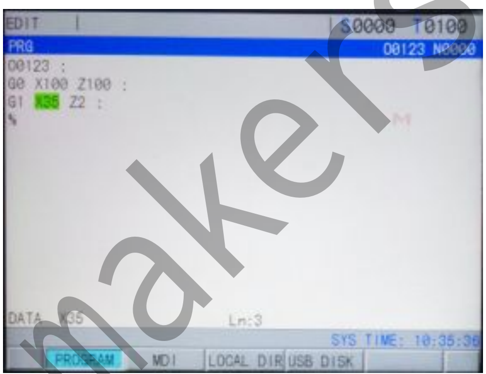
The steps of finding is as follows:

1) Press EDIT key to enter Edit mode;

2) Press PROGRAM key to enter the PRG CONTENT window;

3) Input the characters to be searched , the characters over the 10th byte will be ignored.

4) Press key (or is determined by the location of the character searched to the character where the cursor locates), it displays as follows:



5) After the finding, the CNC system is still in FIND state, press or key again, the next character can be found.

Note 1: If the character is not found, the searching character will disappear

Note 2: During the searching, it doesn't search the characters in the called subprogram, and the character in subprogram is searched in subprogram.

Note 3: The system cannot search and scan the character in macro edit mode.

Searching a line:Put the cursor rapidly move to specified line of program

The steps of finding is as follows:

1) Press EDIT key to enter Edit mode;

2) Press PROGRAM key to enter the PRG CONTENT window;

3) Input the line No. to be searched

Volume II Operation

3) Press input key,the cursor will skip to specified line

6.1.5 Inserting a character

Steps:

1) Select the PRG CONTENT window in Edit mode;

2) Press INSERT ALTER key to enter the INS mode (the cursor is an underline), the window is as follows:

3) Input the character to be inserted

6.1.6 Deleting a character

Steps:

1) Enter the PRG CONTENT window in Edit mode;

2) Move the cursor to the location where you want to delete, press DELETE key to delete the character where the cursor locates;

6.1.7 Altering a character

Steps:

1) Enter the PRG CONTENT window in Edit mode;

2) Move the cursor to the location where you want to alter, press INSERT ALTER key to alter the character instead of the input content

6.1.8 Deleting a single block

Steps:

1) Select the PRG CONTENT window in Edit mode;

2) Move the cursor to the head of the block to be deleted (column 1), press N# key,

And then press DELETE key to delete a single block

6.1.9 Deleting blocks

Steps:

1) Select the PRG CONTENT window in Edit mode;

2) Move the cursor to head line of a block that you want to delete

3) Input the sequence No. of the last block that you want to delete

4) Press DELETE key, the blocks among cursor and marked address will be deleted

6.1.10 Deleting a segment

Steps:

1) Select the PRG CONTENT window in Edit mode;
2) Move the cursor to the 1st character of a block that you want to delete
3) Input the last character of the block that you want to delete
4) Press DELETE key, the segment among cursor and marked address will be deleted

6.1.11 Copying a single block

Steps:

1)Select the PRG CONTENT window in Edit mode;
2) Move the cursor to head line of a block that you want to copy
3)Press N# key first, then press CHANGE ,copy the block where the cursor located in

6.1.12 Copying blocks

Steps:

1)Select the PRG CONTENT window in Edit mode;
2) Move the cursor to 1st character of a block that you want to copy
3) Input the sequence No. of the last block that you want to copy
4) Press CHANGE key, the blocks among cursor and inputed character will be copyed

6.1.13 Deleting a segment

Steps:

1) Select the PRG CONTENT window in Edit mode;
2) Move the cursor to the 1st character of a block that you want to delete
3) Input the last character of the block that you want to delete
4) Press CHANGE key, the segment among cursor and inputed character will be copyed

6.1.14 Pasting a single block

Steps:

1)Select the PRG CONTENT window in Edit mode;
2) Move the cursor to location of program that you want to paste
3)Press DATA OUTPUT key , insert the last copy program content before cursor to finish paste operation

6.2 Deleting program

6.2.1 Deleting a program

Steps:

1) Select the PRG CONTENT window in Edit mode;

2) Press address key O, number key 0, 0, 0, 1 by sequence (by program O0001);

3) Press DELETE key, program O0001 will be deleted.

6.2.2 Deleting all programs

Steps:

1) Select the PRG CONTENT window in Edit mode;

2) Press address key O, symbol key -+ >, number key 9, 9, 9, 9 ;

3) Press DELETE key, all the programs will be deleted.

6.3 Selecting a program

When there are multiple programs in CNC system, a program can be selected by the following 3 methods:

6.3.1 Search

1) Select Edit or Auto mode;

2) Press PROGRAM key to enter the PRG CONTENT window;

3) Press address key O and key in the program No.;

4) Press ⇩ or EOB key, or press ⇩ key in Auto mode, the searched program will be displayed. If the program does not exist, an alarm will be issued by CNC.

Note: In step 4, if the program does not exist in Edit mode, a new program will be created by CNC system after EOB key is pressed.

6.3.2 Scanning

1) Select Edit or Auto mode;

2) Press PROGRAM key to enter the Program window;

3) Press address key O;

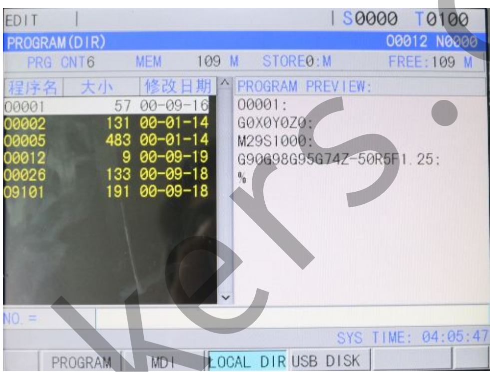
4) Press ⇩ or ⇧ key to display the next or previous program;

5) Repeat step 3 and 4 to display the saved programs one by one.

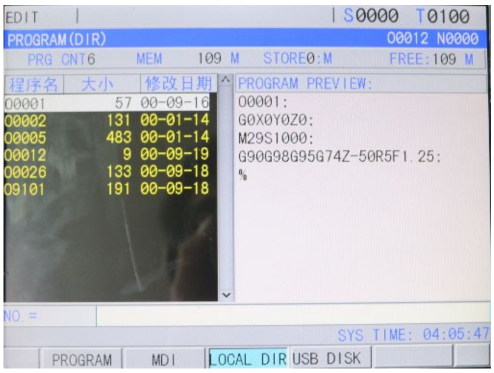
6.3.3 Cursor

1) Select Auto mode **(must be in non-running state)**;

2) Press PROGRAM key to enter the PRG LIST window;



3) Press ⇧, ⇩, ⇦ or ⇨, , key to move the cursor to the program name to be selected ("PRG SIZE" "NOTE" content changed as the cursor moves);



4) Press EOB key or DATA INPUT.

6.4 Renaming a program

1) Select the PRG CONTENT window in Edit mode;

2) Press address key O, and key in the new program name;

3) Press INSERT ALTER key.

6.5 Copy a program

To save the current program to a location:

1) Select the PRG CONTENT window in Edit mode;

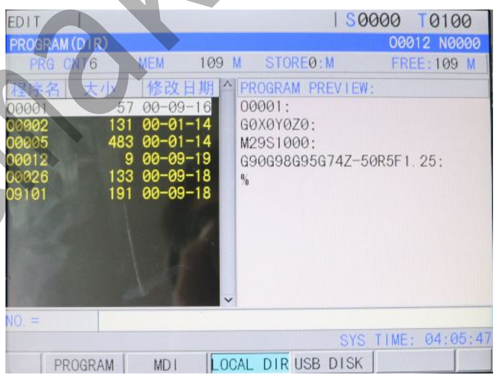
2) Press address key O, and key in the new program No.;

3) Press CHANGE key.

6.6 Program management

6.6.1 Program list

In non-Edit mode, press PROGRAM key to enter the PRG LIST window. In this window, it lists the program names saved in CNC system, and it can display maximum 10 names in a window, if the programs saved exceed 10, it may press 、 key to display the other program list.



1)Open program

Open specified program : O + No.+ EOB

In Edit mode, it will creat program if program No. is not exit.

2)Delete program

1. In Edit mode, press DELETE to delete program specified by cursor

2.In Edit mode,press O + No. + DELETE or No. + DELETE

6.6.2 Part-Prg number

It shows the total number of the part programs (up to 400) that can be saved in CNC system and the current part programs number that have been saved at present.

6.6.3 Memory size and used capacity

They show the total memory capacity (56M) of the CNC and the current capacity that has been occupied.

CHAPTER 7 TOOL OFFSET AND SETTING

The actual location of tool can be overlooked in programming for simplifying programming. Three methods including positioning tool setting, trial tool setting and machine zero tool setting are available in this C1000T system. The tool offset data are obtained from this tool setting operation.

7.1 Tool positioning setting

Steps:

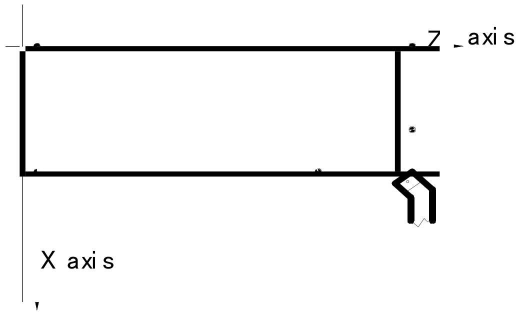


Fig. 7-1

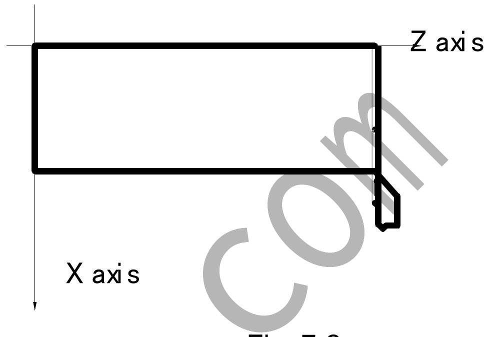


Fig. 7-2

1. Firstly determine if the offset values are zero in X, Z, if not, clear all the tool number offset values;

2. Set the offset No. for 00 (i.e. T0100,T0300), as for the offset value: (method: execute a move code or perform the machine zero return in T0100 state, then clear the offset value automatically as returning to the machine zero)

3. Select a tool by random (usually the 1st tool, this tool will be used as the reference tool);

4. Position the tool nose of the reference tool to a point (toolsetting point), as shown in Fig. 7-1;

5. In PRG STATE window of the MDI mode, set up the workpiece coordinate system by the command G50 X__ Z__;

6. Clear the relative coordinate (U, W), see details in OPERATION, Section 1.3.1;

7. After the tool is moved to a safety height, select another tool and move it to the setting point, as shown in B;

8. Press 

key and move the cursor by ,  key to select the corresponding offset number of that tool;

9. Press address key 

, then press 

key to input the tool offset value of X axis into the corresponding offset number;

10. Press address key 

, then press 

key to input the tool offset value of Z axis into the corresponding offset number;

11. Repeat the steps from 7 to 10 to perform the tool setting operation for other tools.

Note: The original system tool offset should be cleared in positioning tool setting, multiple but one input of the new offset value by 

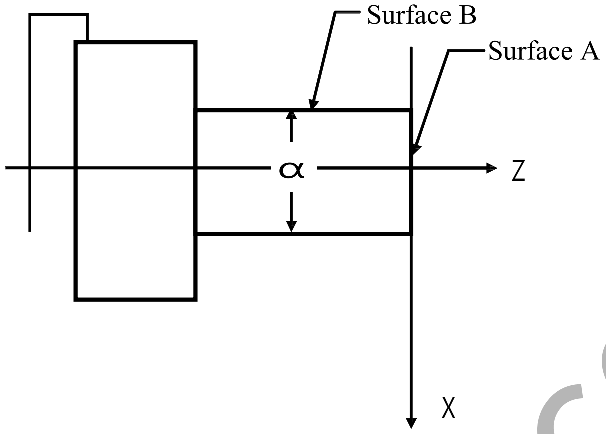
, 

keys are disabled, about the clearing ways, refer to Volume II Operation, Chapter 7.4.4.

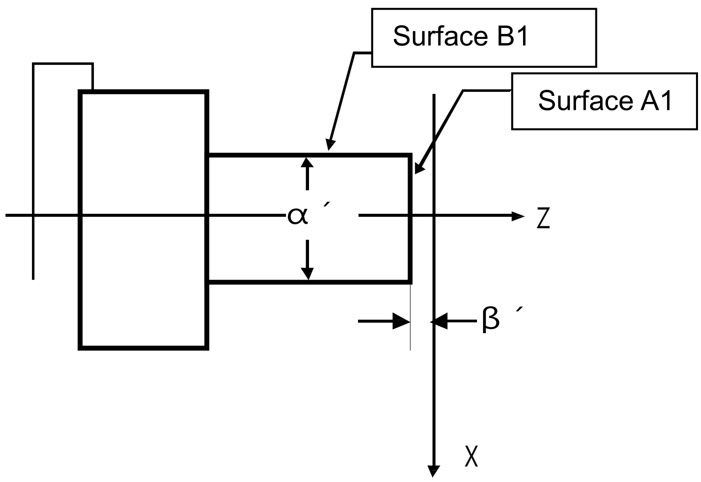
7.2 Trial tool setting

Whether the method of trial tool setting is inactive is defined by the system parameter No.012 Bit5.

Steps (workpiece coordinate system by part end surface):



1. Select a tool by random and make it cut on Surface A;
2. Retract the tool along X axis without Z axis moving and stop the spindle;
3. Press OFFSET key to enter the Offset interface, select the TOOL OFFSET window, then move the cursor by pressing ⇧ , ⇩ key to select the corresponding offset number;
4. Key in by sequence the address key Z , number key 0 and DATA INPUT key;
5. Make the tool cut along Surface B;
6. Retract the tool along Z axis without the movement of X axis, and stop the spindle;
7. Measure the diameter "α" (supposing α=15);
8. Press OFFSET key to enter the Offset interface, select the TOOL OFFSET window, then move the cursor by pressing ⇩ , ⇧ key to select the corresponding offset number;
9. Key in the address key X by sequence, number key 1 , 5 and DATA INPUT key;
10. Move the tool to a safety height to change for another tool;



11. Make the tool to cut on Surface A1;

12. Retract the tool along X axis without Z axis moving and stop the spindle;

13. Measure the distance "β´" between the Surface A1 and the workpiece coordinate origin(supposing β´=1) ;

14. Press OFFSET key to enter the Offset interface, select the TOOL OFFSET window, then move the cursor by pressing ⇩, ⇧ key to select the corresponding offset number;

key; 15. Key in by sequence the address key Z, sign key −+, number key − +, and DATA INPUT

16. Make the tool to cut on Surface B1;

17. Retract the tool along Z axis without the movement of X axis, and stop the spindle;

18. Measure the distance "α' " (supposing α'=10);

19. Press OFFSET key to enter the Offset interface, select the TOOL OFFSET window, then move the cursor by pressing ⇩, ⇧ key to select the corresponding offset number;

20. Press orderly the address key X, number key 1, 0 and DATA INPUT key;

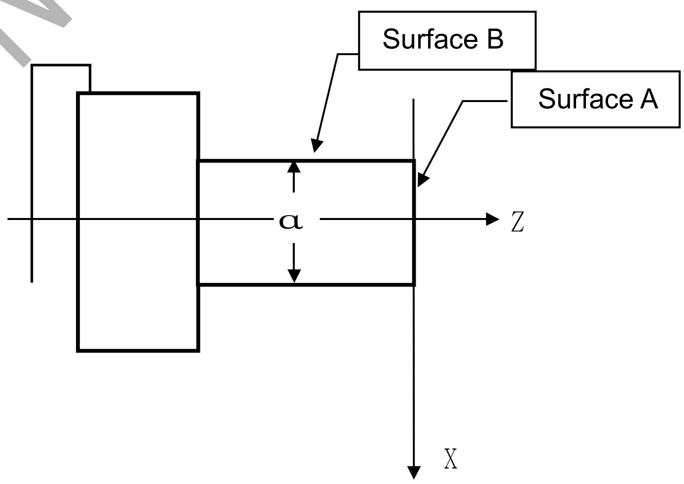
21. Repeat the execution from Step 10 to Step 20 to perform the tool setting operation for other tools.

Note: The offset value may be large by this tool setting method, so the tool compensation should be done by the coordinate offset by the CNC system. (set the BIT4 of the CNC parameter No.003 to 1). Moreover, the tool lengths compensation should be performed by using the T code in the 1st block, or the 1st move block should contain the T code for the tool length compensation.

7.3 Tool setting by machine zero return

There is no reference tool in this tool setting methods, when the tool is worn or to be adjusted, it only needs to be set again, and a machine zero return should be done before the tool setting. The machining could be continued by performing a machine zero return at power on after power-off, which is very convenient for the operation.

Steps (workpiece coordinate system by part end surface):



1. Press MACHINE ZERO key to enter Machine Zero mode, move axes to machine zero;

2. Select a tool by random and set the offset number of the tool to 00 (e.g. T0100, T0300) ;

3. Make the tool to cut on Surface A;

4. Retract the tool along X axis without the movement of Z axis, and stop the spindle;

5. Press OFFSET key to enter the Offset interface, select the TOOL OFFSET window, then move the cursor by pressing ⇩, ⇧ key to select the corresponding offset number;

6. Key in by sequence the address key Z, number key 0 and DATA INPUT key to set the offset value of Z axis;

7. Make the tool cut along Surface B;

8. Retract the tool along Z axis without the movement of X axis, and stop the spindle;

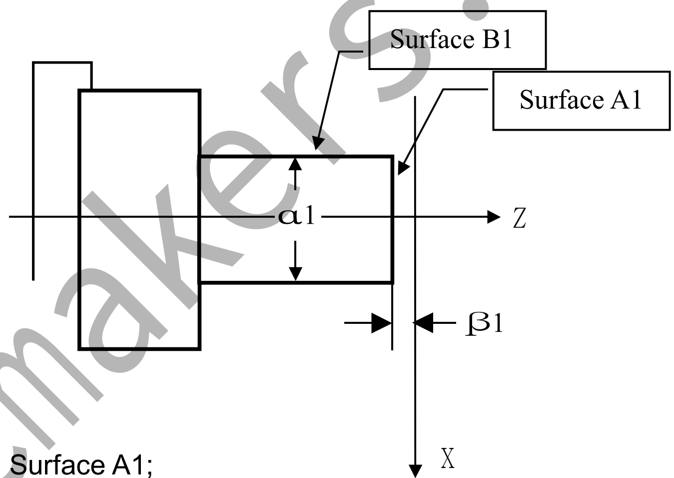
9. Measure the distance "α"(supposing α=15);

10. Press OFFSET key to enter the Offset interface, select the TOOL OFFSET window, then move the cursor by pressing ⇩, ⇧ key to select the corresponding offset number;

11. Key in by sequence the address key X, number key 1, 5 and DATA INPUT key to set the offset value of X axis;

12. Move the tool to a safety height for tool change;

13. Change for another tool, and set the tool offset number to 00 (i.e. T0100, T0300);



14. Make the tool to cut on Surface A1;

15. Retract the tool along X axis without Z axis moving and stop the spindle; measure the distance "β1" between the Surface A1 and the workpiece coordinate system origin(supposing β1=1) ;

16. Press OFFSET key to enter the Offset interface, select the TOOL OFFSET window, then move the cursor by pressing ⇩, ⇧ key to select the corresponding offset number;

17. Key in by sequence the address key Z, sign key −+ >, number key 1, and DATA INPUT key to set Z offset value;

18. Make the tool to cut on Surface B1;

19. Retract the tool along Z axis without the movement of X axis, and stop the spindle;

20. Measure the distance "α1" (supposing α1=10);

21. Press OFFSET key to enter the Offset interface, select the TOOL OFFSET window, then move the cursor by pressing ⇩, ⇧ key to select the corresponding offset number;

Volume II Operation

22. Key in by sequence the address key X , number key 1 , 0 and DATA INPUT key to set X offset value;

23. Move the tool to a safety position;

24. Repeat the execution from Step 12 to Step 23 to perform the tool setting operation for other tools.

Note 1: Machine zero switch must be fixed for the tool setting operation by machine zero return.

Note 2: The workpiece coordinate system setting can't be done by G50 code after tool setting by machine zero return.

Note 3: The tool compensation should be done by coordinate offset by the CNC system (the system parameter No.003 Bit4 is set to 1), further more, the tool lengths compensation should be performed by using the T code in the 1st block, or the 1st move block should contain the T code for the tool length compensation.

7.4 Setting and altering the offset value

Press OFFSET key to enter the Offset interface, it displays the offset numbers of No.000 ~ No.032 by pressing the ,  keys respectively.

Tool offset (2-axis)

Tool wear (2-axis)

7.4.1 Offset setting

1. Press OFFSET key to enter the OFFSET interface, select the desired window by pressing the ,  keys;

2. Move the cursor to the location of the tool offset, wear number to be input;

Volume II Operation

Scanning: Press ⇩ , ⇧ key to move the cursor in sequent

Searching: By following key sequence, it may move the cursor directly to a location to be keyed in P Q + offset number + DATA INPUT

3. After pressing the address key X or Z, the numerical number may be keyed in (decimal point allowed)

4. By pressing the DATA INPUT key, the CNC calculates the offset value automatically and displays the result in the window.

7.4.2 Offset alteration

1. By the method in OPERATION, Section 7.4.1, move the cursor to the location of the offset number that is to be altered;

2. If the offset value of X axis is to be altered, key in U; as for that of Z axis, key in W;

3. Then key in the incremental value;

4. Press the DATA INPUT key to add the current offset value to the value keyed in, the operation result will be displayed as a new offset value.

Example: The set X axis offset value is 5.678

The increment keyed in is U 1.5

Then the new offset value is 7.178 (=5.678+1.5)

7.4.3 Clearing the offset values

1. Move the cursor to the offset number to be cleared;

2. Method (1)

If the offset value of X axis is to be cleared, press X key, then press DATA INPUT key, this offset will be cleared;

If the offset value of Z axis is to be cleared, press Z key, then press DATA INPUT key, this offset will be cleared;

3. Method (2)

If the current offset in X axis is α, input U-α, then press DATA INPUT key, this offset in X axis will be cleared;

If the current offset in Z axis is β, input W-β, then press DATA INPUT key, this offset in Z axis will be cleared.

Volume II Operation

7.4.4 Setting and altering the tool wear

To prevent the mistaken operation of the setting and alteration of the offset value (decimal point missed, mislocated etc.), which may cause the tool collision by oversize offset value, for the visual judgement for the tool wear by the operator, the TOOL WEAR window is set in this C1000T system. When the offset value is needed to be altered due to the inaccurate dimensions by the tool wear, it may set or modify the wear value. The wear input range is defined by the data parameter No.140, and they are saved even at power down.

The setting and alteration methods for the tool wear are approximately identical to that of the tool offset, and the wear value is input by U(X axis), W(Z axis), V(Y axis).

7.4.5 No.0 tool offset moving workpiece coordinate system

When No.012 Bit 6 is set to 1, No. 0 tool offset moving workpiece coordinate system is valid.

After the value is input in No. 0 tool offset, the workpiece coordinate system executes the offset based on the input value.

Volume Ⅱ Operation

Before inputting No.0 tool offset

After inputting No.0 tool offset

As the above figure, after X100, Z100 in No.0 tool offset is input, the workpiece coordinate system offsets X100, Z100.

CHAPTER 8 AUTO OPERATION

Note！

The key functions of C1000T machine panel are defined by PLC program (ladders), please refer to the materials by the machine builder for their significance.

Please note that the following description for the keys function in this chapter is based on C1000T standard PLC program!

8.1 Automatic run

8.1.1 Selection of the program to be run

1. Searching method

1) Select the Edit or Auto mode;

2) Press PROGRAM key to enter the PRG CONTENT window;

3) Press the address key O, and key in the program No.;

4) Press ⇩ or EOB key, the program retrieved will be shown on the screen, if the program doesn't exist, an alarm will be issued.

2. Scanning method

1) Select the Edit or Auto mode;

2) Press PROGRAM key to enter the PRG CONTENT window;

3) Press the address key O;

4) Press the ⇩ or ⇧ key to display the next or previous program;

5) Repeat the step 3, 4 above to display the saved program one by one.

3. Cursor method

a) Select the Auto mode (in non-run state);

b) Press PROGRAM key to enter the PRG LIST window (press or key if needed);

c) Press ⇧, ⇩, ⇦ or ⇨ key to move the cursor to the name of the program to be selected;

d) Press EOB key.

8.1.2 Start of the automatic run

1. Press 

key to select the Auto mode;

2. Press 

key to start the program, and the program automatically runs.

Note: Since the program execution begins from the block where the cursor locates, before pressing the 

key, make a check whether the cursor is located at the block to be executed.

8.1.3 Stop of the automatic run

∗ Stop by code (M00)

1. M00

After the block containing M00 is executed, the auto run is stopped. So the modal function and state are all reserved. Press the 

key or the external run key, the program execution continues.

2. M01

Press 

and the optional stop indicator is ON and the function is valid. After the block with M01 is executed, the system stops the automatic run, the modal function and the state are saved. Press 

or the external run key, and the program continuously runs.

∗ Stop by a relevant key

1. In Auto run, by pressing 

key or external dwell key, the machine keeps the following state:

(1) The machine feed slows down to stop;

(2) The modal function and state are reserved;

(3) The program execution continues after pressing the 

key.

2. Stop by Reset key 

(1) All axes movement is stopped.

(2) M, S function output is inactive (the automatic cut-off of signals such as spindle CCW/CW, lubricating, cooling by pressing 

key can be set by the parameters)

(3) Modal function and state is held on after the auto run.

3. Stop by Emergency stop button

If the external emergency button (external emergency signal active) is pressed under the dangerous or emergent situation during the machine running, the CNC system enters into emergency state, and the machine moving is stopped immediately, all the output (such as spindle rotation, cooling) are all cut off. If the Emergency button is released, the alarm is cancelled and CNC system enters into reset mode.

4. Switching operation mode

When Auto mode is switched to the Machine zero, MPG/Step, Manual, Program zero mode, the current block "dwells" immediately; when the Auto mode is switched to the Edit, MDI mode in Auto mode, the “dwell” is not displayed till the current block is executed.

Note 1: Ensure that the fault has been resolved before cancelling the emergency alarm.

Note 2: The electric shock to the device may be decreased by pressing the Emergency button before power on and off.

Note 3: The Machine zero return operation should be performed again after the emergency alarm is cancelled to ensure the correctness of the position coordinates (but this operation is forbidden if there is no machine zero in the machine).

Note 4: Only the BIT2 (EALM) of the bit parameter No.215 is set to 0, could the external emergency stop be active.

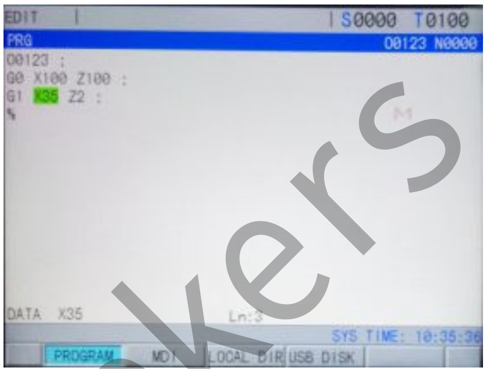
8.1.4 Automatic run from an arbitrary block

Press 

key to enter the Edit mode, press 

key to enter the Program interface, then press  or  key to enter the PRG CONTENT window:

1. Move the cursor to the block to be executed (for example, move the cursor to the 3rd row head if it executes from the 3rd row);



2. If the mode (G, M, T, F code) of the current block where the cursor locates is defaulted and inconsistent with the running mode of this block, the corresponding modal function should be executed to continue next step.

3. Press 

key to enter the Auto mode, then press 

key to start the execution.

8.1.5 Adjustment of the feedrate, rapid rate

In Auto mode, the running speed can be changed by adjusting the feedrate override, rapid override. It doesn’t need to change the settings of the program and parameter.

*** Adjustment of the feedrate override**

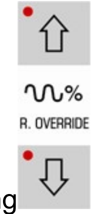
Press 

, 16-level real time feedrate can be obtained.

Note 1: The actual feedrate value is specified by F in feedrate override adjustment;

Note 2: Actual feedrate= value specified by F× feedrate override

＊ Adjustment of rapid override

It can realize the F0, 25％, 50％, 100％ 4-level real time rapid override adjustment by pressing 

or 

.

Note 1: The rapid traverse speeds of X, Z axis are set by the system parameter No.022, No.023 respectively;
X axis actual rapid traverse rate = value set by parameter No.022×rapid override
Z axis actual rapid traverse rate = value set by parameter No.023×rapid override

Note 2: When the rapid override is F0, the min. rapid traverse rate is set by bit parameter No.032.

8.1.6 Spindle speed adjustment

While the spindle speed is controlled by the analog voltage output in Auto mode, it can be adjusted by spindle override.

Press 

to adjust the spindle override for the spindle speed, it can realize 8-level real-time override adjustment between 50％ ~ 120％.

Note： The actual output analog voltage = analog voltage by parameter × spindle override.

Example: When the system parameter No.037 is set to 9999,execute S9999 code to select the spindle override 100%, then the actual output analog voltage ≈ 10×100%=10V.

8.2 Running state

8.2.1 Single block execution

When the program is to be executed for the 1st time, to avoid the programming errors, it may select Single block mode to execute the program.

In Auto mode, the methods for turning on single block switch are as follows:

Method 1: Press the key to make the single block indicator in panel state area to light up, it means that the single block function has been selected;

In Single mode, when the current block execution is finished, the CNC running stops; if next block is to be executed, it needs to press the key again, then repeat this operation till the whole program is finished.

Note 1: The single block stops at the mid point of G28 code.

Note 2: For the single block state in the execution of canned cycle codes G90, G92, G94, G70 ~ G76, refer to the 1st part PROGRAMMING.

Note 3: While the subprogram calling (M98__), or subprogram calling return (M99)is being executed, the single block is inactive. But it is active except for N, O, P addresses in the block that contains M98 or M99 code.

8.2.2 Dry run

Before the program is to be executed automatically, in order to avoid the programming errors, it may select the Dry run mode to check the program.

In Auto mode, the methods for turning on the Dry run switch are as follows:

Method 1: Press the key to make the Dry run indicator in panel state area to light up, it means that the dry run mode has been selected;

In Dry run mode, the machine feed and miscellaneous functions are both active (as machine lock, MST lock are both OFF), that means the dry run switch has nothing to do with the machine feeding, MST functions, so the feedrate by program is inactive and the CNC system runs by the rates in the following table:

	Program command	
	Rapid traverse	Cutting feed
Rapid traverse switch ON	Rapid traverse	Max. manual feedrate
Rapid traverse switch OFF	Manual feedrate or rapid traverse(see note)	Manual feedrate

Note 1: The rate by manual feedrate or rapid rate is set by the BIT6 of the CNC system parameter No.004.

Note 2: The shift of rapid switch in Dry run mode doesn't affect the rate of the current block being executed, but that of the next block.

Note 3: The switch operation of Dry run is inactive if the ladder of this C1000T is defined to be in auto running state (Auto, MDI mode).

8.2.3 Machine lock

In Auto mode, the turning on method of machine lock switch is as follows:

Method 1: Press the key to make the Machine Lock indicator in panel state area to light up, it means that it has entered the machine lock state;

The machine lock and MST lock are usually used together to check the program. While as in the machine lock mode:

1. The machine carriage doesn't move, the "MACHINE" in the INTEGRATED POS window of the Position interface doesn't vary too. The RELATIVE POS and ABSOLUTE POS, DIST TO GO are refreshed continuously, which is the same as that the machine lock switch is OFF.
2. M, S, T commands can be executed normally.

8.2.4 MST lock

In Auto mode, the turning on of MST lock switch is as follows:

Method 1: Press the key to make the MST Lock indicator in panel state area to light up, it means that it has entered the MST lock state;

The machine carriage moves without the M, S, T code being executed. The machine lock and MST lock are usually used together to check the program.

Note: When the MST lock is active, it takes no effect to the execution of M00, M30, M98, M99.

8.2.5 Block skip

If a block in program is not needed to be executed and not to be deleted, this block skip function can be used. When the block is headed with "/ " sign and Block skip indicator lights up (panel key active or external skip input active), this block is skipped without execution in Auto mode.

In Auto mode, the turning on of Block skip switch is as follows:

Method 1: Press the key to make the Block skip indicator in panel state area to light up;

Note: While the block skip switch is off, the blocks headed with "/" signs are executed normally in Auto mode.

8.3 Other operations

1. In Auto mode, press key to switch on/off the cooling;
2. Press any of the , , , , or keys to switch the operation modes;
3. Press the key to reset this CNC system.
4. Automatic lubricating operation (Refer to **Volume II Operation, Chapter 3**).

CHAPTER 9 ZERO RETURN OPERATION

Note!

The key functions of this running state (Auto, MDI mode). machine panel are defined by PLC program (ladders), please refer to the manuals by the machine builder for their significance.

Please note that the following description for the panel key functions in this chapter is based on the C1000T standard PLC program!

9.1 Program zero return

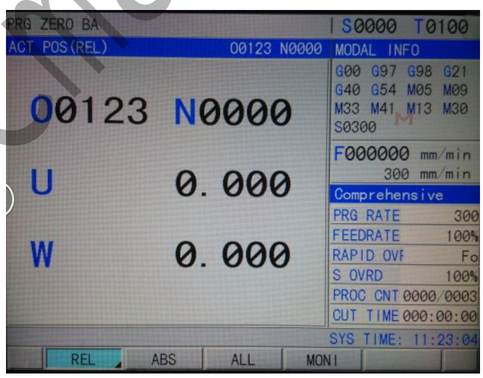
9.1.1 Program Zero

While the part is fixed on the machine, absolute coordinate of current tool position may be set by G50 code according to the relative position between the tool and the part, so a workpiece coordinate system is setup. The tool current position is called **program zero**, and this is the program zero return position.

9.1.2 Program zero return steps

1. Press 

key, it enters the Program zero return mode, the bottom line of the window displays “PROGRAM ZERO”, as the following figure shows:



2. Press the direction key of X, Z or Y axis, it returns to the program zero of X, Y or Z axis;
3. The machine axis moves toward the program zero return, and the axis stops with the program zero return completion indicator ON after the axis returns to the program zero.

Program zero return completion indicator

Note 1: The tool offset is not changed for the program zero return operation, if there is offset, the return position is the point set by G50.

Note 2: Whether the key is held on at program zero return is defined by the bit parameter No.011 BIT2 (ZNIK).

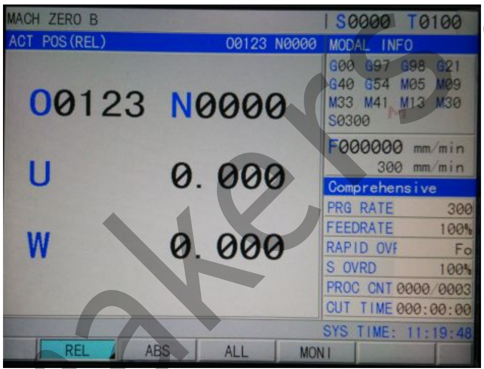
9.2 Machine Zero return

9.2.1 Machine Zero (machine reference point)

The **machine coordinate system** is a reference coordinate system for CNC coordinate operation. It is an inherent coordinate system of the machine. The origin of the machine coordinate system is called machine zero (or mechanical reference point). It is defined by the zero or zero return switch fixed on the machine. Usually this switch is fixed at the positive stroke point of X or Z axis.

9.2.2 Machine Zero return steps

1. Press  key, it enters the Machine zero mode, the bottom line of the window displays “MACHINE ZERO”, as the following figure shows:



2. Press , ,  or  key to return to the machine zero of X, Z or Y axis;

3. The machine axis returns to the machine zero via the deceleration signal, zero signal detection. At the machine zero, the axis stops, and the corresponding machine zero return completion indicator lights up.

Machine zero return completion indicator

Note 1: If there is no machine zero on the machine, machine zero operation is forbidden;

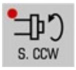
Note 2: The machine zero finish indicator is gone out on condition that:
The axis is moved out from machine zero;
CNC is powered off.

Note 3: After the machine zero operation, the tool length compensation is cancelled by CNC;

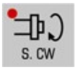
Note 4: Parameters related to machine zero return are referred to Volume III INSTALLATION and CONNECTION.

Note 5: After the machine zero return is executed, the original workpiece coordinate system is set again with G50.

9.3 Other operations in zero return

1. Press 

key, the spindle rotates counterclockwise;
2. Press 

key, the spindle stops;
3. Press 

key, the spindle rotates clockwise;
4. Press 

key, the cooling is switched ON or OFF;
5. Lubricating control(refer to OPERATION, Chapter 3);
6. Press 

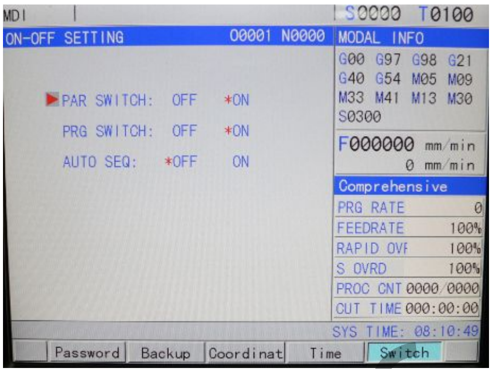
key, the tool change is executed;
7. Tune the spindle override;
8. Tune the rapid override;
9. Tune the feedrate override.

CHAPTER 10 DATA SETTING, BACKUP and RESTORE

10.1 Data setting

10.1.1 Switch setting

In SWITCH SETTING window, the ON-OFF state of PARM SWT (parameter switch), PROG SWT (program switch), AUTO SEG (auto sequence No.) can be displayed and set, as is shown in following figure:



1. Press SETTING key to enter the Setting interface, then press ▤ or ▤ key to enter the SWITCH SETTING window;

2. Press ⇧ or ⇩ key to move the cursor to the item to be set;

3. Press U and R key to shift the ON-OFF state: press U key, "*" moves to the left to set the switch for OFF, press R key, "*" moves to the right to set the switch for ON. Only the PARM SWT is set for ON, could the parameter be altered; so are PROG SWT and AUTO SEG.

Note: When the PARM SWT is shifted from "OFF" to "ON", an alarm will be issued by CNC system. By pressing the CANCEL, RESET key together, the alarm can be cancelled. If the PARM SWT is shifted again, no alarm is issued. For security it should set the PARM SWT for "OFF" after the parameter alteration is finished.

10.1.2 Graphic display

Press SETTING key to enter the path window

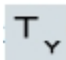
Graphic parameter meaning

1. Coordinate system setting: 8 types of graphic paths can be displayed in this C1000T CNC system depending on the front or rear tool post coordinate system

A: Graphic scaling up and down

In Graphic window, the graphic path can be scaled up and down by the keys  ,  in the edit keypad.

B: The START, STOP and CLEAR of the graphic path

In Graphic window, press the  key once, it starts the drawing; press the  key once, it stops drawing; press  key once, it clears the current graphic path.

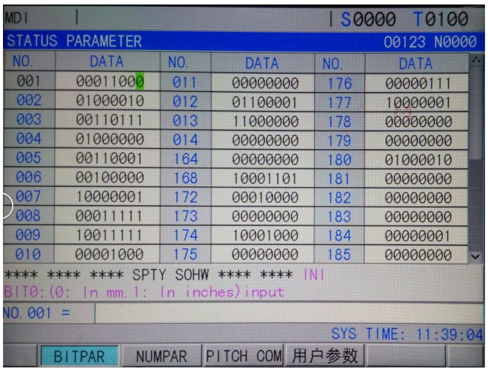
C: Move of graphic path

In Graphic window, it may press the direction keys to move the graphic path

10.1.3 Parameter setting

By the parameter setting, the characteristics of the driver and machine can be adjusted. See Appendix 1 for their significance.

Press  key to enter the Parameter interface, then press  or  key to window the parameter interface, as is shown in the following figure:



A. Alteration of the bit parameter

1. Byte alteration

1) Turn on the parameter switch;

2) Enter the MDI mode;

3) Move the cursor to the parameter No. to be set:

Method 1: Press or key to enter the window containing the parameter to be set, press or key to move the cursor to the No. of the parameter to be set;

Method 2: Press address key P Q, key in parameter No., then press DATA INPUT key.

4) Key in the new parameter value;

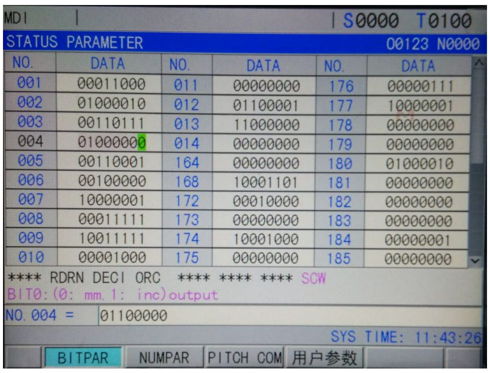
5) Press DATA INPUT key, the parameter value is entered and displayed.

6) For security, the PARM SWT needs to be set for OFF after all parameter settings are finished.

Example:

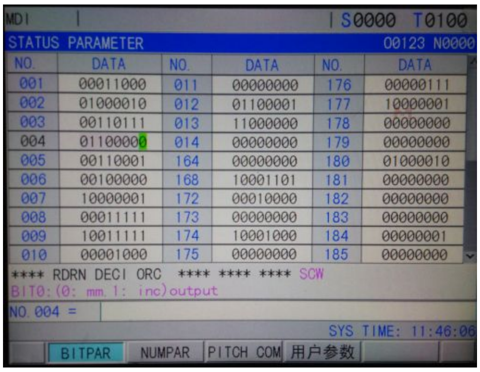
Set the bit parameter No.004 Bit (DECI) to 1, and the other bits unchanged.

Move the cursor to No.004, input 01100000 by sequence in the prompt row, the display is as follows:



Press DATA INPUT key to finish the parameter alteration. The window is shown as follows:

Volume II Operation



2. Alteration by bit:

1) Turn on the parameter switch;

2) Enter the MDI mode;

3) Move the cursor to the No. of the parameter to be set;

Method 1: Press  or  key to enter the window of the parameter to be set, press  or  key to move the cursor to the No. of the parameter to be set;

Method 2: Press address key 

, key in parameter No., then press 

key.

4) Press 

key to skip to a bit of the parameter, and the bit is backlighted. Press  or  key to move the cursor to the bit to be altered, then key in 0 or 1;

5) After all parameters setting is finished, the PARM SWT needs to be set for OFF for security.

Note: After entering a bit of the parameter, press 

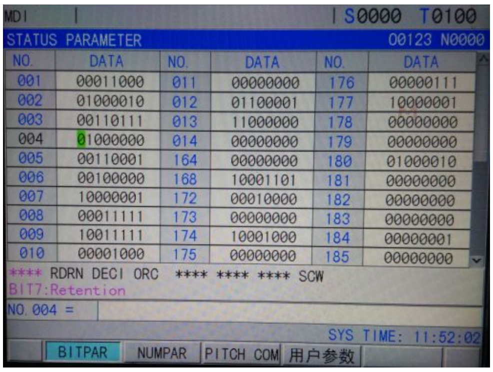
key, it may skip out of the bit and back to the parameter No..

Example:

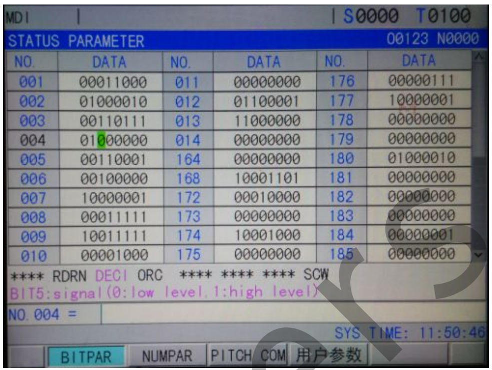
Set the BIT5 (DECI) of the bit parameter No.004 to 1, and the other bits unchanged.

Move the cursor to "No.004" by the steps above, press 

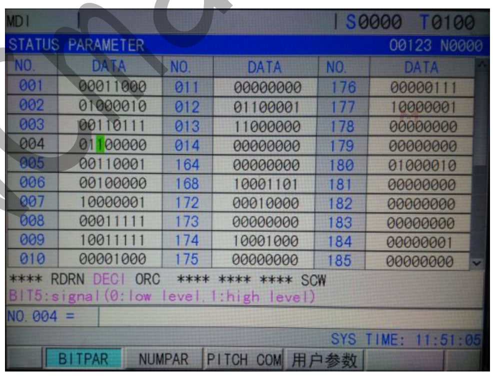
key to skip to a bit of the parameter as follows:



Move the cursor to “BIT5” by pressing  or  key as follows:



Input “1” to finish the alteration.



B Altering data parameter and screw-pitch parameter

1) Turn on the parameter switch;
2) Enter the MDI mode;
3) Move the cursor to the No. of the parameter to be set;
4) Key in the new parameter value;
5) Press 

key, the value is entered and displayed;
6) After all parameters setting is finished, the PARM SWT needs to be set for OFF for security.

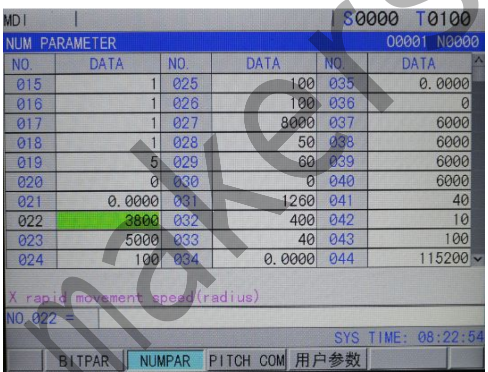
Volume Ⅱ Operation

Explanation: The screw-pitch parameter can only be altered under the 2 level password authority.

Example 1: set the data parameter No.022 to 3800.

Move the cursor to “No.022” by the steps above, key in “3800” by sequence in the cue line as follows:

Press DATA INPUT key to finish the alteration. The window is shown as follows:



Example 2: set X value of the screw-pitch parameter No.000 to 12, Z axis value of that to 30.

Move the cursor to screw-pitch parameter No.000 by the steps above, key in “X12” by sequence in the cue line.Press DATA INPUT key to finish the alteration.

The same as above, key in “Z30” by sequence in the cue line, press 

key to finish the alteration. The window is as follows:

10.2 Data recovery and backup

The user data (such as bit parameter, data parameter, and screw-pitch parameter) can be backup (saved) and restored (read) in this C1000T system. It doesn’t affect the part programs stored in the CNC system while backuping and restoring these data. The backup window is shown as follows:

1. Turn on the parameter switch;

2. Press MDI key to enter the MDI mode, then press SETTING key (or key if necessary) to enter OPERATE DATA window;

3. Press 2 key to enter Backup DATA Window,press 1 to enter RECOVERY DATA Window

4. Move cursor to operated option, press DATA INPUT key to select / cancle operation option

Note 1: Don't cut off the power in the backup and restore operation of the data, and no other operation is suggested to be performed before the operation is prompted to be finished.

Note 2: The user above the 3 password level can perform the backup and restore operation of the bit parameter, data parameter and the screw-pitch parameter.

10.3 Password setting and alteration

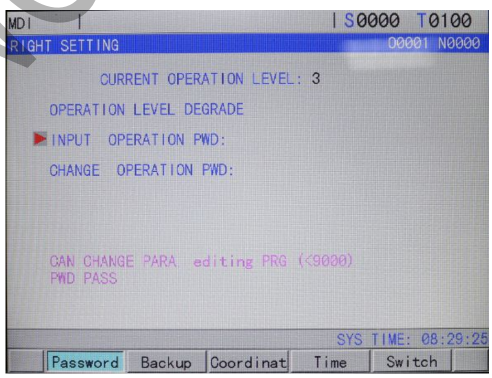
To protect the part programs, CNC parameters from malignant alteration,this C1000T provides password setting function that is graded for 4 levels. By descending sequence, they are machine builder (2) level, equipment management (3) level, technician (4) level, machining operation (5) level. The current password level of the CNC system is displayed for "CURRENT LEVEL:_" in the PASSWORD SETTING window.

2 level: the CNC bit parameter, data parameter, screw-pitch parameter, tool offset data, part program edit, PLC ladder transmission etc. are allowed;

3 level: the initial password is 12345, the CNC bit parameter, data parameter, tool offset data, part program edit operations are allowed;

4 level: the initial password is 1234, tool offset data (for tool setting), macro variables, part program edit operations are allowed; but the CNC bit parameter, data parameter, screw-pitch parameter operations are forbidden.

5 level: no password. Only the machine panel operation is allowed, and the operations of part program edit and selection, the alteration operations of CNC bit parameter, data parameter, screw-pitch parameter, tool offset data are forbidden.



After entering the PASSWORD SETTING window, the cursor locates at the "INPUT PASSWORD:" row. It may press the ⇧ or ⇩ key to move the cursor to the corresponding item.

a) Press ⇧ key once, the cursor shifts a row upward. If the current cursor locates at the

"SET LOWER LEVEL" row (1st row), press  key, the cursor shifts to the "ALTER PASSWORD:" row(end row);

b) Press  key once, the cursor shifts a row downward. If the current cursor locates at the end row, by pressing  key once, the cursor shifts to the 1st row.

10.3.1 Operation level entry

1 Move the cursor to the "INPUT PASSWORD:" row after the system enters the PASSWORD SETTING window;

2 Input the password (an "*" sign added each time inputting a character);

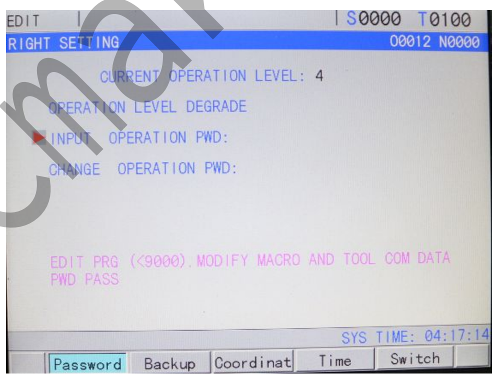
3 Press  key to finish the inputting, and the system enters the corresponding password level.

Note: The length of C1000T password corresponds to the operation level, which can't be added or reduced by user at will. The detailed is as follows:

Operation level	Password length	Initial password
3	5 bytes	12345
4	4 bytes	1234
5	No	No

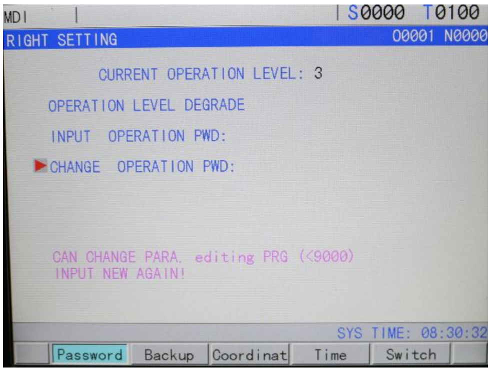
Example:

The current CNC operation level is 4 level, as the following window shows, the 3 level password of CNC is 12345, please alter the current level to the 3 level.



Move the cursor to the "INPUT PASSWORD:" row, key in 12345, then press the  key, the CNC prompts "Modify parameter and edit program", "PASSWORD PASSED.", and the current level is the 3 level. The display is as follows:

Volume II Operation



Note: When current operation level is lower than or equal to the 3 level (3, 4, 5 level), this level is not changed if the CNC system is turned on again. If previous level is the 2 level, it defaults the 3 level when the system is turned on again.

10.3.2 Altering the password

Steps for password alteration:

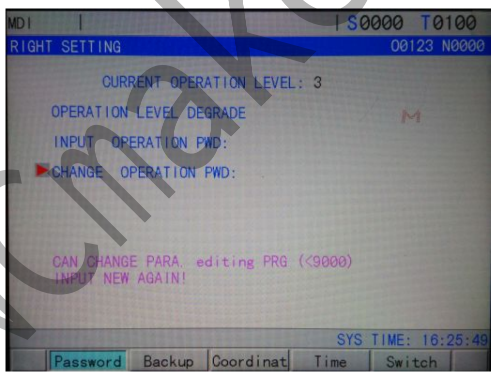
1 After entering the PASSWORD SETTING window, enter the password by the methods in Section10.3.1;

2 Move the cursor to the “ALTER PASSWORD:” row;

3 Key in the new password, then press 

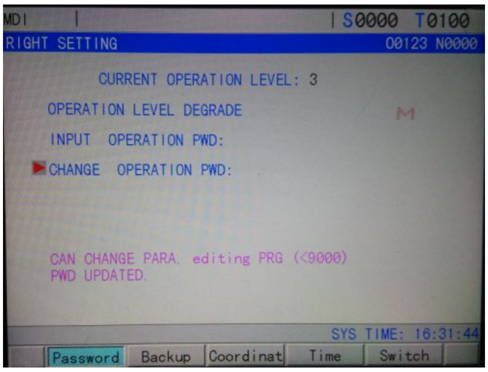
key;

4 The CNC system prompts “PLEASE INPUT USER PASSWORD AGAIN!”, the window display is as follows:

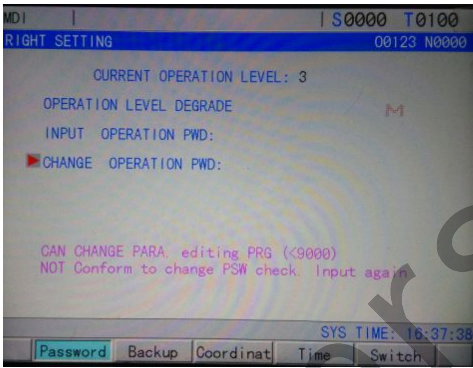


5 After re-inputting the password, press 

key, if the passwords input are identical, CNC prompts “PASSWORD UPDATED.”. So the password alteration is successful.



6 If the inputs of the passwords are not identical, CNC prompts "PASSWORD CHECKOUT ERROR.", the window is as follows:

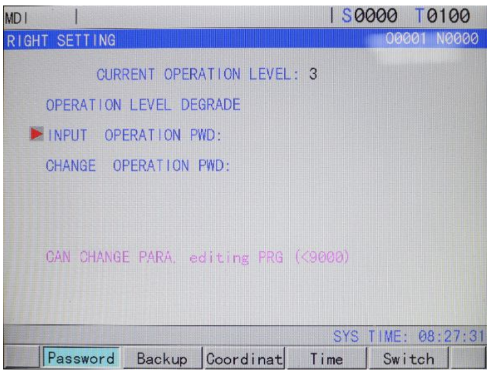


10.3.3 Setting the lower password level

The demotion of the operation level is used to enter a lower level from a higher level, the steps are as follows:

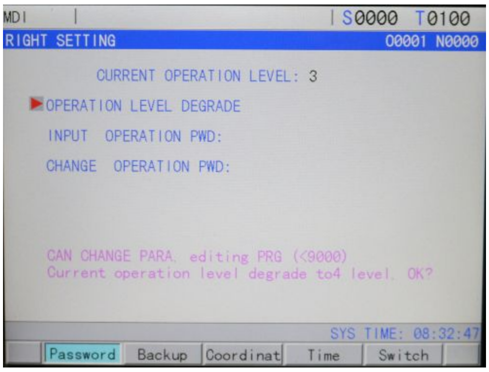
1 After entering the PASSWORD SETTING window, key in the password by the method in Section 10.3.1;

2 Move the cursor to the "SET LOWER LEVEL:" row, if the current CNC operation is the 3 level, the window is as follows:



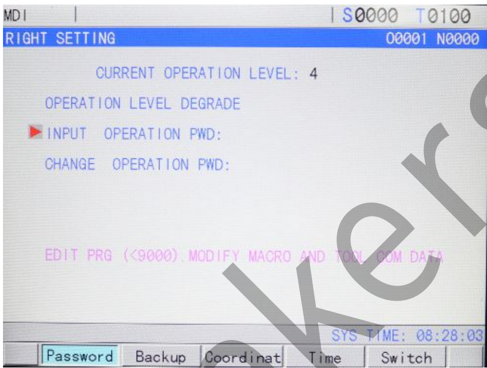
3 Press 

key, the CNC system prompts “CURRENT LEVEL TO 4, MAKE SURE? ”, the window is as follows:



4 Press 

key again, if the demotion is successful, the window is as follows:

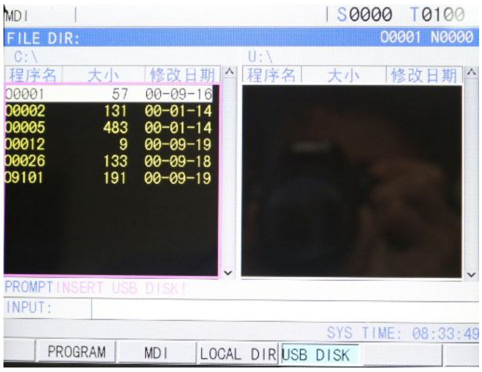


Note: If the current level is the 5 level, the demotion operation is forbidden.

CHAPTER 11 U OPERATION FUNCTION

11.1 File catalog window

In non-edit mode, press PROGRAM to enter the program window, press to enter [File catalog] window, press CHANGE to identify it after U disk is inserted as follows:



The left displays CNC catalog information and the right displays USB disc catalog information. When the system has not checked the U disc, the right does not display the content. The bottom displays the file capacity and user operation prompt. The system only displays ".CNC",".NC" and ".txt" in the current file and other extension names are not displayed.

Press CHANGE and the cursor is switched from CNC to USB, press ⇧ or ⇩ to move it.

11.2 File copy

Move the cursor the required CNC format file , press DATA OUTPUT to copy.

CHAPTER 12 MACHINING EXAMPLES

Machine a part by a bar stock with dimension Φ136mm×180 mm, as follows:

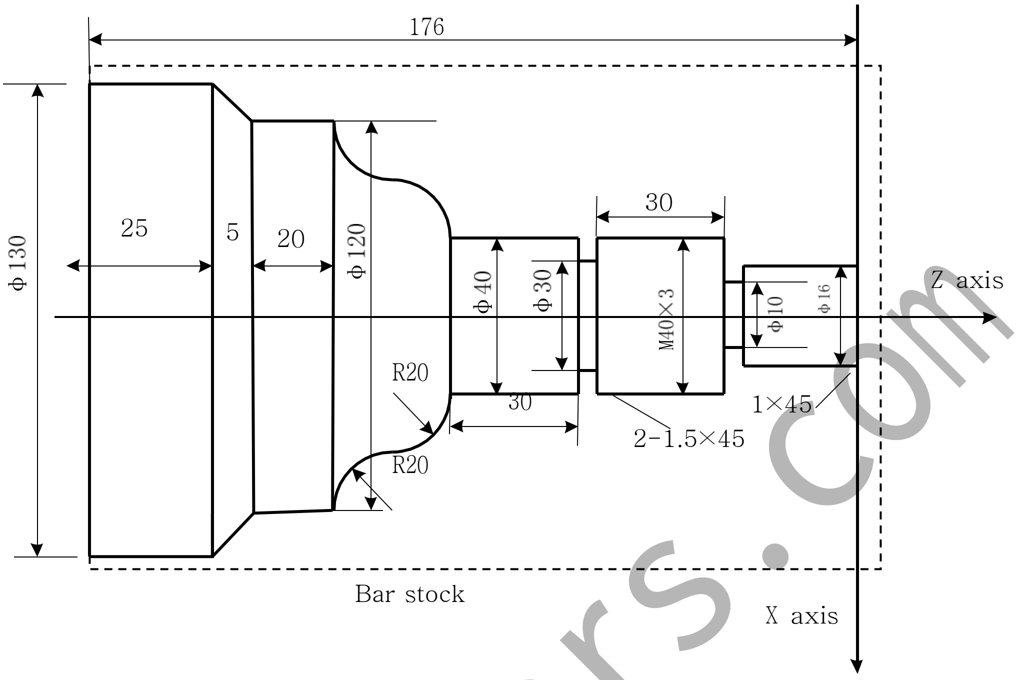
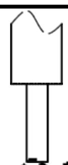


Fig. 14-1

Machine it with 4 tools as follows:

Tool number	Tool shape	Explanation
No. 1		Outer circle rough turning tool
No. 2		Outer circle finish turning tool
No. 3		Grooving tool, tool width 3mm
No. 4		Threading tool, tool nose angle 60°

12.1 Programming

Set up the workpiece coordinate system as Fig.14-1 according to the machining process and the codes introduced in this manual. The programming steps are as follows:

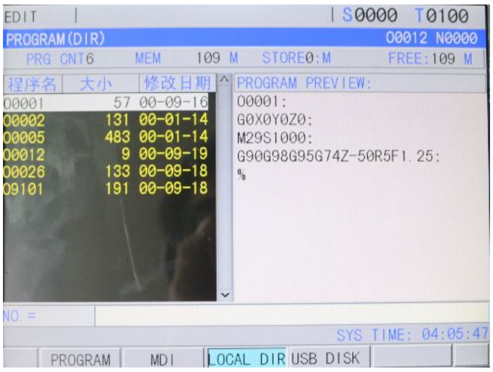
Ｏ０００１；		Name of the part program
Ｎ００００	G0 X150 Z50；	Position to the safety height for tool change
Ｎ０００５	M12；	Clamp the chuck
Ｎ００１０	M3 S800；	Start the spindle with speed 800
Ｎ００２０	M8；	Turn on the cooling Ｎ０
０３０	T0101；	Change for the No. 1 tool
Ｎ００４０	G0 X136 Z2；	Approach the part
Ｎ００５０	G71 U0.5 R0.5 F200；	Cut depth 1mm and retract 1mm
Ｎ００５５	G71 P0060 Q0150 U0.25 W0.5；	0.5mm pre-reserved in X axis, 0.5mm machining allowance in Z axis
Ｎ００６０	G0 X16；	Approach to the end face of the part
Ｎ００７０	G1 Z-23；	Cut the Φ16 outer circle Ｎ
００８０	X39.98；	Cut the end face Ｎ００９
０	W-33；	Cut the Φ39.98 outer circle
Ｎ０１００	X40；	Cut the end face Ｎ０１０
５	W-30；	Cut the Φ40 outer circle Ｎ
０１１０	G3 X80 W-20 R20；	Cut the convex arc Ｎ０１
２０	G2 X120 W-20 R20；	Cut the concave arc Ｎ０１
３０	G1 W-20；	Cut the Φ120 outer circle Ｎ
０１４０	G1 X130 W-5；	Cut the cone
Ｎ０１５０	G1 W-25；	Cut the Φ130 outer circle
Ｎ０１６０	G0 X150 Z185；	Rough cut end and back to the tool change point
Ｎ０１７０	T0202；	Change for the No.2 tool and execute its offset
Ｎ０１８０	G70 P0060 Q0150；	Fine cut cycle
Ｎ０１９０	G0 X150 Z185；	Rough cut end and back to the tool change point
Ｎ０２００	T0303；	Change for the No.3 tool and execute its offset
Ｎ０２１０	G0 Z-56 X42；	Approach to the part

Ｎ０２２０	G1 X30 F100；	Cut the Φ30 groove
Ｎ０２３０	G1 X37 F300；	Return
Ｎ０２４０	G1 X40 W1.5；	Chamfering
Ｎ０２５０	G0 X42 W30；	Keep the width of the grooving
Ｎ０２６０	G1 X40 ；	
Ｎ０２６２	G1 X37 W1.5；	
Ｎ０２６４	G1 X10；	Chamfering
Ｎ０２６６	G0 X17 Z-1；	Cut the Φ10 groove
Ｎ０２６８	G1 X16；	
Ｎ０２７０	G1 X14 Z0 F200；	Chamfering
Ｎ０２８０	G0 X150 Z50；	Return to the tool change point
Ｎ０２９０	T0404 S100；	Changing for the No. 4 tool and set the spindle speed for 100
Ｎ０３００	G0 X42 Z-20；	Approach the part Ｎ０３１０
	G92 X39 W-34 F3；	Thread-cutting cycle Ｎ０３２
０	X38；	Feed 1mm for the 2nd cutting Ｎ
０３２０	X37；	Feed 1mm for the 3rd cutting Ｎ
０３３０	X36.4；	Feed 0.6mm for the 4th cutting
Ｎ０３３２	X36；	Feed 0.4mm for the 5th cutting
Ｎ０３４０	G0 X150 Z50；	Return to the tool change point
Ｎ０３５０	T0100 U0 W0；	Change for the No.1 tool and execute its offset
Ｎ０３６０	M5；	Turn off the spindle
Ｎ０３７０	M9；	Turn off cooling Ｎ
０３８０	M13；	Unclamp the chuck
Ｎ０３９０	M30；	Program ends

12.2 Program input

12.2.1 View a saved program

In a non-Edit mode, press PROGRAM key to enter Program interface, select the PRG LIST window by pressing  or  key, the window is as follows:

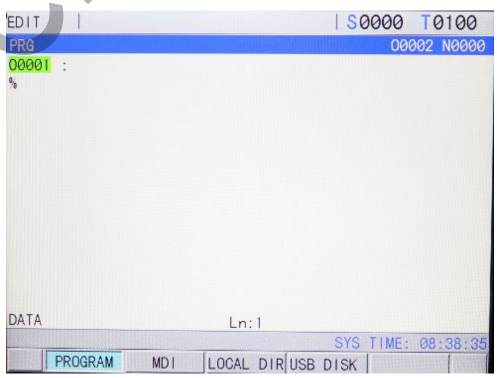


In above window the names of the programs saved can be viewed for renaming the new program.

12.2.2 Creating a new program

In Edit mode, press PROGRAM key to enter the PRG CONTENT window as follows:

Press address key O, choose a name that is not same with the ones in this window (i.e. 0001), key in the number key 0, 0, 0, 1 and the EOB key by sequence to create a new program as follows:

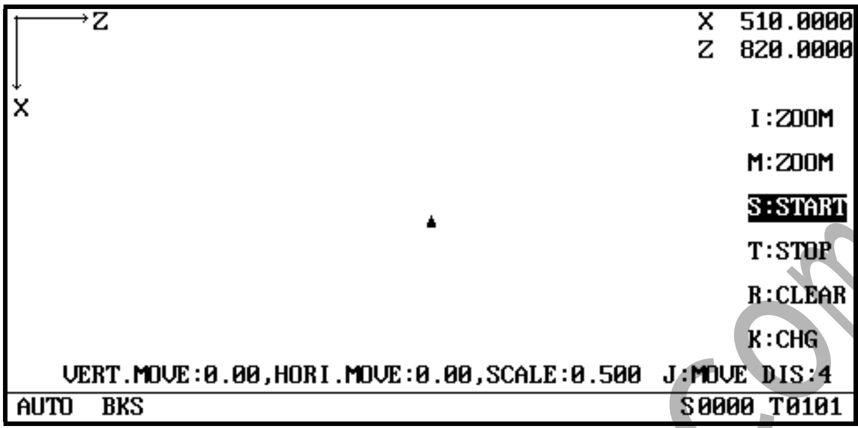


Complete the program editing by inputting the above program word by word.

12.3 Checkout a program

12.3.1 Graphic setting

1. Press GRAPH to enter the graphic window as follows:



12.3.2 Program check

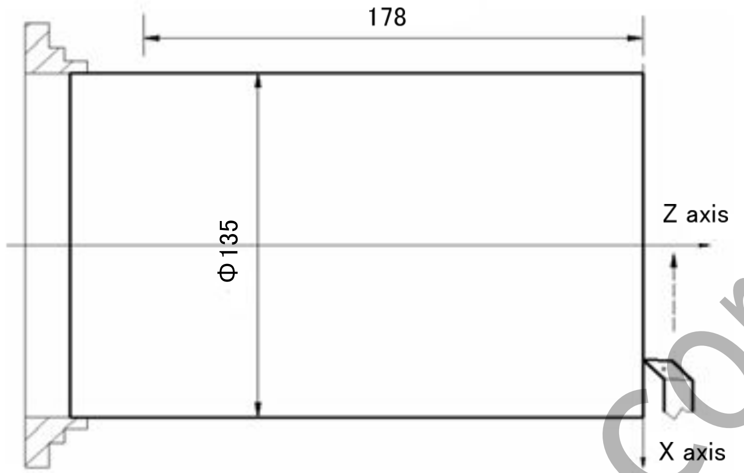
Press or to enter the graph display window, press AUTO to enter auto operation mode, press M.S.T. LOCK, MACHINE LOCK, EDIT to make the auxiliary function lock indicator M.S.T. LOCK, machine lock indicator MACHINE LOCK, and dry run indicator EDIT to enter the corresponding state. Press S] to start the drawing, press CYCLE START to automatically run programs, check the program accuracy by displaying the tool motion path, and the display window is as follows after the run is completed:

If there is error in the program path, make a diagnosis for the error in the program and modify the program. Then make another checkout for the program by the method above till the error is eliminated. In the Graphic interface, press S] key on the panel to start drawing, or press T Y key to stop drawing, or press J B key to clear the drawing.

12.4 Tool setting and running

1. Move the tool to a safe position, run the T0100 U0 W0 command in the PRG STATE window of the MDI mode, and cancel the tool offset;

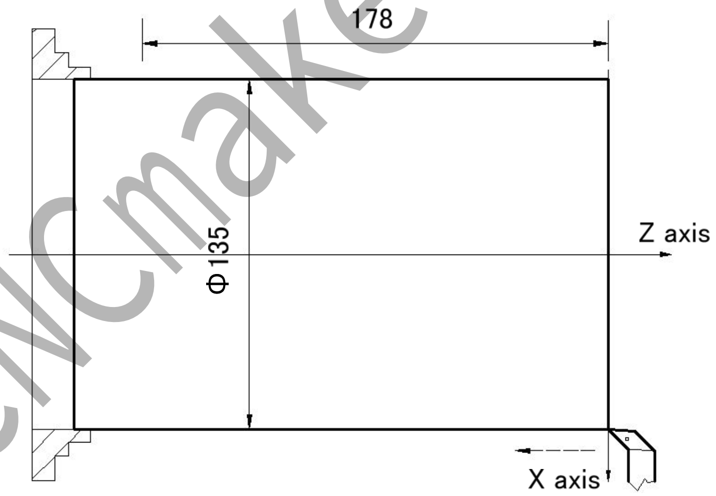
2. Move the tool to cut in the part end surface;



3. Release the tool along X when Z does not move, and stop the spindle, execute G50 Z0 in the PRG STATE window of the MDI mode to set the coordinate of Z axis;

4. Switch to TOOL OFFSET window and input Z0 to No.001 offset;

5. Move the tool and make it to cut along the outer circle of the part;



6. Release the tool along Z when X does not move, and stop the spindle, measure the dimensions of the outer circle of the part (e.g. The measuring value is 135mm);

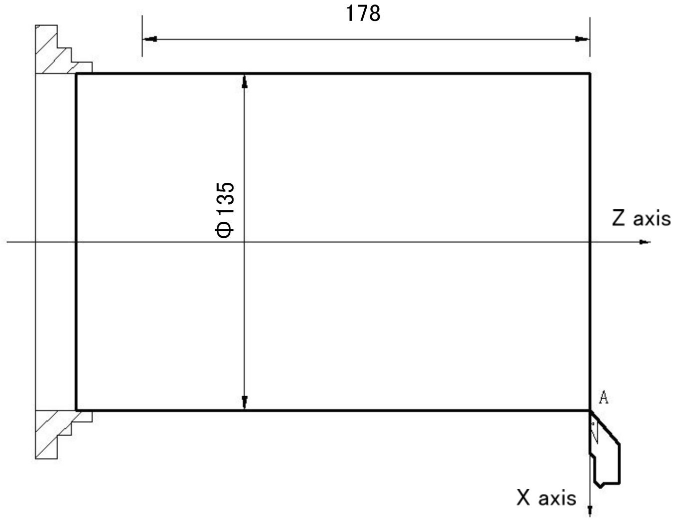
7. Execute G50 X135 command in the PRG STATE window of the MDI mode to set the coordinate of X axis;

8. Switch to the TOOL OFFSET window, and input X135 to No.001 offset;

9. Move the tool to a safe position, and press the 

key in Manual mode to change for the No. 002 tool;

10. Start the spindle and move the tool to the tool setting point, as A point in the following figure;

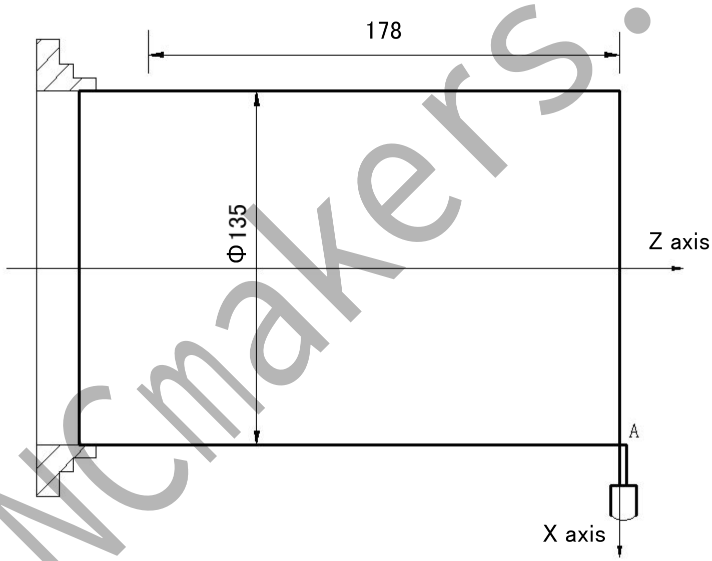


11. Switch to TOOL OFFSET window, move the cursor to No.002 offset and input X135 Z0;

12. Move the tool to a safe position, and press the 

key in Manual mode to change for the No. 003 tool;

13. Start the spindle and move the tool to the tool setting point, as A point in the following figure;

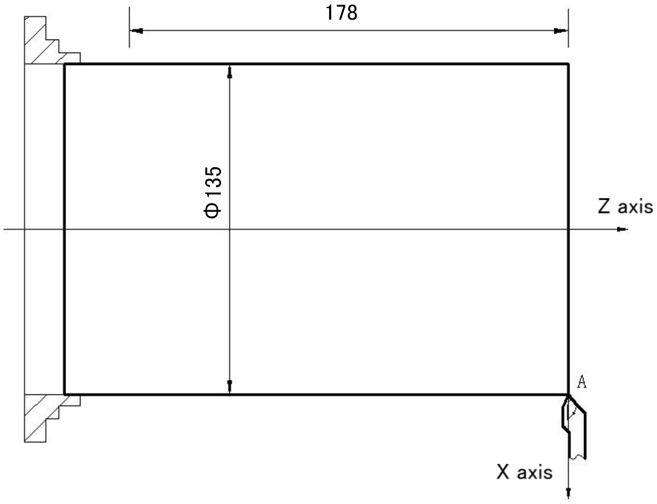


14. Switch to TOOL OFFSET window, move the cursor to No.003 offset and input X135 Z0;

15. Move the tool to a safe position, and press the 

key in Manual mode to change for the No. 004 tool;

16. Move the tool to the tool setting point, as point A in the following figure;



17. Switch to TOOL OFFSET window, move the cursor to No.004 offset and input X135 Z0;

18. Move the tool to a safe position after the tool setting is finished;

19. Press  key to start the machining in Auto mode;

20. If there is any error between the designed and the actual dimensions, the tool offset may be altered till the part dimensions are within the tolerance.

Note: Press  key to make the auto running to pause if dwell is needed during the machining. Also if emergency occurs, it may press the  key, Emergency stop button to cut off the power to terminate the program running.

Volume Ⅲ Connection

CHAPTER 1 INSTALLATION LAYOUT

1.1 C1000T system connection

1.1.1 C1000T back cover interface layout

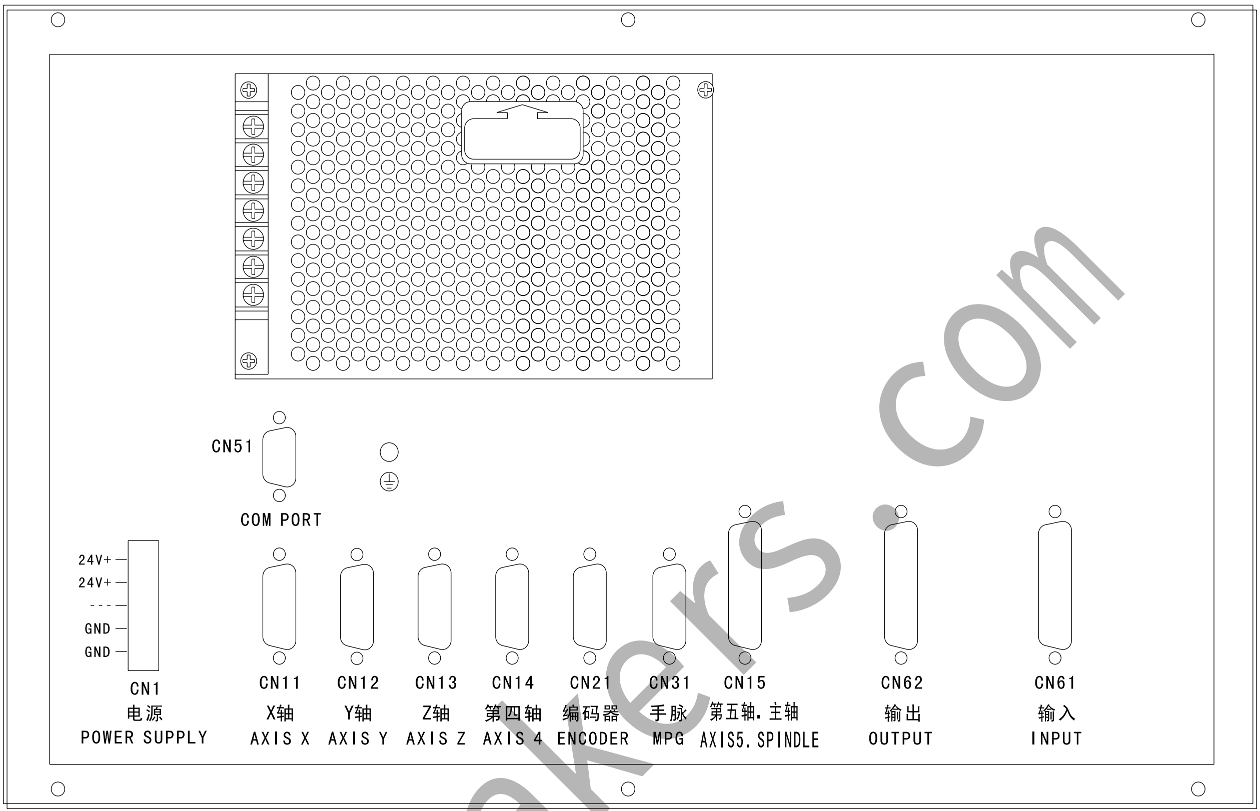


Fig. 1-1 C1000T back cover interface layout

Volume III Connection

1.1.2 Interface explanation

- Power box: for +24V, GND power supply
- Filter(optional): Input terminals for 220V AC power, PE terminal for grounding, output terminals to L, N terminals of CNCmakers Limited
  CNCmakers Limited-PB2 power box ● CN1： power supply interface
- CN11：X axis，pin15 D female, connect with X drive unit
- CN12：Y axis，pin15 D female, connect with Y drive unit
- CN13：Z axis，pin15 D female, connect with Z drive unit
- CN14：4th axis，pin15 D female, connect with 4th drive unit
- CN15：spindle, pin 25 D female, connect with spindle drive unit
- CN21：encoder, pin15 D male, connect with spindle encoder
- CN31：MPG, pin26 D male, connect with MPG
- CN51：communication, pin9 D female, connect PC RS232 interface
- CN61：input, pin44 D male, connect with machine input
- CN62：output, pin44 D female, connect with machine output

1.2 C1000T installation

1.2.1 C1000T external dimensions

See appendix Ⅰ, Ⅱ.

1.2.2 Preconditions of the cabinet installation

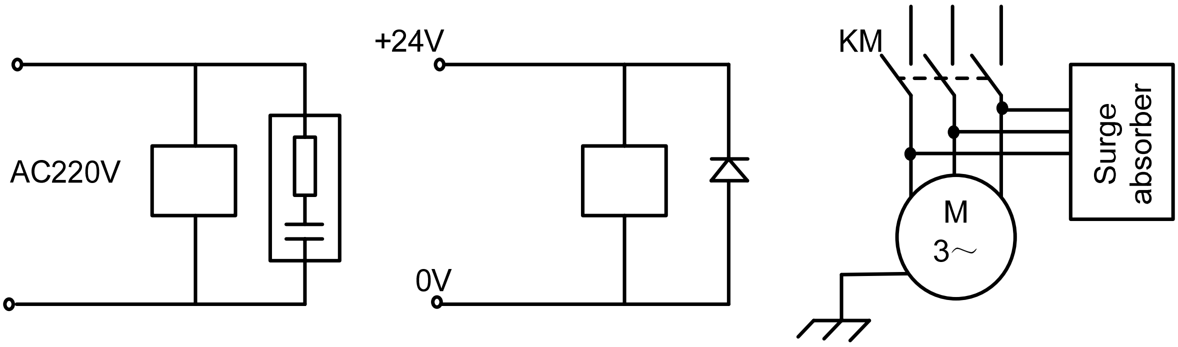
- The dust, cooling liquid and organic resolution should be effectively prevented from entering the cabinet;
- The designed distance between the CNC back cover and the cabinet should be not less than 20cm, the inside and outside temperature difference of the cabinet should be not more than 10℃ when the cabinet inside temperature rises;
- Fans can be fixed in the cabinet to ventilate it;
- The panel should be installed in a place where the cooling can't splash;
- The external electrical interference should be taken into consideration in cabinet design to prevent it from interfering the CNC system.

1.2.3 Measures against interference

In order to insure the CNC stable working, the anti-interference technology such as space electromagnetic radiation shielding, impact current absorbing, power mixed wave filtering are employed in CNC design. And the following measures are necessary during CNC connection:

1. Make CNC far from the interference devices (inverter, AC contactor, static generator, high-voltage generator and powered sectional devices etc.);
2. To supply the CNC via an isolation transformer, the machine with the CNC system should be grounded, the CNC and drive unit should be connected with independent grounding wires at the grounding point;
3. To inhibit interference: connect parallel RC circuit at both ends of AC winding (Fig. 1-3a), RC circuit should approach to inductive loading as close as possible; reversely connect parallel freewheeling diode at both ends of DC winding (Fig. 1-3b); connect parallel surge absorber at

the ends of AC motor winding (Fig. 1-3c);



a) b) c)

Fig. 1-3

4. The CNC leadout cables use the twisted shield cable or shield cable, the cable shield tier is grounded by an terminal at CNC side, signal cable should be as short as possible;
5. To reduce the mutual interference among the CNC signal cables, and among the strong current, the wiring should follow the following:

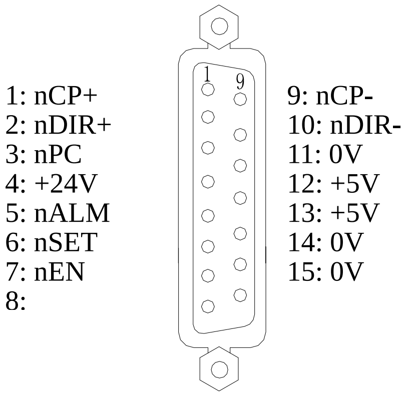
Table 1-1 The Wiring requirement

Group	**Cable type**	**Wiring requirement**
A	AC power cable	Tie up A group cables with a clearance at least 10cm from that of B, C groups, or shield A group cables from electromagnetism
	AC coil	
	AC contactor	
B	DC coil(24VDC)	Tie up B and A group cables separately or shield B group cables; and the further B group cables are from that of C group, the better it is
	DC relay(24VDC)	
	Cables between CNC and strong-power cabinet	
	Cables between CNC and machine	
C	Cables between CNC and servo drive unit	Tie up C and A group cables separately, or shield C group cables; and the cable distance between C group and B group is at least 10cm and they are twisted pair cables.
	Position feedback cable	
	Position encoder cable	
	Handwheel（MPG）cable	
	Other cables for shield	

CHAPTER 2 DEFINITION & CONNECTION OF INTERFACE SIGNALS

2.1 Connection to drive unit

2.1.1 Drive interface definition



Signal	Explanation
nCP+, nCP-	Code pulse signal
nDIR+, nDIR-	Code direction signal
nPC	Zero signal
nALM	Drive unit alarm signal
nEN	Axis enable signal
nSET	Pulse disable signal

Fig.2-1 CN11, CN12, CN13, CN14 interface
(15-core D type female socket)

2.1.2 Code pulse and direction signals

nCP+, nCP- are code pulse signals, nDIR+, nDIR- are code direction signals. These two group signals are both differential output (AM26LS31), it is suggested to receive by AM26LS32 externally, and the interior circuit for them is shown in Fig. 2-2:

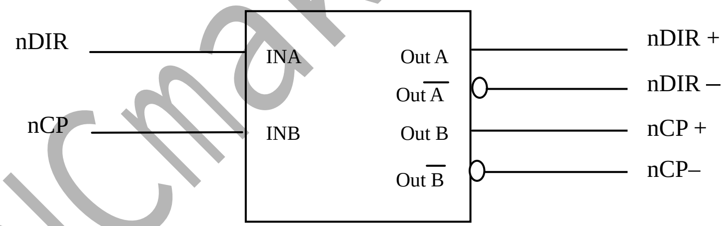


Fig. 2-2 Interior circuit of code pulse and direction signals

2.1.3 Drive unit alarm signal nALM

The low or high level of the drive unit alarm is set by the CNC parameter No.009 Bit0, Bit1, Bit2, Bit3 and Bit4, its interior circuit is shown in Fig. 2-3:

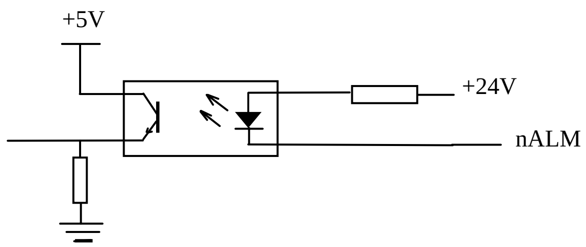


Fig. 2-3 Interior circuit of drive unit alarm signal

This type of input circuit requires that the drive unit transmits signal by the types in Fig. 2-4:

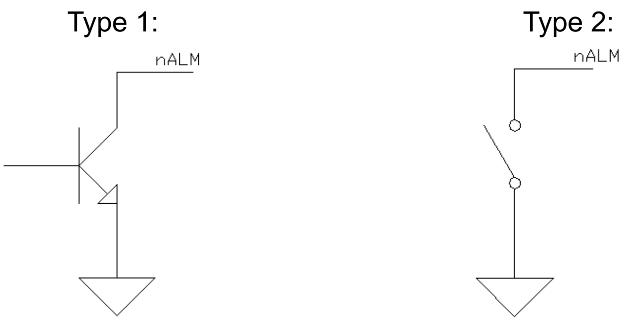


Fig. 2-4 Signal types by drive unit

2.1.4 Axis enable signal nEN

nEN signal output is active as CNC works normally (nEN signal to 0V on); when the drive unit alarm or emergency alarm occurs, CNC cuts off nEN signal output (nEN signal to 0V off). The interior interface circuit is shown in Fig. 2-5:

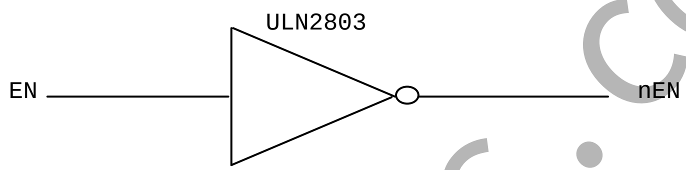


Fig. 2-5 Interior interface circuit for axis enable signal

2.1.5 Pulse disable signal nSET

nSET signal is used to control servo input disable which can enhance the anti-disturbance capability between CNC and drive unit. This signal is at low level if there is pulse output from CNC, high resistance if not. The interior interface circuit of it is shown in Fig. 2-6:

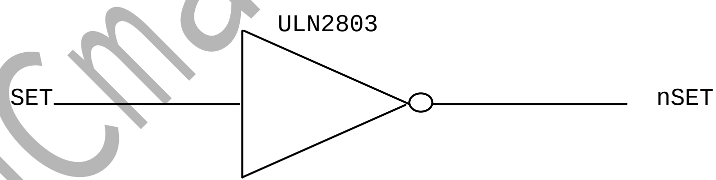


Fig. 2-6 Pulse disable signal circuit

2.1.6 Zero signal nPC

During machine zero return, the one-turn or proximity switch signal from the motor encoder is taken as zero signal. Its interior circuit is shown in Fig.2-7.

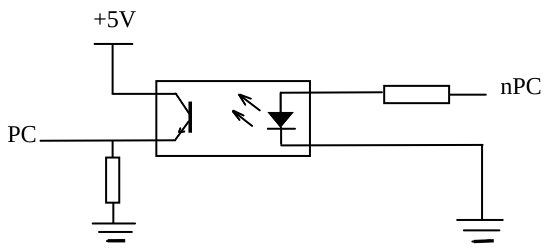


Fig. 2-7 Zero signal circuit

Note: nPC signal uses +24V level.

Volume III Connection

a) The wave of PC signal by user is shown in Fig. 2-8:

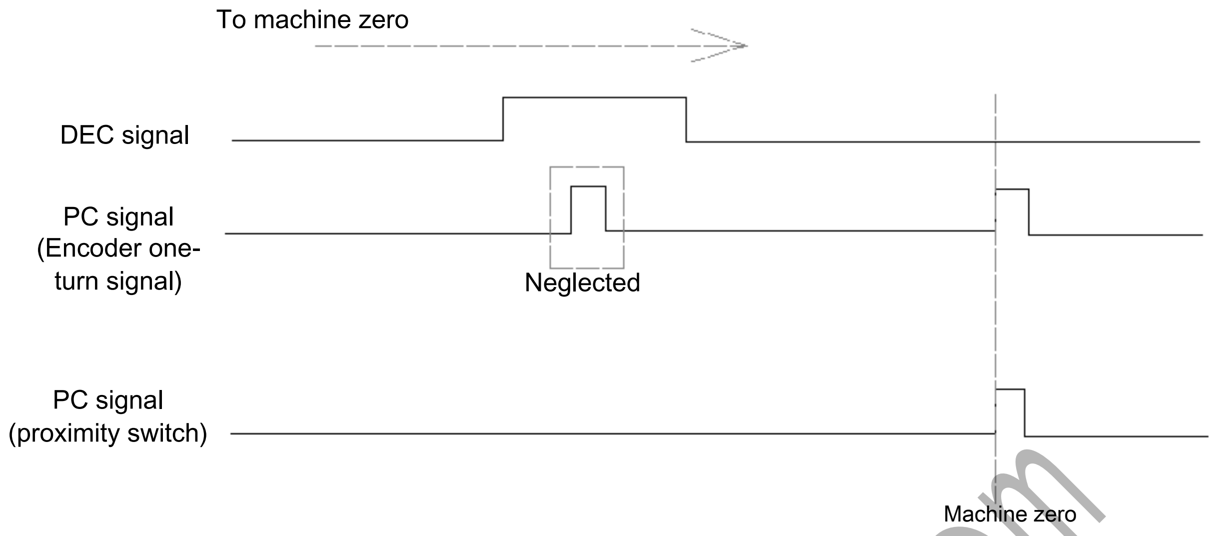


Fig. 2-8 PC signal wave

Note: During the machine zero return, the CNC detects the jumping of the PC signal to judge the reference point after the DEC switch is detached, which is active in both rise edge and trailing edge of the wave.

b) The wiring of NPN Hall element taken as both DEC signal and zero signal is shown in Fig. 2-9:

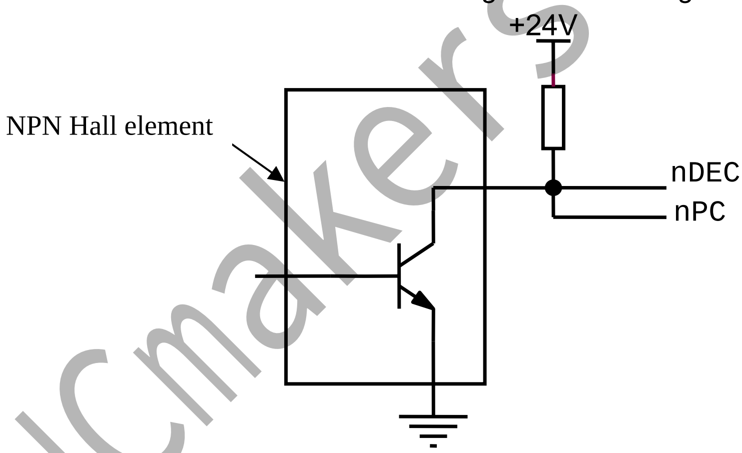


Fig. 2-9 Wiring by a NPN Hall element

c) The wiring of PNP Hall elements taken as both DEC signal and zero signal is shown in Fig. 2-10:

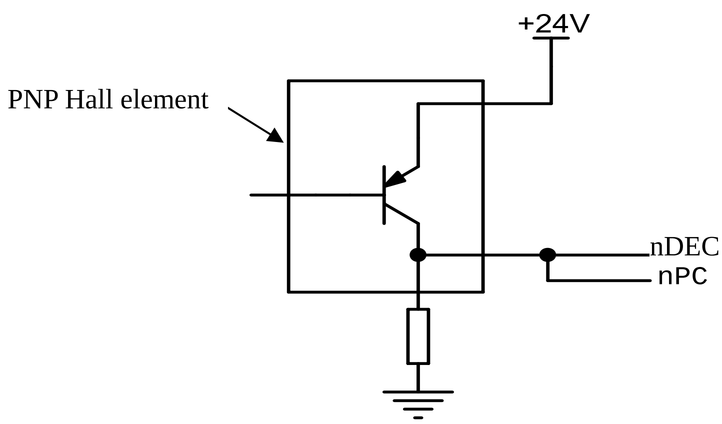
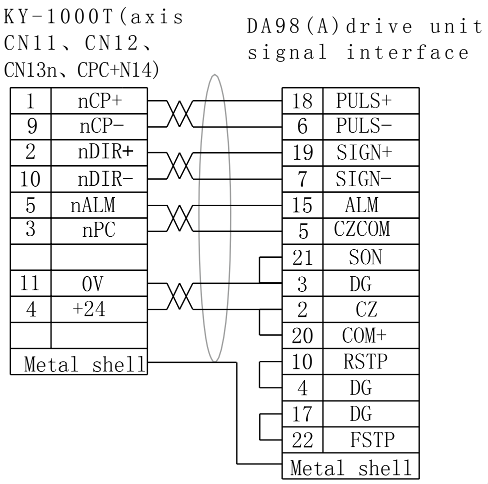


Fig. 2-10 Wiring by a PNP Hall element

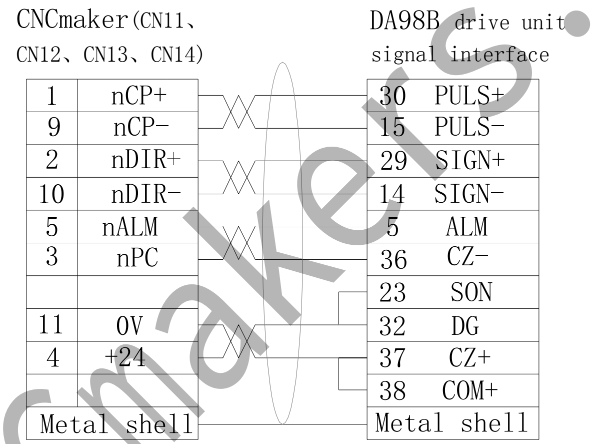
2.1.7 Connection to a drive unit

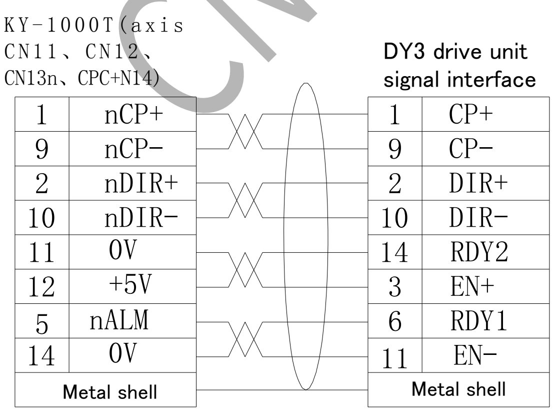
C1000T is connected with our drive unit shown in Fig. 2-11:

C1000T is connected with DA98(A)drive unit



C1000T is connected with DA98B drive unit





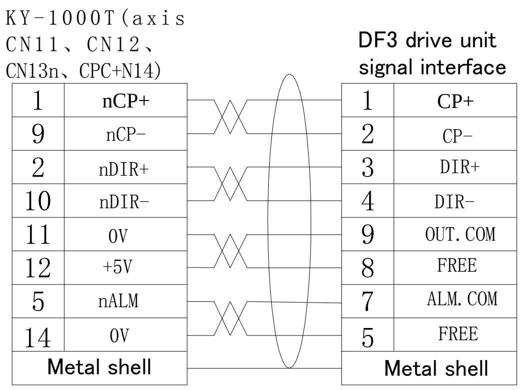


Fig. 2-11 C1000T CNC systems are connected with CNCmakers drive unit

2.2 Being connected with spindle encoder

2.2.1 Spindle encoder interface definition

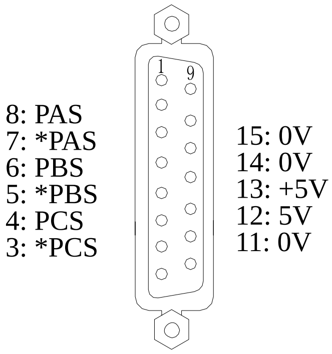


Fig.2-12 CN21 encoder interface
(15-core D type male socket)

Name	Explanation
*PAS/PAS	Encoder A phase pulse
*PBS/PBS	Encoder B phase pulse
*PCS/PCS	Encoder C phase pulse

2.2.2 Signal explanation

*PCS/PCS,*PBS/PBS,*PAS/PAS are the encoder C, B, A phases differential input signals respectively, which are received by 26LS32; *PAS/PAS,*PBS/PBS are orthogonal square wave with phase shift 90°and their maximum signal frequency is less than 1MHz; the encoder pulses for C1000T are set at will by parameter, the setting range is from 100 to 5000.

Its interior circuit is shown in Fig. 2-13: (n=A, B, C)

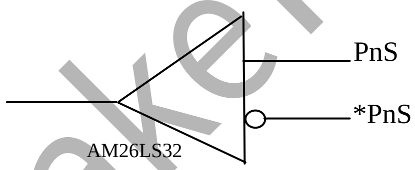


Fig. 2-13 Encoder signal circuit

2.2.3 Being connected with spindle encoder interface

C1000T is connected with spindle encoder shown in Fig. 2-14, and it uses twisted pair cables. (exemplified by CHANGCHUN YIGUANG ZLF-12-102.4BM-C05D encoder):

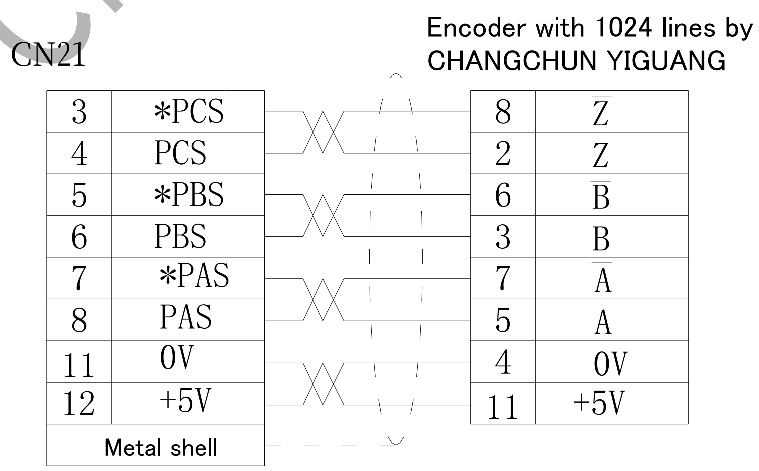


Fig. 2-14 C1000T is connected with the encoder

2.3 Being connected with MPG (Manual Pulse Generator)

2.3.1 MPG interface definition

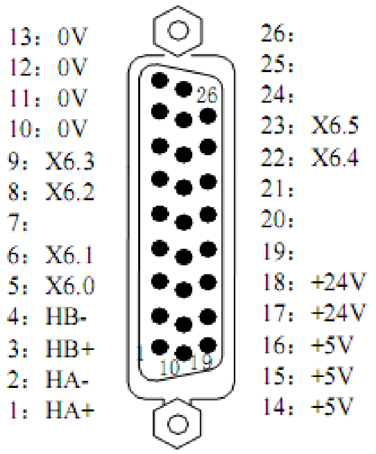


Fig. 2-15 CN31 MPG interface
(26-core DB type male socket)

Signal	Explanation
HA+、HA- HA+、HA-	MPG A phase signal MPG B phase signal
X5.0	X MPG axis selection
.X5.1	Y MPG axis selection
X5.2	Z MPG axis selection
X5.3	increment×1
X5.4	increment×10
X5.5	increment×100
+24V VCC, GND	DC power supply

2.3.2 Signal explanation

HA,HB are the MPG A, B phase input signals respectively. Their interior circuit is shown in Fig. 2-16:

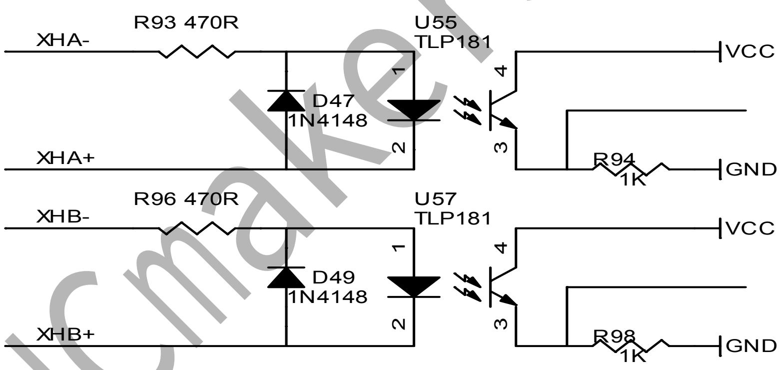
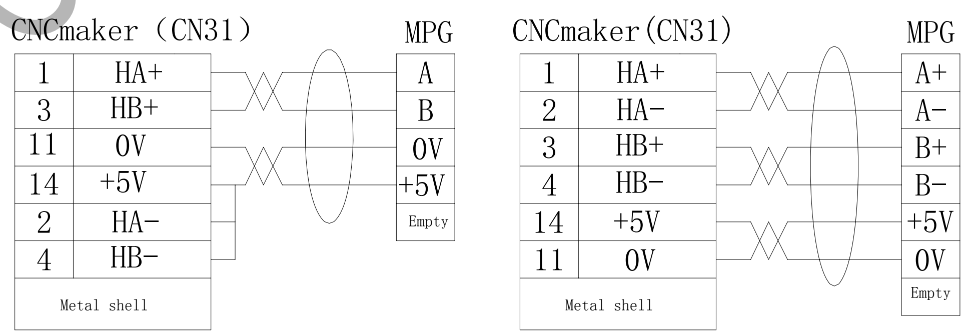


Fig. 2-16 MPG signal circuit

C1000T is connected with MPG shown in Fig. 2-17:

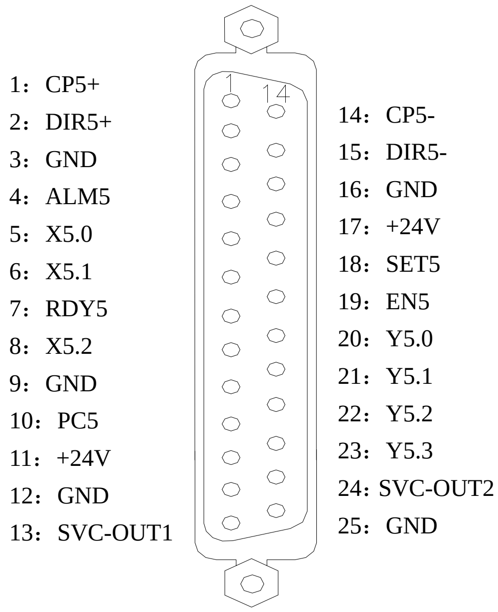


Single-terminal input differential input

Fig. 2-17 CNCmaker is connected with MPG

2.4 Spindle interface

2.4.1 Spindle interface definition



CP5+,CP5-	Spindle pulse signal
DIR5+,DIR5-	Spindle direction signal
ALM5	Spindle alarm signal
RDY5	Spindle ready signal
PC5	Spindle zero signal
SVC-OUT1	Analog voltage output 1
SVC-OUT2	Analog voltage output 2
SET5	Spindle setting signal
EN5	Spindle enabling signal
X5.0~X5.3	PLC address，only LOW is valid
Y5.0~Y5.3	PLC address

Fig. 2-18 CN15 spindle interface（DB25 female）

Note 1: It is valid when PC5 is connected with 0V, and it is different with other feed axes(it is valid when PC of CN11～CN14 axis interface is connected with +24V).

Note 2: They are valid when X5.0～X5.3 are connected with 0V, and they are different with other input signals(other are valid when they are connected with +24V).

Note 3: The internal circuit of PC5，X5.0~X5.3 signals are shown below:

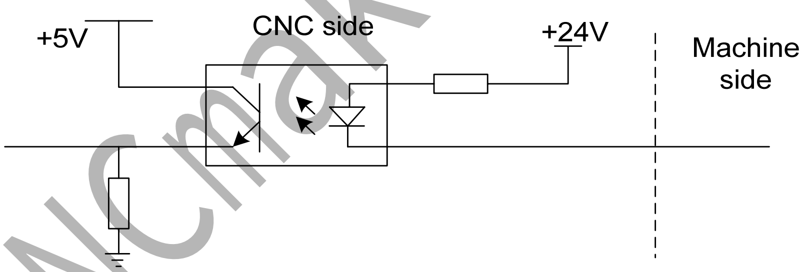


Fig. 2-19 PC5, X5.0～X5.3 circuit

2.4.2 Connection to inverter

The analog spindle interface SVC may output 0~10V voltage, its interior signal circuit is shown in Fig. 2-20:

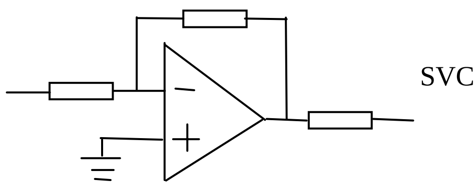


Fig. 2-20 SVC signal circuit

C1000T is connected with the inverter shown in Fig. 2-21:

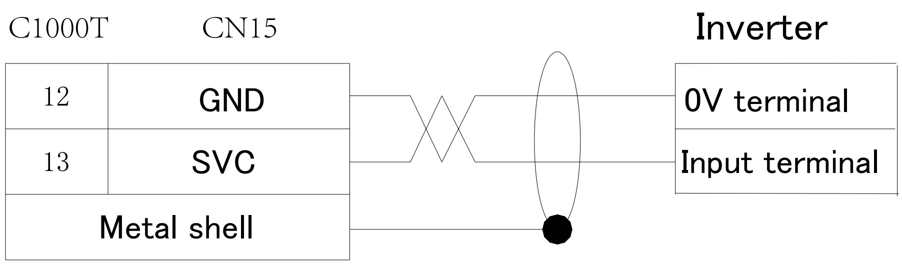
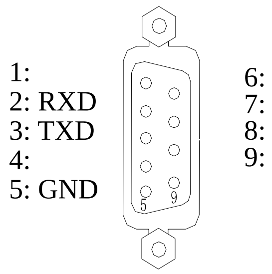


Fig. 2-21 C1000T is connected with the inverter

2.5 C1000T being connected with PC

2.5.1 Communication interface definition



Signal	Explanation
RXD	Receiving data
TXD	Transmitting data
GND	Signal grounding

Fig. 2-22 CN51 communication interface (DB9-female)

2.5.2 Communication interface connection

C1000T can perform the communication by CN51 and PC(optional communication software). C1000T is connected with PC shown in Fig 2-23A:

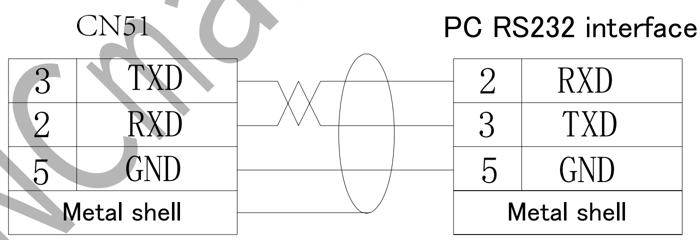


Fig. 2-23A C1000T is connected with PC

The communication between a C1000T system to another C1000T system can be done by CN51 shown in Fig. 2-23B:

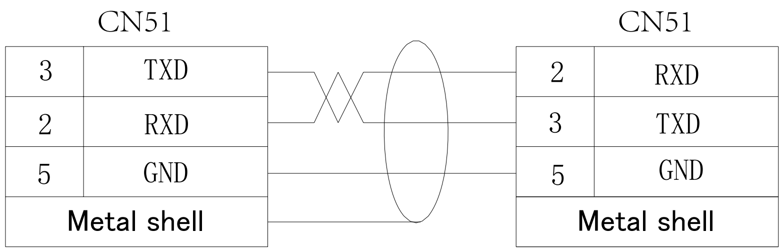


Fig. 2-23B Communication between a C1000T system and another C1000T system

2.6 Power interface connection

The power box interface has been done for its delivery from factory, and the user only need to connect it to a 220V AC power in using.

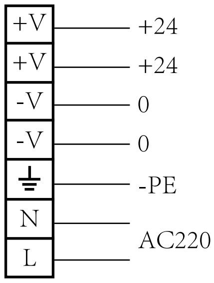


Fig. 2-24 system power interface CN1

2.7 I/O interface definition

> **Note!**
>
> The I/O function significances of the unlabelled fixed addresses of this C1000T turning machine CNC system are defined by PLC programs (ladders), and they are defined by the machine builder when matching with a machine, please refer to the manual by the machine builder.
>
> The fixed address I/O function not be marked are described for C1000T PLC.

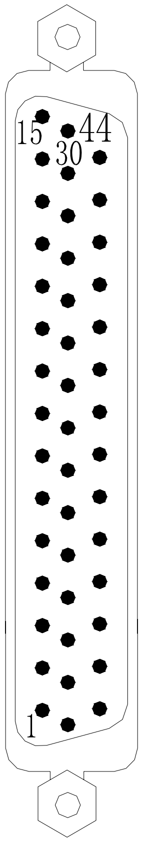


Fig.2-25 input interface(CN61)

Pin	Address	Function	Explanation
21～24	0V	Power supply interface	
17～20 25～28	Floating	Floating	Floating
1	X0.0	SAGT	Guard door check signal
2	X0.1	SP	External feed hold signal
3	X0.2	DIQP	Chuck input signal
4	X0.3	DECX(DEC1)	X deceleration signal
5	X0.4	DITW	Tailstock control signal
6	X0.5	ESP	External emergency stop signal
7	X0.6	PRES	Pressure check signal
8	X0.7	T05	Tool signal /OV1
9	X1.0	T06/ strobe	Tool signal /OV2/strobe signal
10	X1.1	T07/ pregraduation	Tool signal /OV3/pregraducation proximity switch
11	X1.2	T08/ tool post worktable overheat	Tool signal/OV4/ tool worktable overheat check
12	X1.3	DECZ(DEC3)	Z deceleration signal
13	X1.4	ST	External cycle start signal
14	X1.5	M41I	Shifting gear to 1st gear in-position
15	X1.6	M42I	Shifting gear to 2nd gear in-position
16	X1.7	T01	Tool signal
29	X2.0	T02	Tool signal
30	X2.1	T03	Tool signal
31	X2.2	T04	Tool signal
32	X2.3	DECY(DEC2)	Y deceleration signal
33	X2.4	DEC4	4th deceleration signal
34	X2.5	DEC5	5th deceleration signal
35	X2.6	TCP	Tool post clamping signal
36	X2.7	AEY/BDT	External skip
37	X3.0	LMIX	X overtravel
38	X3.1	LMIY	Y overtravel
39	X3.2	LMIZ	Z overtravel
40	X3.3	WQPJ	Inner chuck releasing/outer chuck clamping in-position signal
41	X3.4	NQPJ	Inner chuck clamping/outré chuck releasing in-position signal
42	X3.5	SKIP	G31 skip signal
43	X3.6	AEX	X tool measure position arrival signal （G36）
44	X3.7	AEZ	Z tool measure position arrival signal（G37）

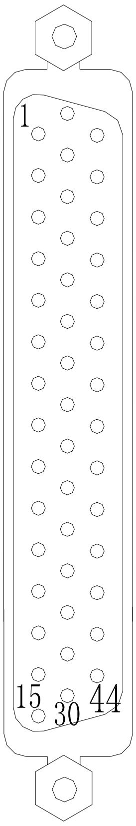


Fig.2-26 output interface (CN62)

Pin	Address	Function	Explanation
17,18,19, 26,27,28	0V	Power supply interface	Power supply 0V terminal
20～25	+24V	Power supply interface	Power supply ＋24V terminal
1	Y0.0	M08	Cooling output
2	Y0.1	M32	Lubricating output
3	Y0.2		Reserved
4	Y0.3	M03	Spindle rotation (CCW)
5	Y0.4	M04	Spindle rotation (CW)
6	Y0.5	M05	Spindle stop
7	Y0.6		reserved
8	Y0.7	SPZD	Spindle brake
9	Y1.0	S1/M41	Spindle machine gear output 1
10	Y1.1	S2/M42	Spindle machine gear output 2
11	Y1.2	S3/M43	Spindle machine gear output 3
12	Y1.3	S4/M44	Spindle machine gear output 4
13	Y1.4	DOQPJ(M12)	Chuck clamping output
14	Y1.5	DOQPS(M13)	Chuck releasing output
15	Y1.6	TL+	Tool post CCW rotation
16	Y1.7	TL-	Tool post CW rotation
29	Y2.0	TZD	Tool post worktable brake
30	Y2.1	INDXS	Tool post worktable graduation coil
31	Y2.2	YLAMP	Three-color lamp-yellow
32	Y2.3	GLAMP	Three-color lamp -green
33	Y2.4	RLAMP	Three-color lamp -red
34	Y2.5	DOTWJ(M10)	Tailstock forward
35	Y2.6	DOTWS(M11)	Tailstock backward
36	Y2.7		reserved
37	Y3.0		reserved
38	Y3.1		reserved
39	Y3.2	UO0	User macro output 0
40	Y3.3	UO1	User macro output 1
41	Y3.4	UO2	User macro output 2
42	Y3.5	UO3	User macro output 3
43	Y3.6	UO4	User macro output 4
44	Y3.7	UO5	User macro output 5

Note 1: Various functions can be defined to some of the input and output interfaces, and they are indicated by "/"sign in the table above.

Note 2: If output function is active, the output signal is through on to 0V. If output function is inactive, the output signal is cut off by high impedance.

Note 3: If input function is active, the input signal is through on to +24V. If input function is inactive, the input signal is cut off with it.

Note 4: The effectiveness of +24V, COM terminals are equivalent to those of the CNCmakers Limited-v power box terminals that have the same names.

2.7.1 Input signal

Input signal means the signal from machine to CNC, when this signal is through on with +24V, the input is active; when it is off with +24V, the input is inactive. The contact of input signal at machine side should meet the following conditions:

Capacity of the contact: DC30V, 16mA above

Leakage current between contacts in open circuit: 1mA below

Voltage drop between contacts in closed circuit: 2V below (current 8.5mA, including cable voltage drop)

There are two external input types for input signals: one type is input by contact switch whose signals are from keys, stroke switch and contacts of relay at machine side, as shown in Fig. 2-27:

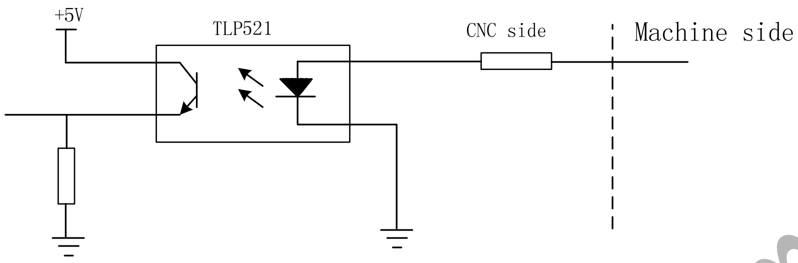


Fig. 2-27

The other type is input by switch with no contacts (transistor) as shown in Fig. 2-28A, 2-28B:

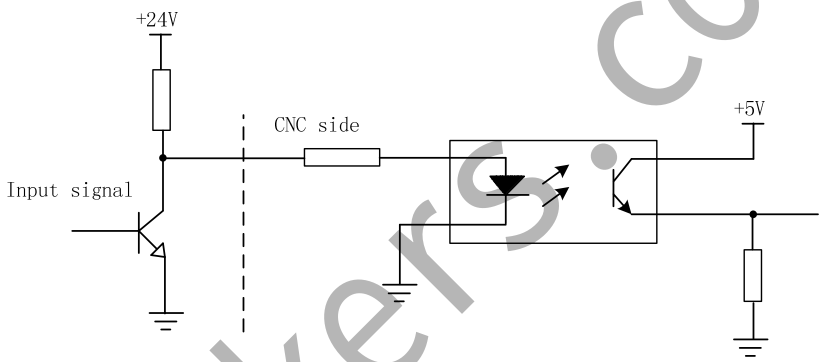


Fig. 2-28A NPN connection

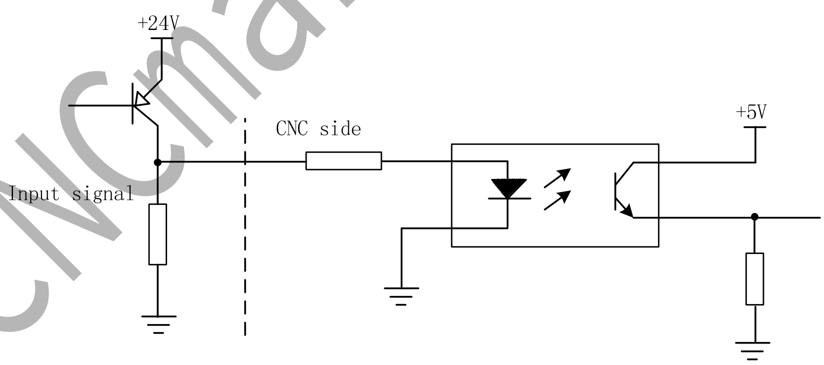


Fig. 2-28B PNP connection

The input interface signals defined by PLC of C1000T-V system involve XDEC, ZDEC, ESP, ST, SP/SAGT, BDT/DITW, DIQP, OV1～OV8, T01～T08, TCP and so on.

2.7.2 Output signal

The output signal is used for the machine relay and indicator, if it is through on with 0V, the output function is active; if it is off with 0V, the output function is inactive. There are 36 digital volume outputs that they all have the same structure in I/O interface as shown in Fig. 2-29:

Volume III Connection

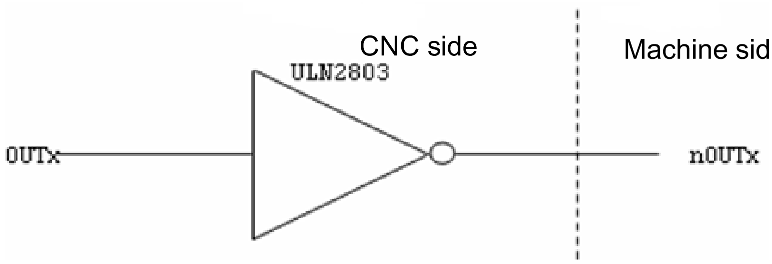


Fig. 2-29 Circuit for digital volume output module

The logic signal OUTx output from the main board is sent to the input terminal of inverter (ULN2803) via a connector. And there are 2 types for nOUTx output: 0V, or high impedance. Its typical application is as follows:

- To drive LED

A serial resistance is needed to limit the current (usually 10mA) that goes through the LED by using ULN2803 output to drive LED, which is shown in Fig. 2-30:

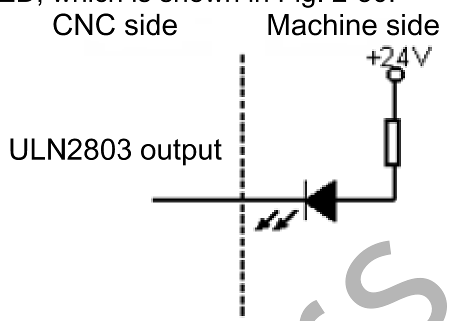


Fig. 2-30

- To drive filament indicator

An external preheat resistance is needed to decrease the current impact at power on by using ULN2803 output to drive filament indicator, and this resistance value should be within a range that the indicator cannot be lighted up as shown in Fig. 2-31:

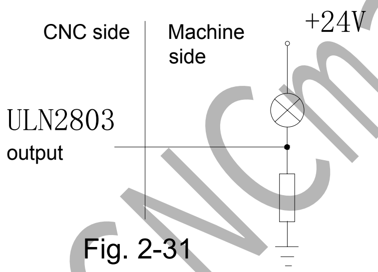


Fig. 2-31

- To drive inductive load (such as relay)

To use ULN2803 output to drive an inductive load, it requires to connect a freewheeling diode near the coil to protect output circuit and reduce interference as shown in Fig. 2-32:

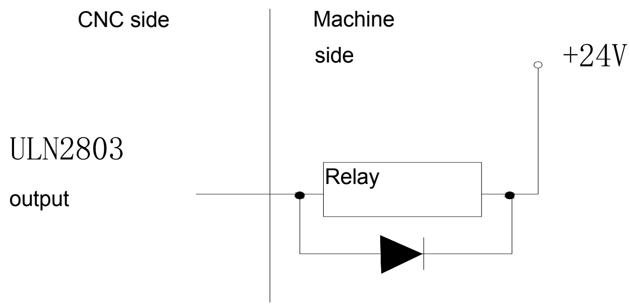


Fig.2-32

The meaning of the output signal in the I/O interface is defined by the PLC program, and the output signals defined by the standard PLC program include S1～S4(M41～M44)、M3～M5、M8、M10、M11、M32、TL-、TL+、U00～U05、DOQPJ、DOQPS、SPZD and so o.

2.8 I/O function and connection

Note！

The I/O function significance of this C1000T turning machine CNC system is defined by PLC programs (ladders), and they are defined by the machine builder when matching with a machine, please refer to the manual by the machine builder.

The fixed address I/O function not be marked are described for C1000T PLC.

2.8.1 Stroke limit and emergency stop

- **Relevant signal**

ESP: emergency stop signal, alarm issued if the system is not connected with +24V

LMIX：X overtravel limit check input

LMIY：Y overtravel limit check input

LMIZ：Z overtravel limit check input

0	0	0
Pin		

ESP							
CN61.6							

- **Signal diagnosis**

Signal	ESP	LM	LMIY	LMIZ
Diagnosis address	X0.5	X		X3.2
Interface pin	CN61.6	CN61.37	CN61.38	CN61.39

- **Control parameter**

Bit parameter

1	7	6

				MESP		

KYP =0: Check ESP signal

=1: Do not check ESP signal

- **PLC bit parameter**

K	1	0

LMIT	LMIS						

LMIT =1：Travel limit check function of each axis is valid.

=0：Travel limit check function of each axis is invalid

LMI =1：The system alarms for overtravel when the travel limit check signal is not connected with +24V.

=0：The system alarms for overtravel when the travel limit check signal is connected with +24V

- **Signal connection**

The ESP signal circuit is shown in Fig.2-33:

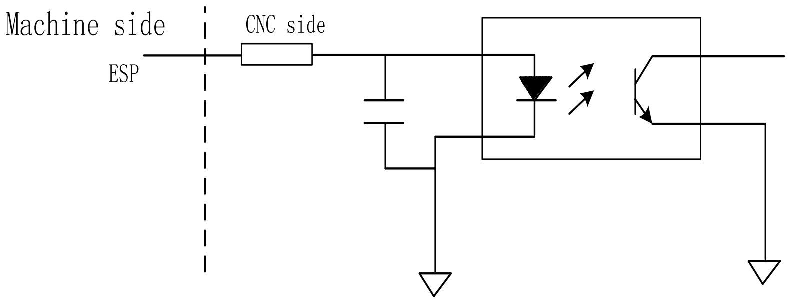


Fig. 2-33 The ESP signal circuit

- **Machine external connection**

(1) The series connection between the emergency stop and travel switch is shown in Fig. 2-34A:

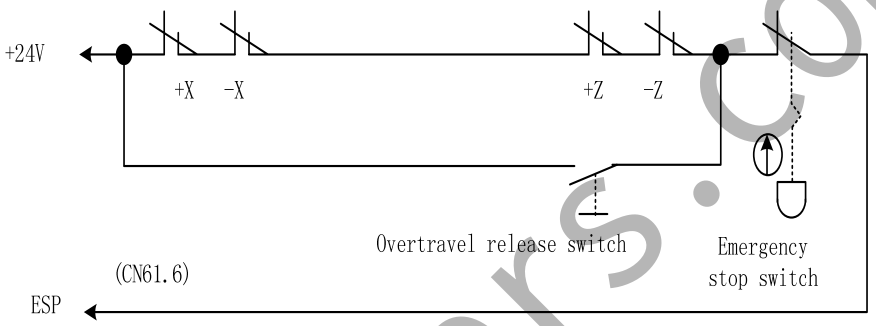


Fig.2-34A Series connection between emergency stop and travel switch

Bit paramete

k	1	1		CHET	TCPS	CTCP	TSGN	CHTB	CHTA

CHTA: tool change mode selection bit 0
CHTB: tool change mode selection bit 1 (see following table)

CHTB	CHTA	Tool post type
0	0	Standard tool change mode B
0	1	Standard tool change mode A
1	0	Yantai AK31
1	1	Unused

TSGN =0 : tool signal HIGH(turn on +24V) is valid
=1 : tool signal LOW(turn off +24V) is valid
CTCP =0 : do not check tool post locking signal
=1 : check tool post locking signal
TCPS =0 : tool post locking signal LOW(turn off +24V) is valid
=1 : tool post locking signal HIGH(turn on +24V) is valid

CHET=0 ： do not check tool signal after the tool change is completed

=1 ： check tool signal after the tool change is completed

CHOT=0 ： do not check tool post overheat

=1 ： check tool post worktable overheat

Time upper limit for changing a tool

0	7	8		TLMAXT

Time upper limit for moving max. tools in tool change

0	8	2		T1TIME

Tool change time T1: Tool post delay time from CCW stop to CW output (ms)

0	8	4		TMAX

Total tools number

0	8	5		TCPTIME

Tool change time T2: tool post CW clamping time

- **Signal connection**

1. The T01～T08, TCP signals input are employed with photocoupler, its interior circuit is shown in Fig. 2-35:

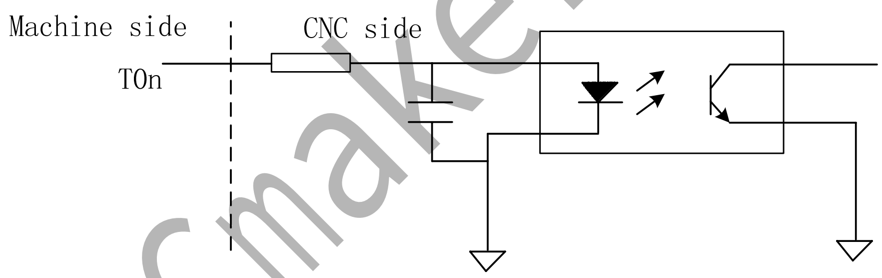


Fig. 2-35

2. TL+, TL- are tool post CCW/CW output signal, its interior circuit is shown in Fig.2-36:

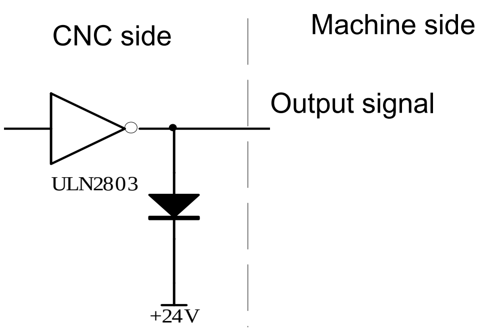


Fig. 2-36

3. The external circuit of the tool number signal is shown in Fig. 2-37, when the tool number signal is low level active, it requires an external pull-up resistor.

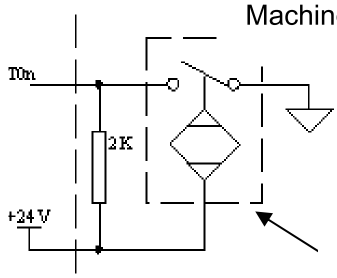


Fig. 2-37

- **Function description (defined by standard PLC program)**

The control sequence and control logic of the tool change are defined by PLC program. There are 4 tool change modes defined as follows by standard PLC program:

1.**CHTB=0，CHTA =0，CHET =0: tool change mode B**

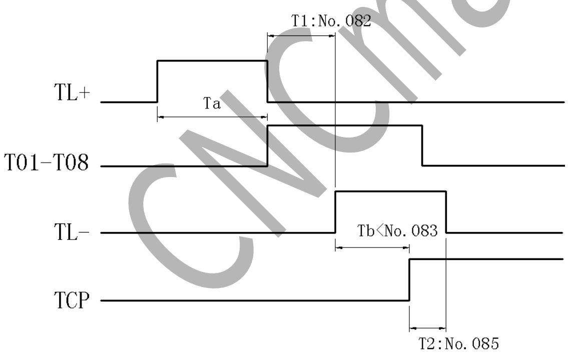
①During the tool change process, CNC outputs TL+ signal until the tool in-position signal is detected, then CNC turns off TL+ signal output and outputs TL- signal after a delay time specified by data parameter No.082. Then CNC detects TCP signal till it is detected, the CNC turns off TL- signal after a delay time specified by the data parameter No.085. So the tool change is over.

②When CHET(K0011.5) is set to 1(check tool signal after the tool change ends) and the tool post (CCW) rotation time ends to confirm whether the current tool input signal is consistent with the current tool No., if not, the system alarms.

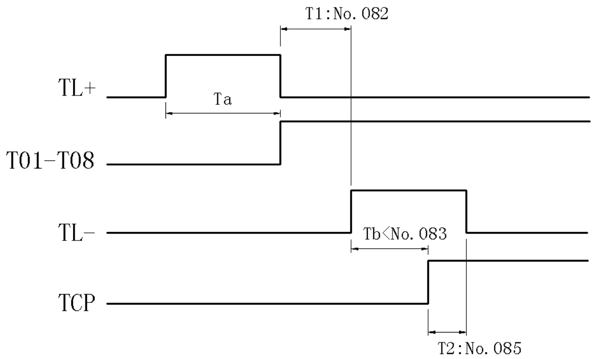
③ The tool change process ends.

④ After the system outputs the tool post rotation(CCW)signal, if the CNC doesn't receive the TCP signal within the time set by data parameter No.083, an alarm will be issued and the TL- signal will be turned off.

⑤ When the tool post has no tool post locking signal, CTCP(K0011.3) is set to 0, at the time, the system does not check the tool post locking signal.



or



Sequence of tool change mode

B **2.CHT =1，tool change mode A**

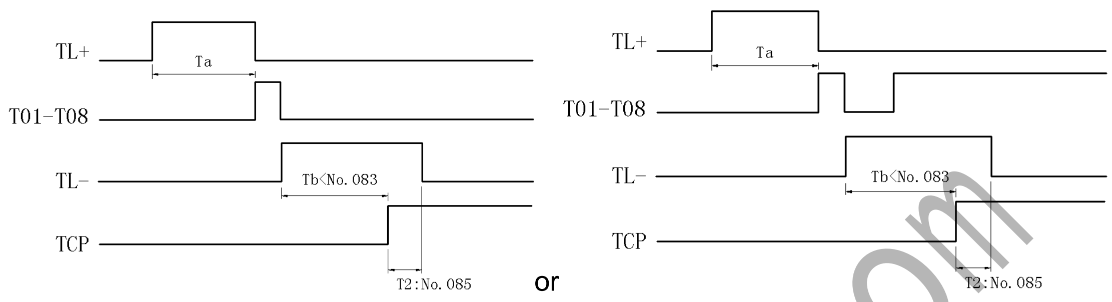
① After the tool change is executed, the system outputs the tool rotation(CW)signal TL+ and checks the tool in-position signal, and then after it has checked the tool signal and closes TL+, last checks whether the tool signal skips, if done, it outputs the tool rotation (CCW) signal TL-. Then, the system checks the locking signal TCP, it delays the time set by No.085 and closes TL- after it has received the TCP;

② When CHET（K0011.5）is set to 1（check tool signal after the tool change ends），the system confirms whether the current tool input signal is consistent with the current tool number after the tool post (CCW) rotation time ends, if not, the system alarms;

③ The tool change process ends.

④ After the system outputs the too rotation(CCW) signal, when it has not received TCP signal in the time set by No.83, it alarms and closes TCP signal.

⑤ If the tool post has no locking signal, CTCP（K0011.3）is set to 0，at the time, the system does not check the tool post locking signal.



Sequence of tool change mode A

Note 1: No.082 setting is invalid, the system does not check in the delay time between the tool post (CW)stop and the tool post (CCW) rotation locked.

Note 2: Except for No.82, relative parameter setting and function of other tool post control are valid.

2.8.3 Machine zero return

- Relative signal

DECX：X deceleration signal；

DECY：Y deceleration signal；

DECZ：Z deceleration signal；

DEC4：4th deceleration signal；

DEC5：5th deceleration signal；

PCX：X zero signal；

PCY：Y zero signal；

PCZ：Z zero signal；

PC4：4th zero signal；

PC5：5th zero signal；

- Diagnosis data

0 0 0				DEC5	DEC4	DECZ	DECY	DECX
Interface pin				CN61.34	CN61.33	CN61.12	CN61.32	CN61.4

- Control parameter

K 2 2	DEC4T	DECY	DECZ	DECX				

DEC4T=0: 4TH decelerates as DEC signal is LOW level
=1: 4TH decelerates as DEC signal is HIGH level
DECY=0: Y decelerates as DEC signal is LOW level
=1: Ydecelerates as DEC signal is HIGH level
DECZ=0: Z decelerates as DEC signal is LOW level
=1: Z decelerates as DEC signal is HIGH level
DECX=0: X decelerates as DEC signal is LOW level
=1: X decelerates as DEC signal is HIGH level

0 0 6			ZMOD					

ZMZ =1: Z machine zero return block before
=0: Z machine zero return block after

0 0 7								ZPLS

ZCX =1: The deceleration signal and one-turn signal in parallel during machine zero
=0: The deceleration signal in parallel during machine zero

0 1 2								ISOT

ISOT =1: Manual rapid traverse active prior to machine zero return after power on
=0: Manual rapid traverse inactive prior to machine zero return after power on

1 8 3				MZR5	MZR4	MZRY	MZRZ	MZRX

ZMRn =1: The direction of machine zero return is negative

Volume III Connection

=0: The direction of machine zero return is positive

● Data parameter

0	3	3	ZRNFL

ZRNFL =Low rate of axes reference return

1	1	3	ZRNFH

ZRNFH =High-speed of X,Z axes reference return

1	7	7	ZRNFHY

ZRNFY =High-speed of Y axes reference return

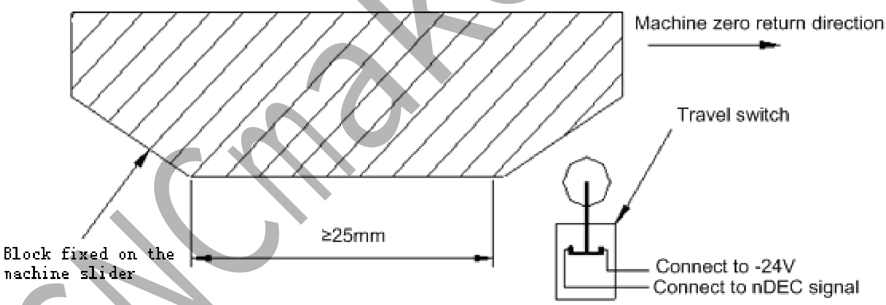
1	7	8	ZRNFH4

ZRNFH4 =High-speed of 4TH axes reference return

1	7	9	ZRNFH5

ZRNFH5 =High-speed of 5TH axes reference return

The connection to ACservo motor: using a travel switch and servo motor one-turn signal separately



② The circuit of deceleration signal

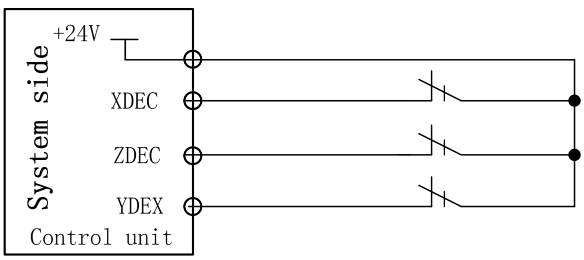
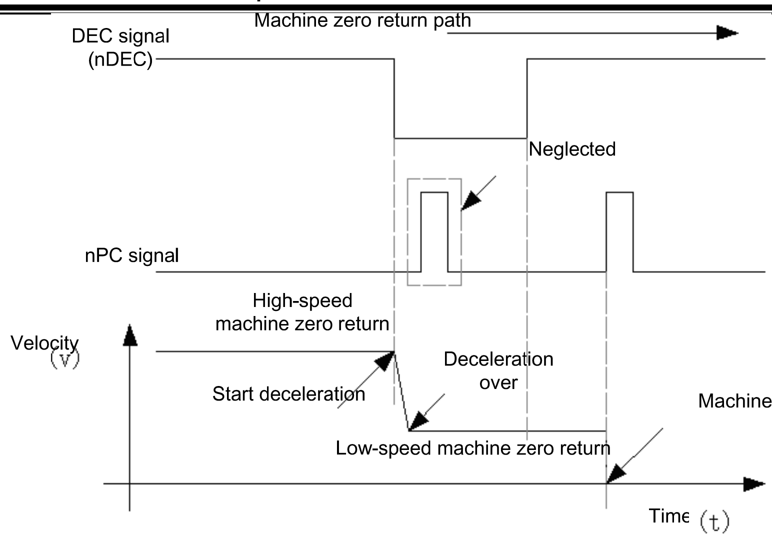


Fig. 2-40

③ Sequence of machine zero return

When the BIT0 (ZMX) of the bit parameter No.006 is set to 0, and the BIT5(DECI) of the bit parameter No.004 is set to 0, the system chooses the machine zero return mode B, and the deceleration signal low level is active.

So the sequence of machine zero return mode B is shown as follows:



Return process of machine zero mode B

A: Select Machine zero mode, press the manual positive or negative feed key(machine zero return direction set by bit parameter No.183), the corresponding axis moves to the machine zero by a rapid traverse speed. As the axis press down the deceleration switch to cut off deceleration signal, the feeding slows down immediately, and it continues to run in a fixed low speed.

B: When the deceleration switch is released, the deceleration signal contact is closed again. And CNC begins to detect the encoder one-turn signal (PC), if this signal level skips, the motion will be halted. And the corresponding zero return indicator on the operator panel lights up for machine zero return completion.

When the BIT0 (ZMX) of the bit parameter No.006 are both set to 1, and the BIT5(DECI) of the bit parameter No.004 is set to 0, it chooses the machine zero return mode C, and the deceleration signal low level is active.

So the sequence of machine zero return mode C is shown as follows:

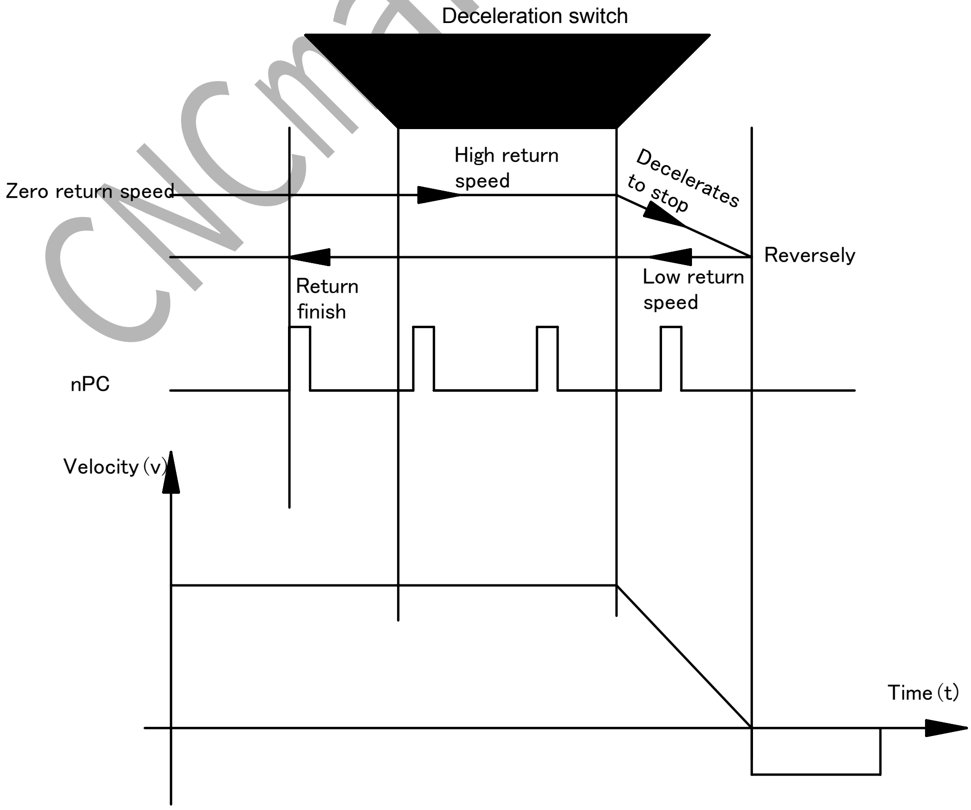


Fig. 2-41-b

Volume III Connection

Return process of machine zero mode C

A: Select Machine zero mode, press the manual positive or negative feed key (return direction set by bit parameter No.183), the corresponding axis moves to the machine zero by a rapid traverse speed. As the axis press down the deceleration switch to cut off deceleration signal, the feeding keeps rapid rate and depart from the deceleration switch, when the DEC signal contact is closed, the feeding slows down to zero, then run reversely to return to machine zero in a low speed.

B: In the reverse running, it presses down the deceleration switch to cut off the DEC signal contact and continues returning; as it departs from the deceleration switch, the deceleration signal contact is closed again. And CNC begins to detect the encoder one-turn signal (PC), if this signal level skips, the motion will be halted. And the corresponding axis zero return indicator on the operation panel lights up for zero return completion.

2.8.4 Spindle control

- **Relevant signal (by standard PLC program)**

Type	Symbol	Interface	Address	Function	Remark
Input signal	SAR	CN15.6	X5.1	Spindle speed arrival signal	It is valid when 0V is input
	SALM	CN15.4	X5.3	Spindle abnormity alarm input	
Output signal	M03	CN62.4	Y0.3	Spindle rotation(CCW)	
	M04	CN62.5	Y0.4	Spindle rotation(CW)	
	M05	CN62.6	Y0.5	Spindle stop	
	SCLP	CN62.7	Y0.6	Spindle clamped	
	SPZD	CN62.8	Y0.7	Spindle brake	
	SVF	CN62.37	Y3.0	Spindle servo OFF	
	SFR	CN15.22	Y5.2	Spindle rotation(CW)	Its function is consistent with that of M03
	SRV	CN15.23	Y5.3	Spindle rotation(CCW)	Its function is consistent with that of M04
Command format	M03			Spindle rotation(CCW)	
	M04			Spindle rotation(CW)	
	M05			Spindle stop	
	M20			Spindle clamped	They are valid in analog spindle
	M21			Spindle released	

- **Control parameter**

Bit parameter

K10							RSJG	

RSJG =1: CNC not turn off M03, M04, M08, M32 output signals when pressing RESET key;

=0: CNC turns off M03, M04, M08, M32 output signals when pressing RESET key.

Volume III Connection

K	1	7				SALM				

SALM =1：The system alarms when the spindle abnormal detection input signal is LOW (it is turned off with 0V);

=0：The system alarms when the spindle abnormal detection input signal is HIGH (it is turned on with 0V) .

1	7	5		SAR						

Bit6 1：The spindle SAR signal is checked before cutting;

0：The spindle SAR signal is not checked before cutting.

Data parameter

0	7	2	SAR_DELEY

Delay check time (ms) of the spindle speed arrival.

0	8	0	MTIME

Execution time of M code.

● **Function description（defined by standard PLC program）**

① After the CNC is turned on, when M05 output is valid, M03 or M04 is executed, M03 or M04 output is valid and remains, at the time, M05 output is closed; when M03 or M04 output is valid, M05 is executed, M03 or M04 is closed, M05 output is valid and remains; the spindle brake SPZD signal output delay is set by No.087（delay time from the spindle stop signal output to the spindle brake SPZD signal output）, the hold time of the brake signal is set by No. 089（the spindle brake output time）.

② When M03（M04）output is valid, M04（M03）is executed, the alarm occurs.

③ When №175.6 is set to 1, the system checks whether the speed arrival signal SAR is valid, if it is valid, the system normally runs, if not, “Check speed arrival……” is displayed.

④ When the spindle speed command and the cutting feed command are in the same block, the system executes the delay check for SAR signal to avoid that the CNC starts the cutting based on the last spindle speed arrival signal SAR, the delay time is set by No.072.

⑤ DC00 is the spindle zero-speed output range. When the actual spindle speed is not more than DC00 setting value, the system defaults the spindle speed to be zero, and can release the chuck, close the hydraulic and other operations.

⑥ M20 and M21 are separate the spindle clamping and releasing command, used to clamp the spindle after positioning, which can avoid that the spindle rotates because of the force in drilling or tapping.

⑦ When M20 is executed, the spindle is clamped. To avoid that the spindle servo motor has too much current, the system delays the time set by DT23 and then controls the spindle to turn off the servo, at the time, the spindle servo reduces the motor excitation and the position control cannot be executed, but the position check is still enabled and the position is not lost.

⑧ When the spindle moves or rotates, M20(spindle clamped) cannot be executed. After the spindle is clamped, it cannot be rotated or moved, otherwise, the PLC alarms.

⑨ When K17.6 sets the spindle to be clamped, the clamped is the 1st or 2nd spindle, which is set based on the actual conditions.

⑩ SALM（X5.3）is the alarm input signal of the spindle abnormality, the signal and the drive alarm signal of the 5th axis use the same interface. When the 5th axis is valid, the interface is used to the 5th drive alarm; when the 5th axis is invalid, the interface is used to the spindle abnormality alarm.

Note 1: In the emergency stop, it turns off M03, M04, M08 signals, and outputs M05 signal;
Note 2: Whether M03, M04 is cancelled is set by BIT3 of the bit parameter No.009 when CNC is reset.
If Bit 1=0, CNC turns off M03, M04 at reset;
If Bit 1=1, M03, M04 is kept at reset.

2.8.5 Spindle switching volume control

- **Relevant signal(defined by standard PLC program)**

S01～S04: Control signal for spindle speed switching volume, they are compound interfaces defined by standard PLC program, and they share common interfaces with M41～M44, U00～U03.

- **Signal diagnosis**

Signal	S4	S3	S2	S1
Diagnosis address	Y1.3	Y1.2	Y1.1	Y1.0
Interface pin	CN62.12	CN62.11	CN62.10	CN62.09

- **Control parameters**

Bit parameter

001				**ACS**				

Bit4 =1: Analog voltage control of spindle speed
=0: Switching volume control of spindle speed

- **Control logic (defined by standard PLC program)**

S1～S4 output are inactive at power on. If any code of them is executed, the corresponding S signal output is active and held on, and the other S signal outputs are cancelled. S1～S4 outputs are cancelled when executing S00 code, and only one of them is active at a time.

2.8.6 Spindle automatic gearing control

- **Relevant signal (defined by standard PLC program)**

M41～M44：spindle automatic gear shifting output signals. It supports 4-gear spindle automatic gear shifting control when the system selects the spindle analog value control(0～10V analog voltage output)

M41I,M42I：spindle automatic gear shifting No.1, 2 gear in-position signals to support gear shifting in-position check function

- **Signal diagnosis**

Signal	M42I	M41I	M44	M43	M42	M41
Diagnosis address	X1.6	X1.5	Y1.3	Y1.2	Y1.1	Y1.0
Interface pin	CN61.15	CN61.14	CN62.12	CN62.11	CN62.10	CN62.09

Volume III Connection

- **Signal connection**

The circuit for M41~M44 is shown in Fig.2-47:

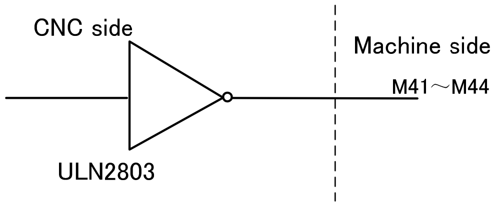


Fig. 2-47

- **Control parameter**

Bit parameter

0	0	1					ACS				

Bit4 =1: Spindle analog volume control, set to 1 if using spindle automatic gearing
=0: Spindle switching volume control

K	1	5					SHT	AGIM	AGIN	AGER

AGER =1: Spindle automatic gearing active
=0: Spindle automatic gearing inactive
AGIN =1: Detect M41I, M42I signal when shifting to gear 1, 2
=0: Not detect M41I, M42I signal when shifting to gear 1, 2
AGIM =1: Active when M41I, M42I signals disconnecting to +24V
=0: Active when M41I, M42I signals connecting to +24V
SHT =1：spindle gear power-down executes the memory
=0：spindle gear power-down does not execute the memory

Data parameter

0	3	7	GRMAX1
0	3	8	GRMAX2
0	3	9	GRMAX3
0	4	0	GRMAX4

GRMAX1,GRMAX2, GRMAX3, GRMAX4: The respective max. speeds of spindle gear 1, 2, 3, 4 when analog voltage output is 10V. Spindle speeds for M41, M42, M43, M44 when spindle automatic gearing is active.

0	6	5	SFT1TME

Delay time 1 when automatic gearing signal output, see function description.

0	6	6	SFT2TME

Delay time 2 when automatic gearing signal output, see function description.

0	6	7	SFTREV

Output voltage of spindle gearing (0~10000, unit: mV)

- **Function description (defined by standard PLC program)**

The spindle automatic gearing is active only under the spindle analog voltage control (BIT4 of the bit parameter No.001 set to 1) and the BIT 7 of the bit parameter No.164 is set to 1; if the spindle auto gearing is inactive, alarm will be issued when M41～M44 is being executed and only one of them is active at a time.

When spindle auto gearing is used to control automatic spindle mechanical gear switching, as CNC executes S□□□□ code, it calculates the analog voltage output to spindle servo or frequency inverter based on the parameter of the current gear by M4n (M41～M44 to data parameters No.037~No.040 respectively) to make the actual speed to be consistent with the S code.

When CNC is powered on, the spindle gear memorizing is set by the BIT1 of bit parameter No.168.

If the BIT4 of bit parameter No.001 is 0, the spindle gear is not memorized at repowering after power down, and the gear 1 will be defaulted, M41~M44 are not output. If BIT4 of bit parameter No.001 is 1, the spindle gear is memorized at repowering after power down.

No gearing is done if the specified gear is consistent with the current gear. If not, gearing will be performed, and the process defined by standard ladders is shown in the following:

①Execute any of M41, M42, M43, M44 codes, output analog voltage to spindle servo or frequency inverter according to a value set by data parameter No.067 (Unit: mV);

②After a delay (gearing time 1) by the data parameter No.065, turn off the original gear output signal and output the new gearing signal;

③If the gear is 1 or 2, and the BIT6 of the bit parameter No.164 is 1, it jumps to ④, or else it jumps to ⑤;

④Check the gear in-position input signal M41I, M42I, it jumps to ⑤ if the gear in-position is done; if not, the CNC waits the gear in-position signal;

⑤After a delay (gearing time 2) by the data parameter No.066, output spindle analog voltage by the current gear according to a value set by data parameter No.037~No.040 (gear 1~4) and finish the gearing.

Note: The output of M41～M44 is held on when CNC is reset or i emergency stop, which is defined by standard PLC ladder.

2.8.7 Spindle eight-point orientation function

- **Related signals（defined by standard PLC program）**

Type	Symbol	Interface	Address	Function	Remark
Input signal	COIN	CN15.8	X5.2	Orientation completion signal	It is valid when 0V is input
	SALM	CN15.4	X5.3	Spindle abnormality alarm signal	
Output signal	STAO	CN62.41	Y3.4	Spindle orientation start signal	
	SP0	CN62.42	Y3.5	Orientation position 0	
	SP1	CN62.43	Y3.6	Orientation position 1	
	SP2	CN62.44	Y3.7	Orientation position 2	
	SFR	CN15.22	Y5.2	Spindle rotation (CW)	
	SRV	CN15.23	Y5.3	Spindle rotation (CCW)	

Note 1: STAO signal and the rotation CCW signal of the 2nd spindle M64 are multiplexed by the interface, so, eight-point orientation function is invalid when the multiple spindle function is valid.

Note 2: STAO, SP0, SP1, SP2 signal and macro output #1102～#1105 are multiplexed by the interface, so, #1102～#1105 is invalid when the eight-point orientation function is valid.

Volume III Connection

2.8.11 External cycle start and feed hold

- **Relevant signal (defined by standard PLC program)**

ST: External cycle start signal, whose function is the same with the CYCLE START key on the machine panel;

SP: External feed hold signal, whose function is the same with the FEED HOLD key on the machine panel, and it shares an interface with SAGT(safety door detect) signal.

- **Signal diagnosis**

Signal	SP	ST
Diagnosis address	X0.1	X1.4
Interface pin	CN61.2	CN61.13

- **Signal connection**

The interior circuit of SP/ST signal is shown in Fig. 2-48:

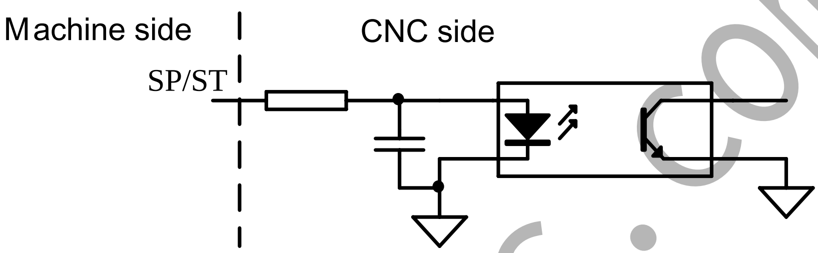


Fig. 2-48

- **Control parameter**

Bit parameter

1	7	6			MST	MSP					

MST =1: External cycle start signal (ST) inactive, it is not the cycle start switch and can be defined by macro.(#1014)

=0: External cycle start signal (ST) active

MSP =1: External feed hold signal (SP) inactive, it is not the stop switch and can be defined by macro command.(#1015)

=0: External feed hold (SP) active, the External feed hold switch is needed, or "feed hold" is displayed by CNC.

- **External connection circuit**

The external connection circuit of SP, ST signals is shown in Fig. 2-49:

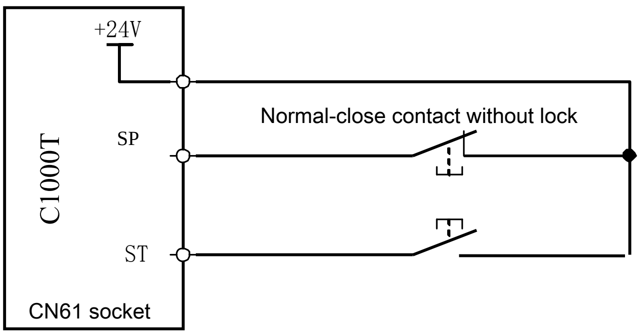


Fig. 2-49

2.8.8 Cooling control

- **Relevant signal (defined by standard PLC program)**

Type	Symbol	Interface	Address	Function	Remark
Output signal	M08	CN62.1	Y0.0	Cooling control output	
Command format	M08			Cooing ON	
	M09			Cooling OFF	

- **Signal connection**

Its internal circuit is shown in Fig. 2-50:

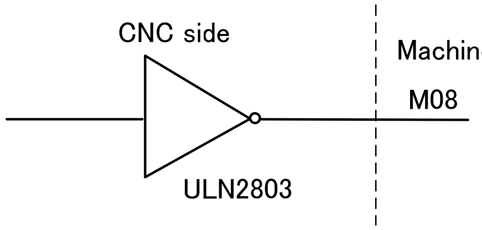


Fig. 2-50 M08 internal circuit

- **Function description (defined by standard PLC program)**

M09 is active, i.e. M08 is inactive, after CNC power on. To execute M08, M08 output is active and cooling is turned on; to execute M09, M08 output is cancelled and cooling is turned off.

Note 1: M08 output is cancelled at CNC emergency stop.

Note 2: Whether M08 is cancelled is set by BIT3 of the bit parameter No.009 when CNC is reset.

When Bit1=0, M08 output is cancelled as CNC is reset;

When Bit1=1, M08 output is not cancelled as CNC is reset;

Note 3: There is no corresponding output signal for M09, and M08 output is cancelled if M09 is executed.

Note 4: The cooling can be controlled by the 

key on operation panel, see details in OPERATION.

- **Function description**

The lubricating defined by standard PLC program for this C1000T system has two types: Non-automatic and automatic lubricating, which are set by parameter:

DT17 =0: Non-automatic lubricating (same as version before)

DT17>0: Automatic lubricating, lubricating time DT17 and lubricating interval time DT16 available

Volume III Connection

1. Manual lubricating function

Press  on the machine operator panel and the system executes the lubricating output, pressing it again and the system cancels the output. Execute M32 and the system executes the output; execute M33 and the system stops it.

NO.112>1: the system executes the timed lubricating output, and the output is executed by pressing , and is cancelled when the output exceeds the one set by No.112. After the system executes M32, the lubricating output is executed within the time set by No.112 and it is cancelled. Execute M33 in the time set by No. 112 and the output is cancelled.

2. Automatic lubricating

When K16.2 is set to 1, the system executes the lubricating in the time set by DT17, and then stops the output. After it keeps the stop in the time set by DT16, it executes the lubricating again repetitively, and executes the cycle in turn. In automatic lubricating, M32, M33,  on the machine panel are valid, and the lubricating time is the one set by DT17.

Note 1: The lubricating output is disabled in emergency stop;

Note 2: K No.0.10 Bit1 sets whether the lubricating output is cancelled in reset:

Bit1 = 0: the lubricating output is disabled in reset;

Bit1 = 1: the lubricating output remains unchanged in reset.

2.8.10 Chuck control

- **Relevant signal (defined by standard PLC program)**

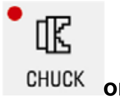
DIQP: Chuck control input signal

DOQPJ: Inner chuck clamping output/outer chuck releasing output signal

DOQPS: Inner chuck releasing output/outer chuck clamping output signal

NQPJ: Inner chuck clamping in-position/outer chuck releasing in-position signal, sharing a common interface with T08, M42I

WQPJ: Inner chuck releasing in-position/outer chuck clamping in-position signal, sharing a common interface with T07, M41I

Note 1: The key  **on the operation panel of C1000T can replace DIQP input signal, and the diagnosis address is X0026.6.**

Note 2: NQPJ, WQPJ with the 2nd spindle signals SALM2, VPO2 multiplex interface, so, the system does not check the chuck in-position signal when the multi-spindle function is valid.

- **Signal diagnosis**

Signal	DIQP	NQPJ	WQPJ	DOQPJ	DOQPS
Diagnosis address	X0.2	X3.4	X3.3	Y1.4	Y1.5
Interface pin	CN61.3	CN61.40	CN61.41	CN62.13	CN62.14

- **Control parameter**

K	1	2					**CCHU**	**NYQP**	**SLSP**	**SLQP**

SLQP =1: Chuck function active

=0: Chuck function inactive

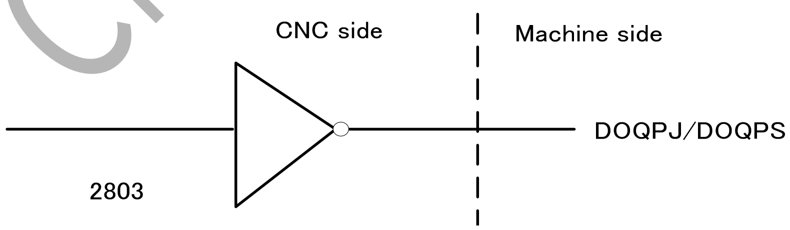


Fig. 2-52

Volume III Connection

- **Sequence**

① When SLQP=1, SLSP=0, NYQP=0, CCHU=1, CNC chooses inner chuck mode, and chuck in-position signal detecting is active:

DOQPS: chuck releasing output; WQPJ: releasing in-position signal;

DOQPJ: chuck clamping output; NQPJ: clamping in-position signal.

DOQPJ and DOQPS output high resistance at power on, when CNC detects that the chuck input signal DIQP is active for the 1st time, DOQPJ is connected to 0V and chuck is clamped.

After M12 is executed, DOQPS (pin 14 of CN62) outputs high resistance, DOQPJ(pin 13 of CN62) outputs 0V, chuck is clamped and CNC waits for NQPJ signal to be in-position.

After M13 is executed, DOQPJ (pin 13 of CN62) outputs high resistance, DOQPS(pin 14 of CN62) outputs 0V, chuck is released and CNC waits for WQPJ signal to be in-position.

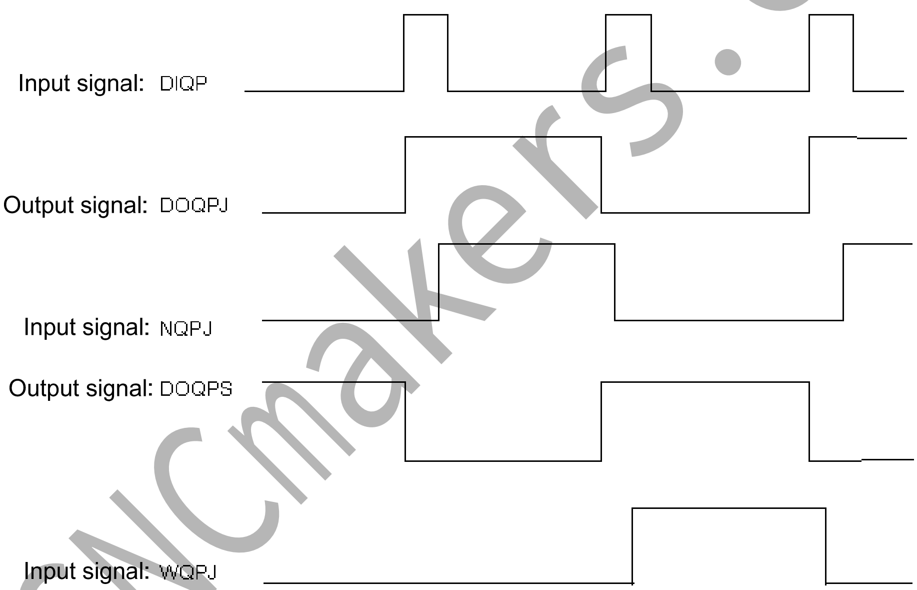


Fig. 2-53 (Chuck clamping, releasing signals are level output)

②When SLQP=1, SLSP=0, NYQP=1, CCHU=1, CNC chooses outer chuck mode, and chuck in-position signal detecting is active:

DOQPS: chuck clamping output; WQPJ: clamping in-position signal;

DOQPJ: chuck releasing output; NQPJ: releasing in-position signal.

DOQPJ and DOQPS output high resistance at power on, when CNC detects that the chuck input signal DIQP is active for the 1st time, DOQPS is connected to 0V and chuck is clamped.

After M12 is executed, DOQPS (CN62.14) outputs 0V, DOQPJ(CN62.13) outputs high resistance, chuck is clamped and CNC waits for WQPJ signal to be in-position.

After M13 is executed, DOQPJ (CN62.13) outputs 0V, DOQPS(CN62.14) outputs high resistance, chuck is released and CNC waits for NQPJ signal to be in-position.

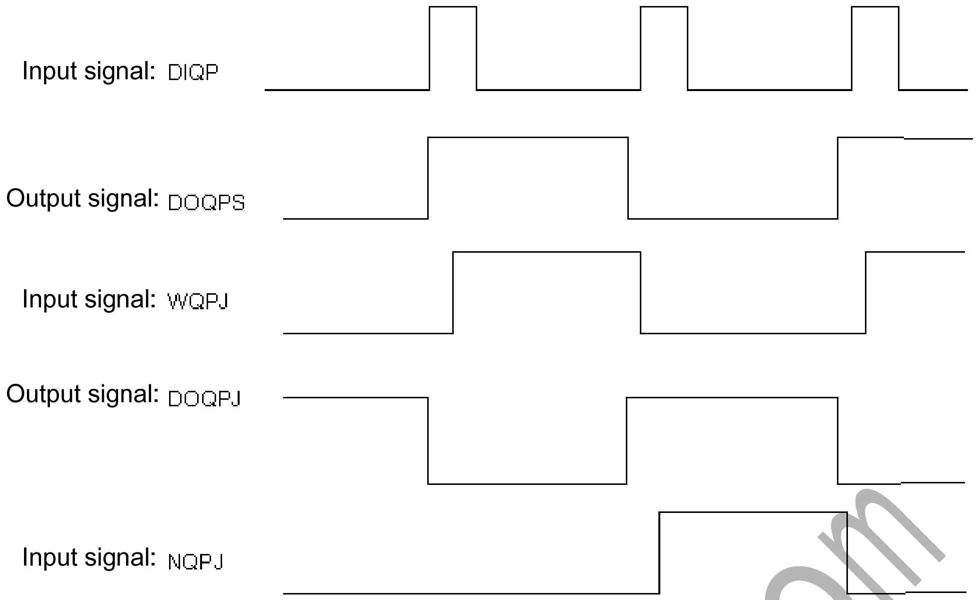


Fig. 2-54 Chuck clamping, releasing signals are level output

As the 2nd chuck input is active, DOQPS outputs 0V, chuck is released. The chuck clamping/releasing signal is output alternatively, i.e. the output is changed each chuck input signal is active.

③The interlock between the chuck and the spindle

When SLQP=1, SLSP=0, M3 or M4 is active, the alarm is issued if M13 is executed and the output is unchanged.

When SLQP=1, SLSP=0, CCHU=1, if M12 is executed in MDI or Auto mode, CNC does not execute next code till it detects the chuck clamping in-position signal is active. When the chuck input signal DIQP is active in Manual mode, the panel spindle CW, CCW key are inactive till it detects the chuck clamping in-position signal is active. In spindle running or auto cycle processing, DIQP input signal is inactive. And DOQPS, DOQPJ is held on at CNC reset and emergency stop.

2.8.15 Tailstock control

- **Relevant signal (defined by standard PLC program)**

DOTWJ: Tailstock forward output signal
DOTWS: Tailstock backward output signal
DITW: Tailstock input signal, DITW and BDT share a common interface.

Note: For C1000T-V, 

can replace DITW input signal, and the diagnosis address is X0026.5.

- **Signal diagnosis**

Signal	DITW	DOTWJ	DOTWS
Diagnosis address	X0.4	Y2.5	Y2.6
Interface pin	CN61.5	CN61.34	CN61.35

- **Control parameter**

State parameter

K	**1**	**3**							**SPTW**	**SLTW**

SLTW =1: Tailstock function active.

=0: Tailstock function inactive.

SPTW =1: No interlock between spindle rotation and tailstock advancing and retracting, tailstock may be moved regardless of the spindle, or spindle may run regardless of tailstock;

=0: Interlock between spindle rotation and tailstock advancing and retracting, tailstock retraction disabled as spindle is running, spindle disabled if tailstock does not advance.

- **Signal connection**

The tailstock circuit is shown in Fig. 2-55:

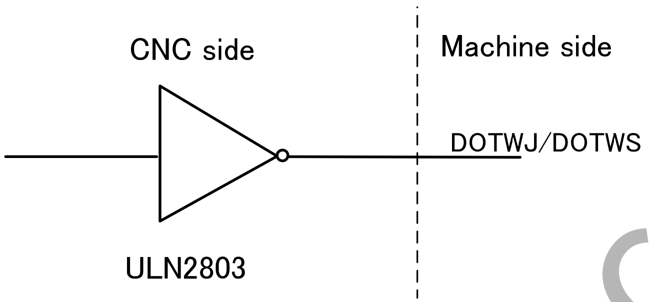


Fig. 2-55

- **Sequence (defined by standard PLC program)**

The sequence of tailstock is shown in Fig. 2-56:

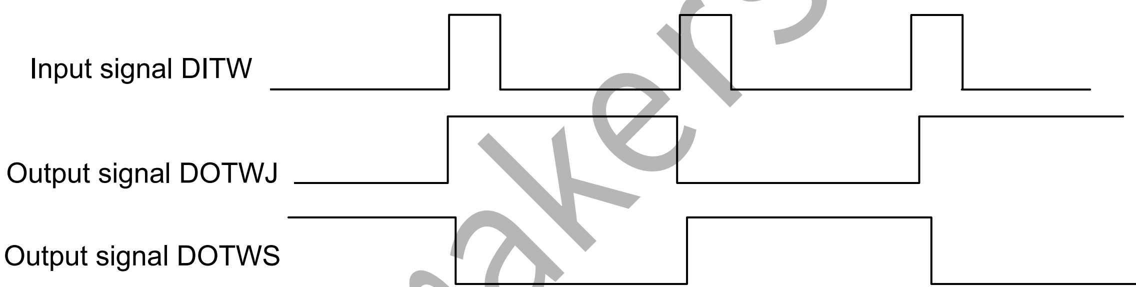


Fig. 2-56 Tailstock sequence

Tailstock advancing (DOTWJ) and retracting(DOTWS) are both inactive when power on; when the tailstock input (DITW) is active for the 1st time, tailstock advancing is active; when it is active for the 2nd time, tailstock retracting is active, so the DOTWJ/ DOTWS signal interlock is output alternatively, i.e. The output changes each time the DITW signal is active. If M10 is executed, DOTWJ (CN62.34) outputs 0V and tailstock advances; if M11 is executed, DOTWS (CN62.35) outputs 0V and tailstock retracts.

DITW signal is inactive as spindle is running. If M11 is executed, alarm will be issued, and its output are held on. And DOTWS, DOTWJ outputs are held on at CNC reset or emergency stop.

2.8.14 Safety door detection

- **Relevant signal**

  SAGT: For safety door detection, sharing a common interface with SP signal
- **Signal diagnosis**

Signal	SAGT
Diagnosis address	X0.0
Interface pin	CN61.1

- **Control parameter**

  State parameter

K	**1**	**4**					**SPB4**	**PB4**		

PB4 =0: Safety door detection inactive

=1: Safety door detection active, SP signal inactive

SPB4 =0: For safety door closing as SAGT is connected with 0V

=1: For safety door closing as SAGT is connected with +24V

- **Function description (defined by standard PLC program)**

①When PB4＝1, SPB4＝0, CNC confirms that the safety door is closed as SAGT is connected to 0V;

②When PB4＝1, SPB4＝1, CNC confirms that the safety door is closed as SAGT is connected to +24V;

③In Auto mode, if CNC detects the safety door is open, alarm is issued as cycle starts;

④In auto running, if CNC detects the safety door is open, the axis feed is held, and alarm is issued by CNC;

⑤The safety door detection function is only active in Auto mode;

2.8.19 Block skip

If a block in a program needs not to be executed and deleted, the block skip function may be selected. When the block is headed with “/” sign, and the block skip switch is turned on (machine panel key or external input of block skip is active), this block will be skipped without execution in auto running.

- **Relevant signal (defined by standard PLC program)**

  AEY/BDT: Block skip signal.
- **Signal diagnosis**

Signal	BDT
Diagnosis address	X2.7
Interface pin	CN61.36

- **Signal connection**

  The AEY/BDT signal circuit is shown in Fig.2-57:

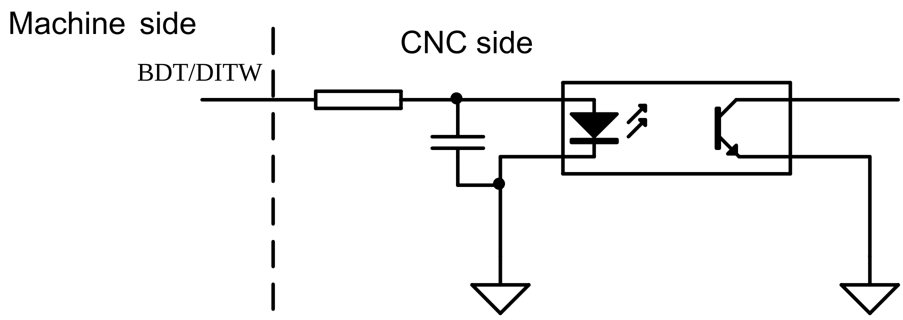


Fig. 2-57

- **Function description (defined by standard PLC program)**

When BDT signal is active, the block headed with "/" sign is skipped without being executed. The BDT input is equivalent to the function of the BLOCK SKIP key on the machine panel.

2.8.20 CNC macro variables

- **Relevant signal**

Macro output signal: standard PLC defines 5 macro output interfaces #1100～#1105;

Macro input signal: standard PLC defines 16 macro output interfaces #1000～#1015

- **Signal diagnosis**

Macro variable number	#1105	#1104	#1103	#1102	#1101	#1100
Diagnosis address	Y3.7	Y3.6	Y3.5	Y3.4	Y3.3	Y3.2

Macro variable number	#1007	#1006	#1005	#1004	#1003	#1002	#1001	#1000
Diagnosis address	X0.7	X0.6	X0.5	X0.4	X0.3	X0.2	X0.1	X0.0

Macro variable number	#1015	#1014	#1013	#1012	#1011	#1010	#1009	#1008
Diagnosis address	X1.7	X1.6	X1.5	X1.4	X1.3	X1.2	X1.1	X1.0

- **Function description (defined by standard PLC program)**

U00～U05 signal output may be changed if macro variable ＃1100～＃1105 are assigned. If they are assigned for "1", it outputs 0V, if they are assigned for "0", it turns off their output signals.

Detect the macro variable ＃1000～＃1015 values (input signal state), they may be used for various processing if combined with other transfer judgement macro code.

2.8.21 Tri-colour indicator

Relevant signals and function definitions:
Y2.2 (CN62.31): normal (non-running, non-alarming)
Y2.3 (CN62.32): running
Y2.4 (CN62.33): alarming

2.8.16 External MPG

● **Related signals**

CN31(MPG)	PLC address	Address character	Function	Remark
5	X6.0	EHDX	X MPG	Applied toPSG-100-05E/L, ZSSY2080 MPG
6	X6.1	EHDY	Y MPG	
8	X6.2	EHDZ	Z MPG	
9	X6.3	EMP0	Increment ×1	
22	X6.4	EMP1	Increment ×10	
23	X6.5	EMP2	Increment ×100	
11, 12, 13	GND			
14,15	+5V			
17,18	+24V			

● **Related parameters**

State parameter

0	0	1						MPG			

Bit3 =0：Step working mode.

=1：MPG working mode.

PLC state parameter

K	1	6		SINC							

SINC =0：MPG, STEP mode ×1000-gear increment is valid.

=1：MPG, STEP mode×1000-gear increment is invalid.

● **Function description**

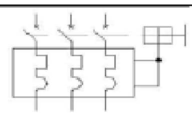
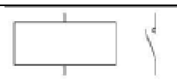
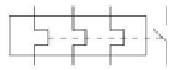
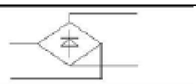
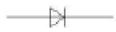
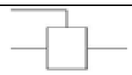
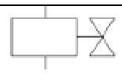
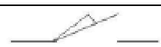
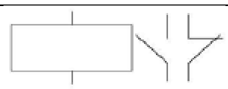
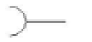
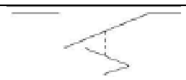
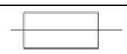
① When SINC is set to 1, MPG/STEP mode ×1000-gear selection is disabled. When x1000-gearis selected before modifying the parameter, the system automatically changes into ×100mm-gear

② When the external MPG, its axis selection does not lock, that is, the axis selection of MPG is disabled, the system changes to the non-axis selection state.

③ When the external MPG axis selection and gear selection input are enabled, the axis selection on the panel and the gear selection keys are disabled; when the external MPG axis selection and gear selection input are disabled, the axis selection on the panel and the gear selection keys are enabled and self-locked.

2.9 Commonly use symbol of electricity drawing

C1000T DC24V power supply and the electromagnetic valve with power working current separately use DC24V, and the electronic component explanations are as follows:

Name	Symbol	Graph	Name	Symbol	Graph
Air breaker	QF		Contactor coil contact and auxiliary contact	KM	
transformer	TC		Heat relay and contact	FR	
Rectifier	VC		Capacity	C	
Motor	M		Resistant	R	
Diode	VD		Hall switch		
Electromagnetic coil	YV		Travel switch	SQ	
Relay coil and contact	KA		Veneer socket		
			Pedal switch	SA	
			Fuse	FU	

CHAPTER 3 PARAMETERS

The CNC bit and data parameters are described in this chapter, various functions can be set by these parameters.

3.1 Parameter description (by sequence)

3.1.1 Bit parameter

The state parameter is expressed as follows:

Parameter No.	BIT7	BIT6	BIT5	BIT4	BIT3	BIT2	BIT1	BIT0

0 0 1	***	***	***	SPTY	SOHW	RDC	***	INI

Bit4 1: Spindle analog voltage control

0: Spindle switching volume control

Bit3 1: MPG mode

0: Step mode

Bit2 1: Programming by radius

0: Programming by diameter

Bit1 1: IS-C incremental system

0: IS-B incremental system

Bit0 1: Inch input

0: Metric input

Default:0 0 0 1 1 0 0 0

0 0 2	***	***	***	LIFJ	MTL	LIFC	ROFT	TLIF

Bit4 1: Tool life management group skip active

0: Tool life management group skip inactive

Bit3 1: Tool life management active in MDI mode

0: Tool life management inactive in MDI mode

Bit2 1: Tool life counting type 2, by times

0: Tool life counting type 1, by times

Bit1 1: Tool nose radius offset active

0: Tool nose radius offset inactive

Bit0 1: Tool life management active

0: Tool life management inactive

Default:0 0 0 0 0 0 1 0

0	0	3	***	***	SCRW	OFTM	***	***	CIMO	LM

Bit5 1: Pitch error offset active
0: Pitch error offset inactive
Bit4 1: Tool offset by coordinate offset
0: Tool offset by move
Bit0 1: Offset automatically change in metric and inch conversion
0: Offset not change in metric and inch conversion
Default:0 0 1 1 0 0 1 1

0	0	4	***	RDRN	DECI	ORC	***	***	PROD	SCW

Bit6 1: G00 is rapid traverse speed in dry run mode
0: G00 is manual feedrate in dry run mode
Bit5 1: DEC signal is high level for machine zero return
0: DEC signal is low level for machine zero return
Bit4 1: Tool offset by radius
0: Tool offset by diameter
Bit0 1: Inch system for min. code unit, active after repowering
0: Metric system for min. code unit, active after repowering
Default:0 1 0 0 0 0 0 0

0	0	5	***	***	SMAL	M30	MO2	***	***	***

Bit5 1: Spindle manual gearing for S code
0: Spindle automatic gearing for S code
Bit4 1: Cursor to beginning after M30 execution
0: Cursor not to beginning after M30 execution
Bit0 1: Axial output wave is pulse
0: Axial output wave is square
Default:0 0 0 1 0 0 0 0

0	0	6	***	***	ZMOD	ZM5	ZM4	ZMY	ZMZ	ZMX

Bit4 1：5th zero return mode C
0：5th zero return mode B
Bit3 1：4th zero return mode C
0：4th zero return mode B
Bit2 1: Y zero return type C
0: Y zero return type B
Bit1 1: Z zero return type C
0: Z zero return type B
Bit0 1: X zero return type C
0: X zero return type B

Default:0 0 0 0 0 0 0 0

0	0	7		DISP	***	SMZ	***	***	***	***	ZPLS

Bit7 1: Enter ABSOLUTE POS page after power on

0: Enter RELATIVE POS page after power on

Bit5 1：Execute the next block after all motion block exactly are executed to the in-position

0：Smooth transition between two blocks

Bit0 1: DECX and PCX signals are in parallel (a proximity switch taken as both DECX and zero signals) during machine zero return

0: DECX and PCX signals are separate (separate DECX and zero signals needed) during machine zero return

Default:1 0 0 0 0 0 0 1

0	0	8		***	***	***	DIR5	DIR4	DIRY	DIRZ	DIRX

Bit4 1：Direction signal (DIR) in 5th positive movement is HIGH

0：Direction signal (DIR) in 5th negative movement is HIGH

Bit3 1：Direction signal (DIR) in 4th positive movement is HIGH

0：Direction signal (DIR) in 4th negative movement is HIGH

Bit2 1: Direction signal (DIR) is high level as Y axis moves positively

0: Direction signal (DIR) is low level as Y axis moves negatively

Bit1 1: Direction signal (DIR) is high level as Z axis moves positively

0: Direction signal (DIR) is high level as Z axis moves negatively

Bit0 1: Direction signal (DIR) is high level as X axis moves positively

0: Direction signal (DIR) is high level as X axis moves negatively

Default:0 0 0 1 1 1 1 1

0	0	9		***	***	***	5ALM	4ALM	YALM	ZALM	XALM

Bit7 1: Spindle alarm signal (ZALM) is low level alarm

0: Spindle alarm signal (ZALM) is high level alarm

Bit4 1：5th alarm signal (ALM5) is LOW alarm

0：5th alarm signal (ALM5) is HIGH alarm

Bit3 1：4th alarm signal (ALM5) is LOW alarm

0：4th alarm signal (ALM5) is HIGH alarm

Bit2 1: Y alarm signal (YALM) is low level alarm

0: Y alarm signal (YALM) is high level alarm

Bit1 1: Z alarm signal (ZALM) is low level alarm

0: Z alarm signal (ZALM) is high level alarm

Bit0 1: X alarm signal (XALM) signal is low level alarm

0: X alarm signal (XALM) signal is high level alarm

Default:0 0 0 1 1 1 1 1

0	1	1		RVCS	***	***	***	NORF	ZNIK	***	***

Bit7 1: Backlash offset type B, the offset data are output by ascending or descending type and the set frequency is inactive;

0: Backlash offset type A, the offset data are output by the set frequency (by bit parameter No.010) or 1/8 of it.

Bit3 1: Manual machine zero return inactive

0: Manual machine zero return active

Bit2 1: Direction key locked during zero return, homing continues to end by pressing direction key once;

0: Direction key unlocked, which is held on during zero return

Default:0 0 0 0 0 0 0 0

0	1	2		***	WSFT	TCAR	***	***	***	***	ISOT

Bit6 1: Workpiece coordinate offset active, defined by offset No.000

0: Workpiece coordinate offset inactive

Bit5 1: Trial tool setting active

0: Trial tool setting inactive

Bit0 1: Prior to machine zero return after power on, manual rapid traverse active

0: Prior to machine zero return after power on, manual rapid traverse inactive

Default:0 0 0 0 0 0 0 0

0	1	3		HPF	RHPG	***	HW5	HW4	HWY	HWZ	HWX

Bit4 1：Coordinates increase in 5th MPG (CCW) rotation

0：Coordinates increase in 5th MPG (CW) rotation

Bit3 1：Coordinates increase in 4th MPG (CCW) rotation

0：Coordinates increase in 4th MPG (CW) rotation

Bit2 1：Coordinates increase in Y MPG (CCW) rotation

0：Coordinates increase in Y MPG (CW) rotation

Bit1 1：Coordinates increase in Z MPG (CCW) rotation

0：Coordinates increase in Z MPG (CW) rotation

Bit0 1：Coordinates increase in X MPG (CCW) rotation

0：Coordinates increase in X MPG (CW) rotation

Default:0 0 0 0 0 0 0 0

1	6	4		JAX	***	***	***	DLF	ZRN	AZR	SJZ

Bit0 1: Memory mechanical zero is memorized

0: Memory mechanical zero is memorized

Bit1 1: The reference point is not set up when the G28 alarm

0: The reference point is not set up when the G28 command uses the block

Bit2 1: The reference point is not set up when alarm the G28 is excepted

0: The reference point is not set up when not alarm the G28 is excepted

Bit3 1: After the reference point, the point is returned to for manual speed

0: After the reference point, the point is returned to for quick speed.

Bit7 1: Not choose multi axis when manual back to zero

0: Choose multi axis when manual back to zero

Default:0 0 0 0 0 1 0 0

1 6 8	WLOE	HLOE	GTAP	THRD	CBOL	CLSE	FBOL	FLSE

Bit0 0: Fast running as a straight line type

1: Fast running for the pre acceleration / deceleration S / post acceleration / deceleration index

Bit1 0: Acceleration and deceleration before the fast running mode

1: Acceleration and deceleration after the fast running mode

Bit2 0: Cutting feed for line

1: The cutting feed is the first and the S type and the acce and deceindex type.

Bit3 0: Acceleration and deceleration before cutting feed mode

1: Acceleration and deceleration after cutting feed mode

Bit4 0: Thread processing and deceleration mode for the front and deceleration line

1: Thread processing and deceleration mode for the first S type

Default:1 0 0 0 1 1 0 1

1 7 5	SPFD	SAR	THDA	VAL5	VAL4	VALY	VALZ	VALX

Bit7 1：In cutting feed, do not permit the spindle stopping rotation; the spindle stops, the feed also stops when the system appears No.404 alarm cutting

0：In cutting feed, permit the spindle stops rotation; the spindle stops rotation, but the feed does not stop

Bit6 1: Detect spindle SAR signal prior to cutting

0: Not detect spindle SAR signal prior to cutting

Bit5 1：Thread machining is the exponential acceleration/deceleration

0：Thread machining is the linear acceleration/deceleration

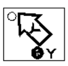
Bit4 1：5th movement key is _positive, is _negative

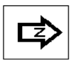
0：5th movement key is _positive, is _negative

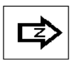
Bit3 1：4th movement key  is positive,  is negative

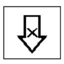
0：4th movement key  is positive， is negative

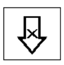
Bit2 1：Y movement key  is positive， is negative

0：Y movement key  is positive， is negative

Bit1 1：movement key  is positive， is negative

0：Z movement key  is positive， is negative

Bit0 1：X movement key  is positive， is negative

0：X movement key  is positive， is negative

Default:0 0 0 0 0 0 0 0

1	8	0		NAT	***	***	***	***	***	***	SPOS

Bit7 1: Function ATAN, ASIN range is 90.0～270.0;

0: Function ATAN, ASIN range is -90.0～90.0

Bit0 1: DIS TO GO displayed in POS&PRG page

0: RELATIVE POS displayed in POS&PRG page

Default:0 0 0 0 0 0 1 0

1	8	3		***	***	***	MZR5	MZR4	MZRY	MZRZ	MARX

Bit4 1：5th press_key to execute the machine zero return

0：5th press_key to execute the machine zero return

Bit3 1：4th press  to execute the machine zero return

0：4th press  to execute the machine zero return

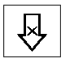
Bit2 1：Y press  to execute the machine zero return

0：Y press  to execute the machine zero return

Bit1 1：Z press  to execute the machine zero return

0：Z press  to execute the machine zero return

Bit0 1：X press  to execute the machine zero return

0：X press  to execute the machine zero return

Default:0 0 0 0 0 0 1 0

1	8	4		***	***	NE9	NE8	***	***	LS1	LS0

Bit6 1: Interface auto test active (CNC repower needed)

0: Interface auto test inactive

Bit0, Bit1, Bit2: Interface language selection

Default:0 0 1 1 0 0 0 0

1	8	5		***	***	***	***	***	***	PRPD	PLA

Bit1 1: Axis rapid traverse rate of PLC by input value

0: Axis rapid traverse rate of PLC by parameter value (X axis: No.022；Z axis:No.023；Y axis:No.134)

Bit0 1: PLC axis control active, active after repowering

0: PLC axis control inactive, active after repowering

Default:0 0 0 0 0 0 0 0

Volume III Connection

1	8	6		RTORI	SRS	***	***	***	TCRG	***	***

Bit7 1：In executing M29, the spindle executes the machine zero return

0：In executing M29, the spindle does not execute the machine zero return

Bit6 1：The spindle selection signal is RGTSPn in the multiple spindle rigid tapping

0：The spindle selection signal is SWSn in the multiple spindle rigid tapping

Bit2 1：In rigid tapping cancel, do not wait for G61.0 to be 0 in executing the next block

0：In rigid tapping cancel, wait for G61.0 to be 0 in executing the next block

Default:0 0 0 0 0 0 0 0

1	8	7		***	***	RCSY	***	***	***	ROSY	ROTY

Bit5 1: Y Cs function is valid

0：Y Cs function is invalid

Bit1 1：sets Y to be rotary axis(A type)，

0：sets Y to be rotary axis(B type)， 10： sets Y to be invalid

Bit0 0 ：sets Y to be linear

1 ： sets Y to be rotary

Default:0 0 0 0 0 0 1 0

1	8	8		***	***	***	***	***	RRLY	RABY	ROAY

Bit2 1：When Y is the rotary axis, the relative coordinate cycle function is valid

0：When Y is the rotary axis, the relative coordinate cycle function is invalid

Bit1 1：Y rotates according to the symbol when it is the rotary axis

0：Y rotates contiguously when it is the rotary axis

Bit0 1：The absolute coordinate cycle function is valid when Y is the rotary axis

0：The absolute coordinate cycle function is valid when Y is the rotary axis

Default:0 0 0 0 0 1 0 1

1	8	9		***	***	RCS4	***	***	***	ROS4	ROT4

Bit5 0 : 4th Cs function is valid

1: 4th Cs function is invalid

Bit1 0： sets 4th to be the rotary axis(B type)

1： sets 4th to be the rotary axis(A type)

Bit0 0： sets 4th to be the linear

1： sets 4th to be the rotary axis

Default:0 0 0 0 0 0 1 0

1	9	0

***	***	***	***	***	RRL4	RAB4	ROA4

Bit2 1：When 4th is the rotary axis, the relative coordinate cycle function is valid

0：When 4th is the rotary axis, the relative coordinate cycle function is invalid

Bit1 1：4th rotates according to the symbol when it is the rotary axis

0：4th rotates contiguously when it is the rotary axis

Bit0 1：The absolute coordinate cycle function is valid when 4th is the rotary axis

0：The absolute coordinate cycle function is valid when 4th is the rotary axis

Default:0 0 0 0 0 1 0 1

1	9	1

***	***	RCS5	***	***	***	ROS5	ROT5

Bit5 0：5th Cs function is valid；

1：5th Cs function is invalid

Bit1 0： sets 5th to be the rotary axis(A type)，

1： sets 5th to be the rotary axis(B type)，

Bit0 0: sets 5th to be the linear

1: sets 5th to be the rotary

Default:0 0 0 0 0 0 1 0

1	9	2

***	***	***	***	***	RRL5	RAB5	ROA5

Bit2 1：When 5th is the rotary axis, the relative coordinate cycle function is valid

0：When 5th is the rotary axis, the relative coordinate cycle function is invalid

Bit1 1：5th rotates according to the symbol when it is the rotary axis

0：5th rotates contiguously when it is the rotary axis

Bit0 1：The absolute coordinate cycle function is valid when 5th is the rotary axis

0：The absolute coordinate cycle function is invalid when 5th is the rotary axis

Default:0 0 0 0 0 0 1 0

2	0	2

***	***	***	JEN5	JEN4	JENY	JENZ	JENX

Bit4 1：Do not stop 5th motion when the skip signal is valid

0：Stop 5th motion when the skip signal is valid

Bit3 1：Do not stop 4th motion when the skip signal is valid

0：Stop 4th motion when the skip signal is valid

Bit2 1：Do not stop Y motion when the skip signal is valid

0：Stop Y motion when the skip signal is valid

Bit1 1：Do not stop Z motion when the skip signal is valid

0：Stop Z motion when the skip signal is valid

Bit0 1：Do not stop X motion when the skip signal is valid

0：Stop X motion when the skip signal is valid

Default:0 0 0 0 0 0 0 0

2	0	3		***	***	***	ABP5	ABP4	ABPY	ABPZ	ABPX

Bit4 1：5th pulse outputs based on the two-phase quadrature

0：5th pulse outputs based on (pulse+direction)

Bit3 1：4th pulse outputs based on the two-phase quadrature

0：4th pulse outputs based on (pulse+direction)

Bit2 1：Y pulse outputs based on the two-phase quadrature

0：Y pulse outputs based on (pulse+direction)

Bit1 1：Z pulse outputs based on the two-phase quadrature

0：Z pulse outputs based on (pulse+direction)

Bit0 1：X pulse outputs based on the two-phase quadrature

0：X pulse outputs based on (pulse+direction)

Default:0 0 0 0 0 0 0 0

2	0	5		YTP	***	ABP5	***	MCL	MKP	***	SEQ

Bit7 1：The third axis is the linkage axis
0:The third axis is not the linkage axis

Bit5 1：Show startup interface

0: Not show startup interface

Bit3 1: Delete program when reset under the state interface

0: Not delete program when reset under the state interface

Bit2 1：Delete program after executive program at the interface

0：Not delete program after executive program at the interface

Bit0 1：Insert number automatically

0：Not insert number automatically

Default:0 0 0 0 0 0 0 0

2	0	6		ITL	***	***	***	***	***	***	SCBM

Bit7 1： All axis interlocking signals are valid

0： All axis interlocking signals are invalid

Bit0 1: Travel detection before moving

0: Not travel detection before moving

Default:0 0 0 0 0 0 1 0

2 0 6	ITL	***	***	***	***	***	***	SCBM

Bit7 1： All axis interlocking signals are valid
0： All axis interlocking signals are invalid
Bit0 1: Travel detection before moving
0: Not travel detection before moving

Default:0 0 0 0 0 0 1 0

2 1 0	CALT	ALS	***	***	***	***	***	***

Bit6 1： Automatic corner rate function is valid
0： Automatic corner rate function is invalid
Bit7 1: Acceleration control when exponential type plus deceleration cutting feed
0: Acceleration freedom when exponential type plus deceleration cutting feed

Default:0 0 0 0 0 0 0 0

2 1 1	***	***	TDR	***	***	***	***	***

Bit5 1: Dry running is valid during the operation of tapping
0: Dry running is invalid during the operation of tapping

Default:0 0 0 0 0 0 0 0

2 1 2	DWL	***	SOC	RSC	***	***	***	***

Bit7 1: G04 is pause in every turn of the feed mode
0: G04 is not pause in every turn of the feed mode

Bit5 1: After the spindle speed control spindle override
0: Before the spindle speed control spindle override

Bit4 1:G90 spindle speed when G0 positioning according to the current coordinate
0:G90 spindle speed when G0 positioning according to the Final coordinate

Default:0 0 0 0 0 0 0 0

2 1 3	OVU	DOV	TDR	***	ORI	***	PCP	SSOG

Bit7 1: Rigid tapping knife back rate is 10%
0: Rigid tapping knife back rate is 1%

Bit6 1: Rigid tapping knife back rate is valid
0: Rigid tapping knife back rate is invalid

Bit5 1: Rigid tapping knife, knife back use the same time constant
0: Rigid tapping knife, knife back dose not use the same time constant

Bit3 1: The spindle stop when flexible tapping at the beginning
0: The spindle does not stop when flexible tapping at the beginning

Bit1 1: Tapping into high speed deep hole tapping cycle
0: Not into high speed deep hole tapping cycle

Bit0 1: Tapping mode of spindle control for servo
0: Tapping mode of spindle control as follow

Default:0 0 0 0 0 0 0 0

2 1 4	LEDT	LOPT	OHPG	***	***	SOVD	FOVD	ROVD

Bit7 1: Use external editor lock
0: Not use external editor lock

Bit6 1: Use external operation panel lock
0: Not use external operation panel lock

Bit5 1: Use the external hand wheel
0: Not use the external hand wheel

Bit2 1: Use band switch on the main shaft speed adjustment
0: Use the operating panel on the speed adjustment of the main shaft

Bit1 1: Use band switch on the cutting feed rate adjustment
0:Use the operating panel on the cutting feed rate adjustment

Bit0 1: Use band switch on the fast running rate adjustment
0: Use the operating panel on the fast running rate adjustment

Default:0 0 0 0 0 0 0 0

2 1 5	***	***	***	***	LALM	EALM	SALM	FALM

Bit3 1：Lgnore hard limit alarm
0: Not lgnore hard limit alarm

Bit2 1：Lgnore emergency stop alarm
0: Not lgnore emergency stop alarm

Bit1 1: Lgnore alarm of the spindle drive
0: Not lgnore alarm of the spindle drive

Bit0 1：Lgnore alarm of the feed shaft drive
0：Not lgnore alarm of the feed shaft drive

Default:0 0 0 0 0 0 0 0

Volume Ⅲ Connection

3.1.2 Data parameter

0 1 5	CMRX(X axis)multiplier coefficient
0 1 6	CMRZ(Z axis) multiplier coefficient

［Data range］ 1 ～65536
Default value 1

0 1 7	X axes frequency division coefficient
0 1 8	Z axes frequency division coefficient

［Data range］ 1 ～65536

Electronic gear ratio formula:

$$\frac{CMR}{CMD} = \frac{P}{\mathrm{L} \times 1000}$$

P: Feedback corresponding to the number of pulses when motor rotation

L: Movement of machine tools when motor rotation
Default value 1

0	1	9

Run-out length in threading

［Data range］ 0～225

Thread run-out width= THDCH×0.1×screw lead

Default value 5

0	2	1

Voltage offset value when spindle max. speed analog voltage output is 10V

［Data unit］ V

［Data range］ -0.2～0.2(Unit: mV)

Default value 0

0	2	2

X axis max. rapid traverse speed (radius)

［Data unit］

Setting unit	Data unit
Metric machine	mm/min
Inch machine	0.1inch/min

［Data range］ 0～90000

Default value 5000

0	2	3

Z max. rapid traverse speed

［Data unit］

Setting unit	Data unit
Metric machine	mm/min
Inch machine	0.1inch/min

［Data range］ 0～90000

Default value 5000

0	2	4
0	2	5

ACC&DEC time constant of LINTX(X axis) rapid traverse

ACC&DEC time constant of LINTX(Z axis) rapid traverse

［Data unit］ ms

［Data range］ 1～4000

Default value 100

0	3	1

the set speed when the manual feedrate override is 100%

［Data unit］

Setting unit	Data unit
Metric machine	mm/min
Inch machine	0.1inch/min

［Data range］ 0～8000

Default value 1260

0	3	2	**Rapid traverse rate as axis rapid override is F0**

［Data unit］

Setting unit	Data unit
Metric machine	mm/min
Inch machine	0.1inch/min

［Data range］ 6～4000

Default value 1260

0	3	3	**Low rate as axes return to machine zero**

［Data unit］

Setting unit	Data unit
Metric machine	mm/min
Inch machine	0.1inch/min

［Data range］ 6～4000

Default value 40

0	3	4	**X backlash compensation.**
0	3	5	**Z backlash compensation**

［Data unit］

Setting unit	Data unit
Metric machine	mm
Inch machine	inch

［Data range］ 0~0.5000

Default value 0

Note ： X is the diameter value.

0	3	7	**Max. speeds of gear1**
0	3	8	**Max. speeds of gear2**
0	3	9	**Max. speeds of gear3**
0	4	0	**Max. speeds of gear4**

［Data unit］ r/min

［Data range］ 10～9999

Default value 6000

0	4	1	**Exponential ac-deceleration start speed and deceleration final speed in manual feed**

［Data unit］

Setting unit	Data unit
Metric machine	mm/min
Inch machine	inch/min

［Data range］ 0～8000

Default value 40

0	4	2	**Block No. increment for auto block No. insertion**

［Data range］ 1 ～100

Default value 10

0	4	3	**(G96) Spindle min. speed under the constant surface speed control**

［Data unit］ r/min

［Data range］ 0～9999

Default value 100

0	4	4	**Serial communication baud rate**

［Data unit］ bit/s

［Data range］ 1200,2400,4800,9600,19200,38400 57600 115200

Default value 115200

0	4	5	**X positive max. travel**
0	4	6	**Z positive max. travel**

［Data unit］

Setting unit	Data unit
Metric machine	mm
Inch machine	inch

［Data range］ -99999999～99999999

Default value 9999. 9999

Note: If the BIT2 of the parameter No.001 is set for diameter, the X axis value is specified by diameter; if for radius, the X axis value is specified by radius.

0	4	7	X negative max. travel
0	4	8	Z negative max. travel

［Data unit］

Setting unit	Data unit
Metric machine	mm
Inch machine	inch

［Data range］ -99999999～99999999

Default value −9999. 9999

Note: If the BIT2 of the parameter No.001 is set for diameter, the X axis value is specified by diameter; if for radius, the X axis value is specified by radius.

0	5	1	Each feeding for G71, G72 rough turning cycle

［Data unit］

Setting unit	Data unit
Metric machine	mm
Inch machine	inch

［Data range］ 0.0010～99.9999

Default value 0

0	5	2	Each retraction for G71, G72 rough turning cycle

［Data unit］

Setting unit	Data unit
Metric machine	mm
Inch machine	inch

［Data range］ 0～99.9999

Default value 0

0	5	3

X axis rough turning retraction in G73

［Data unit］

Setting unit	Data unit
Metric machine	mm
Inch machine	inch

［Data range］ -9999.9999～9999.9999

Default value 0

0	5	4

Z axis rough turning retraction in G73

［Data unit］

Setting unit	Data unit
Metric machine	mm
Inch machine	inch

［Data range］ -9999.9999～9999.9999

Default value 0

0	5	5

Cycle times of G73

［Data unit］ times

［Data range］ 1 ～9999

Default value 0

0	5	6

Z axis retraction of G74 or G75

［Data unit］

Setting unit	Data unit
Metric machine	mm
Inch machine	inch

［Data range］ 0～99.9999

Default value 0

0	5	7	Repetitions of G76 finish machining

［Data unit］ times
［Data range］ 1 ～99
Default value 1

0	5	8	Tool nose angle of G76 cycle

［Data unit］ deg
［Data range］ 0 ～99
Default value 0

0	5	9	Min. cut depth of G76 cycle

［Data unit］

Setting unit	Data unit
Metric machine	mm
Inch machine	inch

［Data range］ 0～99.9999

Default value 0

0	6	0	Finish allowance of G76

［Data unit］

Setting unit	Data unit
Metric machine	mm
Inch machine	inch

［Data range］ 0～99.9999

Default value 0

0	6	5	spindle gear shifting time 1
0	6	6	spindle gear shifting time 2

［Data unit］ ms
［Data range］ 0～60000
Default value 1000

0	6	7	Output voltage of spindle gearing

［Data unit］ mV
［Data range］ 0～10000
Default value 100

0	6	8

spindle speed in manual (MPG) mode

［Data unit］ r/min

［Data range］ 0～3000

Default value 0

0	6	9

Time span of low pressure alarm

［Data unit］ ms

［Data range］ 0～60000

Default value 0

0	7	0

Spindle encoder pulses

［Data unit］ line/revolution

［Data range］ 100～5000

Default value 1024

0	7	1

Output time of reset signal

［Data unit］ ms

［Data range］ 50～400

Default value 200

0	7	2

Delay detecting time of SAR signal

［Data unit］ ms

［Data range］ 0～4080

Default value 0

0	7	3

Max. spindle motor clamping speed

［Data unit］ r/min

［Data range］ 0 ～4095

Setting value =（max. clamp speed of spindle/max. speed of spindle motor）×4095.

Default value 4095

0	7	4

Min. spindle motor clamping speed

［Data unit］ r/min

［Data range］ 0 ～4095

Setting value =（min. clamping speed of spindle motor/max. speed of spindle motor）×4095

Default value 0

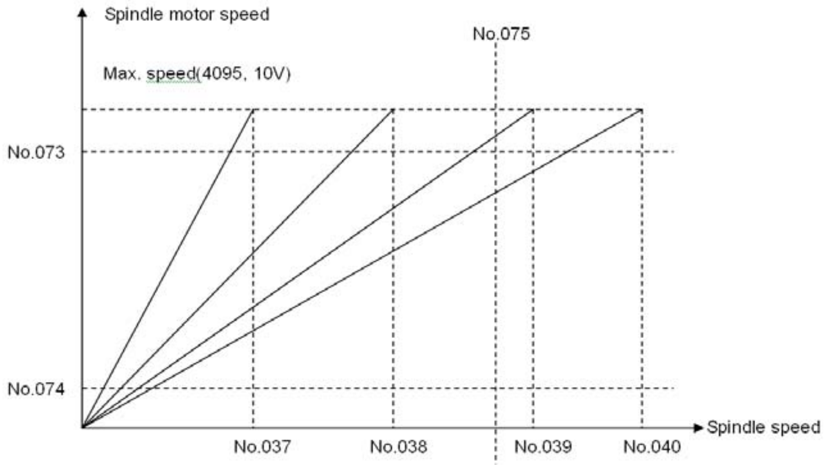
0	7	5

Maximum spindle rev speed

［Data unit］ r/min

［Data range］ 0 ～9999

Default value 6000



0 7 6 **Upper time limit of shifting one tool**

［Data unit］

Setting unit	Data unit
Metric machine	mm
Inch machine	inch

［Data range］ 0～9999

Default value 300

0 7 8 **Upper time limit of shifting maximum tools**

［Data unit］ ms

［Data range］ 100～60000

Default value 15000

0 8 0 **M code execution time**

［Data unit］ ms

［Data range］ 100～5000

Default value 500

0 8 1 **S code execution time**

［Data unit］ ms

［Data range］ 100～5000

Default value 500

0	8	2

Delay time from tool post CCW stop to tool post CW clamping start

［Data unit］ ms

［Data range］ 0 ～4000

Default value 0

0	8	3

Alarm time of not receiving *TCP signal

［Data unit］ ms

［Data range］ 0～4000

Default value 500

0	8	4

Total tools number selection

［Data unit］ piece

［Data range］ 1～32

Default value 4

Note： Set to 1 when using the row rest

0	8	5

Tool post CW clamping time

［Data unit］ ms

［Data range］ 0～4000

Default value 1000

0	8	7

Delay time from M05 output to SPZD output

［Data unit］ ms

［Data range］ 0～10000

Default value 0

0	8	9

SPZD output time

［Data unit］ ms

［Data range］ 0～60000

Default value 50

0	9	0

M code to allow the number of digits

［Data range］ 1~2

Default value 2

0	9	1

S code to allow the number of digits

［Data range］ 1~6

Default value 5

0	9	2

T code to allow the number of digits

［Data range］ 1~6

Default value 4

0	9	3

The beginning of the amount of the MDI input tool is forbidden

［Data range］ 0~9999

Default value 0

0	9	4

The number of tool offset from the MDI input is prohibited

［Data range］ 0~9999

Default value 0

0	9	5

Time to pause in a single direction

［Data unit］ s

［Data range］ 0 ～10

Default value 0

0	9	6
0	9	7

X Ptch offset No. of PECOPRGX

Z Pitch offset No. of PECOPRGX

［Data unit］

Setting unit	Data unit
Metric machine	mm
Inch machine	inch

［Data range］ -99.9999～99. 9999

Default value 0

0	9	8
0	9	9

X Pitch offset No. of PECOPRGX machine zero

Z Pitch offset No. of PECOPRGZ machine zero

［Data range］ 0 ～255

Default value 0

1	0	2
1	0	3

X Pitch offset interval of PECINTX

Z Pitch offset interval of PECINTZ

［Data unit］

Setting unit	Data unit
Metric machine	mm
Inch machine	inch

［Data range］ 1～99. 9999

Default value 0

1	0	6

Spindle fluctuation alarm limit in threading (not detect spindle fluctuation alarm

［Data unit］ %

［Data range］ 0 ～100

Default value 0

1 0 7 **Short axis speed in threading run-out (run-out by threading feedrate if set to 0)**

［Data unit］

Setting unit	Data unit
Metric machine	mm/min
Inch machine	inch/min

［Data range］ 0～8000

Default value 0

1 0 8 **Spindle jog time**

［Data unit］ ms

［Data range］ 0～60000

Default value 0

1 0 9 **Spindle jog speed (r/min)**

［Data unit］ r/min

Default value 40

1 1 0 **Gear ratio of encoder to spindle: spindle gear teeth number**

［Data range］ 1 ～255

Default value 1

1 1 1 **Gear ratio of encoder to spindle: encoder gear teeth number**

［Data range］ 1 ～255

Default value 1

1 1 2 **Lubricating time (no limit if set to 0)**

［Data unit］ ms

［Data range］ 0～60000

Default value 1

1 1 3 **Axes rapid traverse rates in machine zero return**

［Data unit］

Setting unit	Data unit
Metric machine	mm/min
Inch machine	inch/min

［Data range］ 10～9999

Default value 4000

1 1 4 **Offset of X machine zero**

1 1 5 **Offset of Z machine zero**

［Data unit］

Setting unit	Data unit
Metric machine	mm/min
Inch machine	inch/min

［Data range］ 0～100

Default value 0

1	1	9	Z negative max. travel

［Data range］ -9999.9999~9999.9999

Default value -9999

1	2	0	X 1st reference point machine coordinates
1	2	1	Z 1st reference point machine coordinates
1	2	2	X 2nd reference point machine coordinates
1	2	3	Z 2nd reference point machine coordinates
1	2	4	X 3rd reference point machine coordinates
1	2	5	Z 3rd reference point machine coordinates
1	2	6	X 4th reference point machine coordinates
1	2	7	Z 4th reference point machine coordinates

［Data unit］

Setting unit	Data unit
Metric machine	mm
Inch machine	inch

［Data range］ -9999.9999~9999.9999

Default value 0

1	2	8	G54_X Offset of the X axis coordinate system 1
1	2	9	G54_Z Offset of the X axis coordinate system 1

［Data unit］

Setting unit	Data unit
Metric machine	mm
Inch machine	inch

［Data range］ -999.999~999.999

Default value 0

1	3	0	G55_X Offset of the X axis coordinate system 2
1	3	1	G55_Z Offset of the X axis coordinate system 2

［Data unit］

Setting unit	Data unit
Metric machine	mm
Inch machine	inch

［Data range］ -999.999~999.999

Default value 0

1	3	2
1	3	3

G56_X Offset of the X axis coordinate system 3
G56_Z Offset of the X axis coordinate system 3

［Data unit］

Setting unit	Data unit
Metric machine	mm
Inch machine	inch

［Data range］ -999.999~999.999

Default value 0

1	3	4
1	3	5

G57_X Offset of the X axis coordinate system 4
G57_Z Offset of the X axis coordinate system 4

［Data unit］

Setting unit	Data unit
Metric machine	mm
Inch machine	inch

［Data range］ -999.999~999.999

Default value 0

1	3	6
1	3	7

G58_X Offset of the X axis coordinate system 5
G58_Z Offset of the X axis coordinate system 5

［Data unit］

Setting unit	Data unit
Metric machine	mm
Inch machine	inch

［Data range］ -999.999~999.999

Default value 0

1	3	8
1	3	9

G59_X Offset of the X axis coordinate system 6
G59_Z Offset of the X axis coordinate system 6

［Data unit］

Setting unit	Data unit
Metric machine	mm
Inch machine	inch

［Data range］ -999.999~999.999

Default value 0

Volume III Connection

1	4	0

+ or – limit of each input wear in TOOL OFFSET&WEAR interface

［Data unit］

Setting unit	Data unit
Metric machine	mm
Inch machine	inch

［Data range］ 0.001~99.9999

Default value 1

1	4	6	**Y pulse output multiplication coefficien**
1	4	7	**4TH pulse output multiplication coefficien**
1	4	8	**5TH pulse output multiplication coefficien**

［Data range］ 1 ～65536

Default value 1

1	4	9	**Y pulse output division coefficient**
1	5	0	**4TH pulse output division coefficient**
1	5	1	**5TH pulse output division coefficient**

［Data range］ 1 ～65536

Default value 1

1	5	3

current being used ladder numbe

［Data range］ 0 ～15

Default value 1

1	5	4

Max. arc radius error

［Data unit］

Setting unit	Data unit
Metric machine	mm
Inch machine	inch

［Data range］ 0.0001~1

Default value 0.01

1	5	5	**Y max. rapid traverse speed**
1	5	6	**4TH max. rapid traverse speed**
1	5	7	**5TH max. rapid traverse speed**

［Data unit］

Setting unit	Data unit
Metric machine	mm/min
Inch machine	inch/min
Rotary axis	deg/min

［Data range］ 0～90000

Default value 5000

Volume Ⅲ Connection

1	5	8	**Y acceleration/deceleration time constant value in rapid traverse**
1	5	9	**4TH acceleration/deceleration time constant value in rapid traverse**
1	6	0	**5TH acceleration/deceleration time constant value in rapid traverse**

［Data unit］ ms

［Data range］ 1 ～4000

Default value 100

1	6	2	**initial speed of CS acceleration/deceleration**

［Data unit］ deg/min

［Data range］ 0～4000

Default value 10

1	6	3	**acceleration/deceleration time constant of CS**

［Data unit］ ms

［Data range］ 0～4000

Default value 100

1	6	6	**initial speed of linear acceleration/deceleration in rigid tapping**

［Data unit］

Setting unit	Data unit
Metric machine	mm/min
Inch machine	inch/min

［Data range］ 0～4000

Default value 10

1	6	7	**linear acceleration/deceleration time constant in rigid tapping tool in-feed**

［Data unit］ ms

［Data range］ 0～4000

Default value 200

1 6 9	override value in rigid tapping tool retraction (override is 100% when it is set to 0)

［Data range］ 0.8~1.2

Default value 1

1 7 1	max. spindle speed in rigid tapping

［Data unit］ r/min

［Data range］ 0～6000

Default value 800

1 7 4	Low speed of Y machine zero return
1 7 5	Low speed of 4TH machine zero return
1 7 6	Low speed of 5TH machine zero return

［Data unit］

Setting unit	Data unit
Metric machine	mm/min
Inch machine	inch/min
Rotary axis	deg/min

［Data range］ 6～4000

Default value 40

1 7 7	High speed of Y machine zero return
1 7 8	High speed of 4TH machine zero return
1 7 9	High speed of 5TH machine zero return

［Data unit］

Setting unit	Data unit
Metric machine	mm/min
Inch machine	inch/min
Rotary axis	deg/min

［Data range］ 10～9999

Default value 4000

1	8	0
1	8	1
1	8	2

Y backlash compensation value
4TH backlash compensation value
5TH backlash compensation value

［Data unit］

Setting unit	Data unit
Metric machine	mm/min
Inch machine	inch/min
Rotary axis	deg/min

［Data range］ 0～0.5

Default value 0

1	8	3
1	8	4
1	8	5

Y pitch error compensation inverval
4TH pitch error compensation inverval
5TH pitch error compensation inverval

［Data unit］

Setting unit	Data unit
Metric machine	mm/min
Inch machine	inch/min
Rotary axis	deg/min

［Data range］ 1～9999.9999

Default value 10

1	8	6
1	8	7
1	8	8

Corresponding pitch error compensation position number of Y
Corresponding pitch error compensation position number of 4TH
Corresponding pitch error compensation position number of 5TH

［Data range］ 0 ～255

Default value 0

1	8	9
1	9	0
1	9	1

Offset value of Y machine zero
Offset value of 4TH machine zero
Offset value of 5TH machine zero

［Data unit］

Setting unit	Data unit
Metric machine	mm/min
Inch machine	inch/min
Rotary axis	deg/min

［Data range］ 0～100

Default value 0

Volume Ⅲ Connection

1	9	2	Positive max. travel of Y
1	9	3	Positive max. travel of 4th
1	9	4	Positive max. travel of 5th

［Data unit］

Setting unit	Data unit
Metric machine	mm/min
Inch machine	inch/min
Rotary axis	deg/min

［Data range］ -9999.9999~9999.9999

Default value 9999.9999

1	9	2	Negative max. travel of Y
1	9	3	Negative max. travel of 4th
1	9	4	Negative max. travel of 5th

［Data unit］

Setting unit	Data unit
Metric machine	mm/min
Inch machine	inch/min
Rotary axis	deg/min

［Data range］ -9999.9999~9999.9999

Default value -9999.9999

2	0	1	**Y 1st reference point machine coordinates**
2	0	2	**4TH 1st reference point machine coordinates**
2	0	3	**5TH 1st reference point machine coordinates**
2	0	4	**Y 2nd reference point machine coordinates**
2	0	5	**4TH 2nd reference point machine coordinates**
2	0	6	**5TH 2nd reference point machine coordinates**
2	0	7	**Y 3rd reference point machine coordinates**
2	0	8	**4TH 3rd reference point machine coordinates**
2	0	9	**5TH 3rd reference point machine coordinates**
2	1	0	**Y 4th reference point machine coordinates**
2	1	1	**4TH 4th reference point machine coordinates**
2	1	2	**5TH 4th reference point machine coordinates**

［Data unit］

Setting unit	Data unit
Metric machine	mm
Inch machine	inch
Rotary axis	deg

［Data range］ -9999.9999~9999.9999

Default value 0

225	**Axis name definition of Y**
226	**Axis name definition of 4th**
227	**Axis name definition of 5th**

［Data range］ 2,3,4,5

Axis name	Setting value	Axis name	Setting value
A	3	C	5
B	4	Y	2

Default value 2(225),3(226),4(227)

237	**Surface speed control as the axis of the counter reference**

［Data range］

Setting value	Meaning
0	X axis
1	Z axis
2	Y axis
3	4^{th} axis
4	5^{th} axis

Default value 0

238	**Arc interpolation control precision**

［Data range］ 0~0.5

Default value 0.03

240	X axis pitch error compensation ratio
241	Z axis pitch error compensation ratio
242	Y axis pitch error compensation ratio
243	4TH axis pitch error compensation ratio
244	5TH axis pitch error compensation ratio

［Data range］ 0~9999.9999

Default value 0.001

245	**Reverse gap compensation to determine the reverse accuracy (X0.0001**

［Data unit］

Setting unit	Data unit
Metric machine	mm
Inch machine	inch

［Data range］ 000.1~1

Default value 0.01

2	4	6	Compensation step for X axis space with fixed frequency
2	4	7	Compensation step for Z axis space with fixed frequency
2	4	8	Compensation step for Y axis space with fixed frequency
2	4	9	Compensation step for 4TH axis space with fixed frequency
2	5	0	Compensation step for 5TH axis space with fixed frequency

［Data range］ 0~99.9999

Default value 0.003

2	5	1	**Time constant of the reverse gap to the lifting speed mode**

［Data unit］ ms

［Data range］ 0~400

Default value 20

2	5	2	**With the constant acceleration deceleration clamping index**

［Data unit］ ms

［Data range］ 0~1000

Default value 50

2	5	3	**Exponential type and FL speed**

［Data unit］

Setting unit	Data unit
Metric machine	mm/min
Inch machine	inch/min

［Data range］ 0 ～9999

Default value 10

2	5	4	The constant the handwheel incomplete operation mode acceleration

［Data range］ 0~1000

Default value 50

2	5	5	Deep hole tapping cycle or blank back value

［Data unit］

Setting unit	Data unit
Metric machine	mm/min
Inch machine	inch/min

［Data range］ 0 ～100

Default value 0

2	5	6	Deceleration time constant linear spindle and tapping (first gear)
2	5	7	Deceleration time constant linear spindle and tapping (second gear)
2	5	8	Deceleration time constant linear spindle and tapping (third gear)

［Data unit］ ms

［Data range］ 0~9999

Default value 200

2	5	9	The time constant of the spindle and the tapping when retracting (first gear)
2	6	0	The time constant of the spindle and the tapping when retracting (second gear)
2	6	1	The time constant of the spindle and the tapping when retracting (third gear)

［Data unit］ ms

［Data range］ 0~9999

Default value 200

2	6	3	Principal axis instruction times multiplication factor (CMR) (first gear)
2	6	4	Principal axis instruction times multiplication factor (CMR) (second gear)
2	6	5	Principal axis instruction times multiplication factor (CMR) (third gear)

［Data range］ 0~9999

Default value 512

2	6	6	The main axis of the fractional frequency factor (CMD) (first gear)
2	6	7	The main axis of the fractional frequency factor (CMD) (second gear)
2	6	8	The main axis of the fractional frequency factor (CMD) (third gear)

［Data range］ 0~9999

Default value 125

2	7	0	X axis offset of the origin of the external workpiece
2	7	1	Z axis offset of the origin of the external workpiece
2	7	2	Y axis offset of the origin of the external workpiece
2	7	3	4TH axis offset of the origin of the external workpiece
2	7	4	5TH axis offset of the origin of the external workpiece

［Data unit］

Setting unit	Data unit
Metric machine	mm
Inch machine	inch

［Data range］ -999.999~999.999

Default value 0

2	7	5	G54 Offset of the Y axis coordinate system 1
2	7	6	G54 Offset of the 4th axis coordinate system 1
2	7	7	G54 Offset of the 5th axis coordinate system 1
2	7	8	G55 Offset of the Y axis coordinate system 2
2	7	9	G55 Offset of the 4th axis coordinate system 2
2	8	0	G55 Offset of the 5th axis coordinate system 2
2	8	1	G56 Offset of the Y axis coordinate system 3
2	8	2	G56 Offset of the 4th axis coordinate system 3
2	8	3	G56 Offset of the 5th axis coordinate system 3
2	8	4	G57 Offset of the Y axis coordinate system 4
2	8	5	G57 Offset of the 4th axis coordinate system 4
2	8	6	G57 Offset of the 5th axis coordinate system 4
2	8	7	G58 Offset of the Y axis coordinate system 5
2	8	8	G58 Offset of the 4th axis coordinate system 5
2	8	9	G58 Offset of the 5th axis coordinate system 5
2	9	0	G59 Offset of the Y axis coordinate system 6
2	9	1	G59 Offset of the 4th axis coordinate system 6
2	9	2	G59 Offset of the 5th axis coordinate system 6

［Data unit］

Setting unit	Data unit
Metric machine	mm
Inch machine	inch

［Data range］ -9999.999~9999.999

Default value 0

2	9	3	**Positive max. travel of Y(Second travel limit)**
2	9	4	**Positive max. travel of 4th(Second travel limit)**
2	9	5	**Positive max. travel of 5th(Second travel limit)**

[Data unit]

Setting unit	Data unit
Metric machine	mm
Inch machine	inch

[Data range] -9999.999~9999.999

Default value 9999

2	9	6	**Negative max. travel of Y(Second travel limit)**
2	9	7	**Negative max. travel of 4th(Second travel limit)**
2	9	8	**Negative max. travel of 5th(Second travel limit)**

[Data unit]

Setting unit	Data unit
Metric machine	mm
Inch machine	inch

[Data range] -9999.999~9999.999

Default value -9999

3	0	5	L type time constant for the acceleration and deceleration fast X axis
3	0	6	L type time constant for the acceleration and deceleration fast Z axis

[Data unit] ms

[Data range] 1~4000

Default value 100

3	0	7	L type time constant for the acceleration and deceleration fast X axis
3	0	8	S type time constant for the acceleration and deceleration fast Z axis

[Data unit] ms

[Data range] 1~4000

Default value 100

3	0	9	L type time constant for the acceleration and deceleration fast X axis
3	1	0	L type time constant for the acceleration and deceleration fast Z axis

[Data unit] ms

[Data range] 1~4000

Default value 80

3	1	1	E type time constant for the acceleration and deceleration fast X axis
3	1	2	E type time constant for the acceleration and deceleration fast Z axis

[Data unit] ms

[Data range] 1~4000

Default value 60

Volume III Connection

3	1	3

L type time constant for the acce and dece before Cutting feed

［Data unit］ ms
［Data range］ 0~4000

Default value 100

3	1	4

S type time constant for the acce and dece before Cutting feed

［Data unit］ ms
［Data range］ 1~4000

Default value 100

3	1	5

L type time constant for the acce and dece after cutting feed

［Data unit］ ms
［Data range］ 1~4000

Default value 80

3	1	6

E type time constant for the acce and dece after Cutting feed

［Data unit］ ms
［Data range］ 1~4000

Default value 60

3	1	7

JOG feed of each axis line type time constant for the acce and dece

［Data unit］ ms
［Data range］ 1~4000

Default value 100

3	1	8

JOG feed of each axis exponentia time constant for the acce and dece

［Data unit］ ms
［Data range］ 1~4000

Default value 120

3	1	9	**Hand wheel line type time constant for the acce and dece**

［Data unit］ ms
［Data range］ 1~4000

Default value 120

3	2	0	**Hand wheel exponential type time constant for the acce and dece**

［Data unit］ ms
［Data range］ 1~4000

Default value 80

3	2	1	**Line type time constant for the acce and dece in the thread cutting(gear1)**
3	2	2	**Line type time constant for the acce and dece in the thread cutting(gear2)**
3	2	3	**Line type time constant for the acce and dece in the thread cutting(gear3)**

［Data unit］ ms
［Data range］ 1~4000

Default value 100

3	2	4	**S type time constant for the acce and dece in the thread cutting(gear1)**
3	2	5	**S type time constant for the acce and dece in the thread cutting(gear2)**
3	2	6	**S type time constant for the acce and dece in the thread cutting(gear3)**

［Data unit］ ms
［Data range］ 1~4000

Default value 100

3	2	7	**Spindle backlash compensation value(gear1)**
3	2	8	**Spindle backlash compensation value(gear2)**
3	2	9	**Spindle backlash compensation value(gear3)**

［Data unit］

Setting unit	Data unit
Metric machine	mm
Inch machine	inch

［Data range］ 0~9999.999

Default value 0

3	3	0	**Cutting feed position accuracy**

Volume Ⅲ Connection

［Data unit］

Setting unit	Data unit
Metric machine	mm
Inch machine	inch

［Data range］ 0.01~0.5

Default value 0.03

3 3 1 **Circular interpolation method to acceleration limit**

［Data unit］

Setting unit	Data unit
Metric machine	mm/s/s
Inch machine	inch/s/s

［Data range］ 100~5000

Default value 1000

3 3 2 **Low speed limit of the circular interpolation method to the acce**

［Data unit］

Setting unit	Data unit
Metric machine	mm/min
Inch machine	inch/min

［Data range］ 0~2000

Default value 200

3 3 3 **The top of hand wheel does not completely run mode**

［Data unit］

Setting unit	Data unit
Metric machine	mm/min
Inch machine	inch/min

［Data range］ 0~3000

Default value 2000

3 3 4 **The highest clamping speed of the handwheel/step feed**

［Data unit］

Setting unit	Data unit
Metric machine	mm/min
Inch machine	inch/min

［Data range］ 0~3000

Default value 1000

3	4	5	L type time constant before the acce and dece fast Y axis
3	4	6	L type time constant before the acce and dece fast 4TH axis
3	4	7	L type time constant before the acce and dece fast 5TH axis

［Data unit］ ms

［Data range］ 1~4000

Default value 100

3	4	8	S type time constant before the acce and dece fast Y axis
3	4	9	S type time constant before the acce and dece fast 4TH axis
3	5	0	S type time constant before the acce and dece fast 5TH axis

［Data unit］ ms

［Data range］ 1~4000

Default value 100

3	5	1	L type time constant before the acce and dece fast Y axis
3	5	2	L type time constant before the acce and dece fast 4TH axis
3	5	3	L type time constant before the acce and dece fast 5TH axis

［Data unit］ ms

［Data range］ 1~4000

Default value 80

3	5	4	E type time constant before the acce and dece fast Y axis
3	5	5	E type time constant before the acce and dece fast 4TH axis
3	5	6	E type time constant before the acce and dece fast 5TH axis

［Data unit］ ms

［Data range］ 1~4000

Default value 60

3	5	7	Y axis direction and overshoot
3	5	8	4TH axis direction and overshoot
3	5	9	5TH axis direction and overshoot

［Data unit］

Setting unit	Data unit
Metric machine	mm/min
Inch machine	inch/min

［Data range］ -99.9999~99.9999

Default value 0

Volume III Connection

3	6	3	**Gain adjustment data for spindle speed analog output**

［Data range］ 0.98~1.02

Default value 100

3	6	4	**The Max setting value of inverter**

［Data range］ 4000~65536

Default value 65535

3	6	6	**Need to process the total number of parts**

［Data range］ 0~9999

Default value 0

CHAPTER 4 MACHINE DEBUGGING METHODS AND MODES

The trial run methods and steps at initial power on for this C1000T are described in this chapter. The corresponding operation can be performed after the debugging by the following steps.

4.1 Emergency stop and limit

This C1000T system has software limit function, it is suggested that hardware limit is employed by fixing the stroke limit switches in the positive or negative axes. The connection is as follows (2 axes):

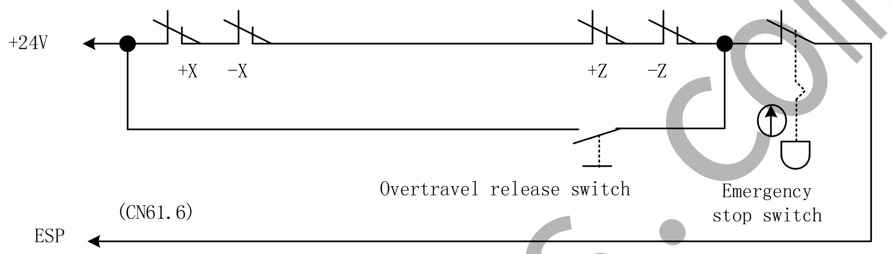


Fig. 4-1

So the BIT3 (ESP) of bit parameter No.172 should be set to 0.
The diagnostic message DGN000.7 monitors the emergency stop input signal.
In Manual or MPG mode, slowly move the axes to testify the validity of stroke limit switch, correctness of alarm display, validity of overtravel release button. When the overtravel occurs or Emergency Stop button is pressed, "emergency stop" alarm will be issued by CNC system. The alarm can be cancelled by pressing down the OVERTRAVEL key for reverse moving.

4.2 Drive unit configuration

BIT4, BIT3, BIT2, BIT1, BIT0 (5ALM, 4ALM, YALM, ZALM, XALM separately corresponds to 5th, 4th, Y, Z, X) of bit parameter No.009 forCNCmakers Limited drive unit are all set to 1 according to the alarm logic level of the drive unit.

If the machine moving direction is not consistent with the move code, modify BIT4, BIT3, BIT2, BIT1 and BIT0 (DIR5, DIR4, DIRY, DIRZ, DIRX separately corresponds to 5th, 4th, Y, Z, X) of bit parameter No.008.

The manual move direction can be set by BIT4,BIT3,BIT2,BIT1, BIT0 (5VAL, 4VAL, YVAL, ZVAL, XVAL separately corresponds to 5th, 4th, Y, Z, X movement key) of bit parameter No.175.

4.3 Gear ratio adjustment

The data parameter No.015～No.018 can be modified for electronic gear ratio adjustment to meet the various mechanical transmission ratio if the machine travel distance is not consistent with the displacement distance displayed by the CNC.

Formula:

$$\frac{CMR}{CMD} = \frac{\delta \times 360}{\alpha \times L} \times \frac{Z_M}{Z_D}$$

Volume III Connection

CMR: Code multiplier coefficient (data parameter No.015, No.016, No.146, No.147, No.148)

CMD: Code frequency division coefficient (data parameter No.017, No.018, No.149, No.150, No.151)

α : Pulse volume, motor rotation angle for a pulse

L: Screw lead

δ: Min. input code unit of CNC(0.001mm for C1000T Z axis, 0.0005mm for X axis of C1000T)

Z_M : gear teeth number of lead screw

Z_D: gear teeth number of motor

Example: if gear teeth number of lead is 50, gear teeth number of motor is 30, pulse volume α =0.075º, screw lead is 4mm,

The electronic gear ratio of X、Z axis is:

$$\frac{CMR}{CMD} = \frac{\delta \times 360}{\alpha \times L} \times \frac{Z_M}{Z_D} = \frac{0.001 \times 360}{0.075 \times 4} \times \frac{50}{30} = \frac{2}{1}$$

Then data parameter No.015(CMRX) =2, No.017(CMDX) =1; No.016(CMRZ) =2, No.018(CMDZ) =1.

If the electronic gear ratio numerator is more than the denominator, the allowed CNC max. speed will decrease. For example: the data parameter No.016(CMRZ) =2, No.018(CMDZ) =1, so the allowed Z axis max. speed is 8000mm/min.

If the electronic gear ratio numerator is not equal to the denominator, the allowed CNC positioning precision may decrease. For example: the data parameter No.016(CMRZ) =1, No.018(CMDZ) =5, so the pulse is not output as the input increment is 0.004, but a pulse is output if the input increment is 0.005.

When matching with the step drive, choose the drive unit with step division function as possible as it can, and properly select mechanical transmission ratio. The 1:1 electronic gear ratio should be ensured to avoid the excessive difference between the numerator and the denominator of this CNC electronic gear ratio.

4.4 ACC&DEC characteristic adjustment

Adjust the relative CNC parameters according to the factors such as the drive unit, motor characteristics and machine load:

Data parameter No.022, No.023, №155, №156, №157: X, Z, Y, 4th, 5th axis rapid traverse rate;

Data parameter No.024, No.025, №158, №159, №160: linear ACC&DEC time constant of X, Z, Y , 4th, 5th axis rapid traverse rate;

Data parameter No.026: X axis exponential ACC&DEC time constant in threading;

Data parameter No.028: Exponential ACC&DEC start/end speed in threading;

Data parameter No.029: Exponential ACC&DEC time constant of cutting and manual feeding;

Data parameter No.030: Exponential ACC&DEC start/end speed in cutting feeding;

BIT5 (SMZ) of bit parameter No.007: for smooth transition between cutting feedrates of adjacent blocks

The larger the ACC&DEC time constant is, the slower the ACC&DEC is, the smaller the machine movement impact and the lower the machining efficiency is, and vice versa.

If ACC&DEC time constants are equal, the higher the ACC&DEC start/end speed is, the faster the ACC&DEC is, the bigger the machine movement impact and the higher the machining efficiency is, and vice versa.

The principle for ACC&DEC characteristic adjustment is to properly reduce the ACC&DEC time constant and increase the ACC&DEC start/end speed to improve the machining efficiency on the condition that there is no alarm, motor out-of-step and obvious machine impact. If the ACC&DEC time constant is set too small, and the start/end speed is set too large, it is easily to cause faults such as drive unit alarm, motor out-of-step or machine vibration.

When the bit parameter No.007 BIT5(SMZ) =1, the feedrate drops to the start speed of the ACC&DEC at the cutting path intersection, then it accelerates to the specified speed of the next block to obtain an accurate positioning at the path intersection, but this will reduce the machining efficiency. When BIT5=0, the adjacent cutting path transits smoothly by the ACC&DEC. The feedrate does not always drop to the start speed when the previous path is finished and a circular transition (non-accurate positioning) will be formed at the path intersection. The machining surface by this path transition has a good finish and a higher machining efficiency. When the stepper motor drive unit is applied, the BIT5 of the bit parameter No.007 should be set to 1 to avoid the out-of-step.

When the stepper motor drive unit is applied, the out-of-step may occur on the condition that rapid traverse speed is too large, ACC&DEC time constant is too small, ACC&DEC start/end speed is too large. The suggested parameter setting is as follows (the electronic gear ratio 1:1):

Data parameter No.022≤2500	Data parameter No.023≤5000
Data parameter No.155≤5000	Data parameter No.158≥350
Data parameter No.024≥350	Data parameter No.025≥350
Data parameter No.029≥150	Data parameter No.028≤100
Data parameter No.026≥200	Data parameter No.030≤50

If AC servo drive unit is applied to this system, the machining efficiency can be improved by a larger start speed and a smaller ACC&DEC time constant setting. If optimum acc/dec characteristics are required, the ACC&DEC time constant may be set to 0 which can be gotten by adjusting the AC servo acc/dec parameters. The suggested settings for these parameters are as follows(electronic gear ratio is 1:1):

Data parameter No.022＝5000	Data parameter No.023＝10000
Data parameter No.155＝10000	Data parameter No.158≤60
Data parameter No.024≤60	Data parameter No.025≤60
Data parameter No.029≤50	Data parameter No.028≤500
Data parameter No.026≤50	Data parameter No.030≤400

The parameter settings above are recommended for use, please refer to the actual conditions of the drive unit, motor characteristic and motor load for their proper setting.

Related signal

DECX: X axis deceleration signal;

DECY: Y axis deceleration signal;

DECZ: Z axis deceleration signal;

DEC4: 4TH axis deceleration signal;

DEC5: 5TH axis deceleration signal;

PCX: X axis zero signal;

PCY: Y axis zero signal;

PCZ: Z axis zero signal;

PC4: 4TH axis zero signal;

PC5: 5TH axis zero signal;

DGN DATA

0	0	0					DEC5	DEC4	DECZ	DECY	DECX
Interface pin							CN61.34	CN61.33	CN61.12	CN61.32	CN61.4

Control PAR

K	2	2	DEC4T	DECY	DECZ	DECX				

DEC4T=0: 4th decelerates signal is low level;

=1: 4th decelerates signal is high level。

DECY=0: Y decelerates signal is low level;

=1: Y decelerates signal is high level。

DECZ=0: Z decelerates signal is low level;

=1: Z decelerates signal is high level。

DECX=0: X decelerates signal is low level;

=1: X decelerates signal is high level。

0	0	6			ZMOD					

ZMOD=1: Return to zero mode selection Block before;

=0: Return to zero mode selection after Block;

0	0	7								ZPLS

ZPLS=1: Return to zero mode choice, there is a turn signal;

=0: Return to zero mode choice, without a turn signal。

0	1	2								ISOT

ISOT=1: After electric power, the machine can move quickly and effectively;

=0: After the power, the machine to the zero point, the manual is invalid。

1	8	3				MZR5	MZR4	MZRY	MZRZ	MZRX

MZRX=1: Select the zero direction is negative;

=0: Select the zero direction is positive。

5 Mechanical (machine) zero adjustment

0	3	3	ZRNFL

ZRNFL: Low rate back to zero。

1	1	3	ZRNFH

ZRNFH: X、Z high rate back to zero。

1	7	7	ZRNFHY

ZRNFHY: Y high rate back to zero。

1	7	8	ZRNFH4

ZRNFH4: 4th high rate back to zero。

1	7	9	ZRNFH5

ZRNFH5: 5th high rate back to zero。

Adjust the relevant parameters based on the active level of the connection signal, zero return type and direction applied:

BIT0, BIT1, BIT2，BIT3，BIT4 (ZMX, ZMZ, ZMY，ZM4, ZM5) of the bit parameter No.006: X, Z, Y, 4^{th} axis machine zero return mode B or C selection.

BIT0, BIT1, BIT2, BIT3, BIT4 (ZCX, ZCZ, ZCY, ZC4, ZC5) of the bit parameter No.007: whether a proximity switch is taken as both deceleration and zero signals.

Data parameter No.033: low deceleration speeds of each axis in machine zero return.
Data parameter No.113：high speed of each axis in machine zero return.
BIT0, BIT1, BIT2, BIT3, BIT4(MZRX, MZRZ, MZRY, MZR4, MZR5) of the bit parameter No.183: each axis zero return direction: negative or positive.

Only the stroke limit switch validity is confirmed, can the machine zero return be performed.

The machine zero is usually fixed at the max. travel point, and the effective stroke of the zero return touch block should be more than 25mm to ensure a sufficient deceleration distance for accurate zero return. The more rapid the machine zero return is, the longer the zero return touch block should be. Or the moving carriage will rush over the block and it may affect the zero return precision because of the insufficient deceleration distance.

Usually there are 2 types of machine zero return connection:

① The connection to AC servo motor: using a travel switch and servo motor one-turn signal separately

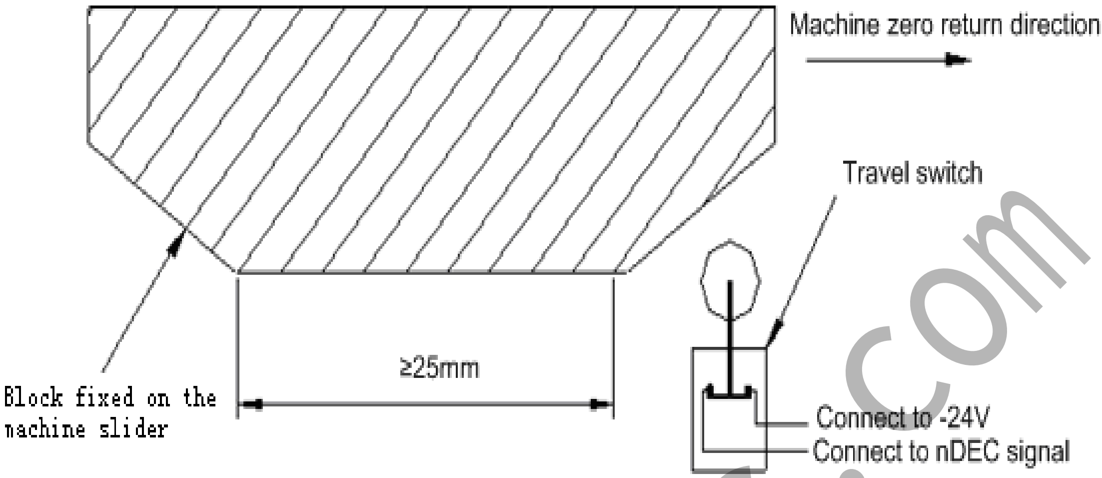


Fig. 4-2

By this connection, when the deceleration switch is released in machine zero return, the one-turn signal of encoder should be avoided to be at a critical point after the travel switch is released. In order to improve the zero return precision, and it should ensure the motor reaches the one-turn signal of encoder after it rotates half circle.

The parameter setting is as follows:

Bit parameter No.004 BIT5(DECI) =0

Bit parameter No.006 BIT0(ZMX) , BIT1(ZMZ) , BIT2(ZMY), BIT3(ZM4), BIT4(ZM5) =0

Bit parameter No.007 BIT0(ZCX) , BIT1(ZCZ) , BIT2(ZCY) , BIT3(ZC4), BIT4(ZC5)=0

Bit parameter No.011 BIT2(ZNLK) =1

Bit parameter No.014 BIT0(ZRSCX) , BIT1(ZRSCZ) , BIT2(ZRSCY), BIT3(ZRSC4), BIT4(ZRSC5) =1

Data parameter No.033=200

Data parameter No.183 BIT0(MZRX) , BIT1(MZRZ) , BIT2(MZRY), BIT3(MZR4) , BIT4(MZR5) =0

② The connection to stepper motor: schematic diagram of using a proximity switch taken as both deceleration signal and zero signal

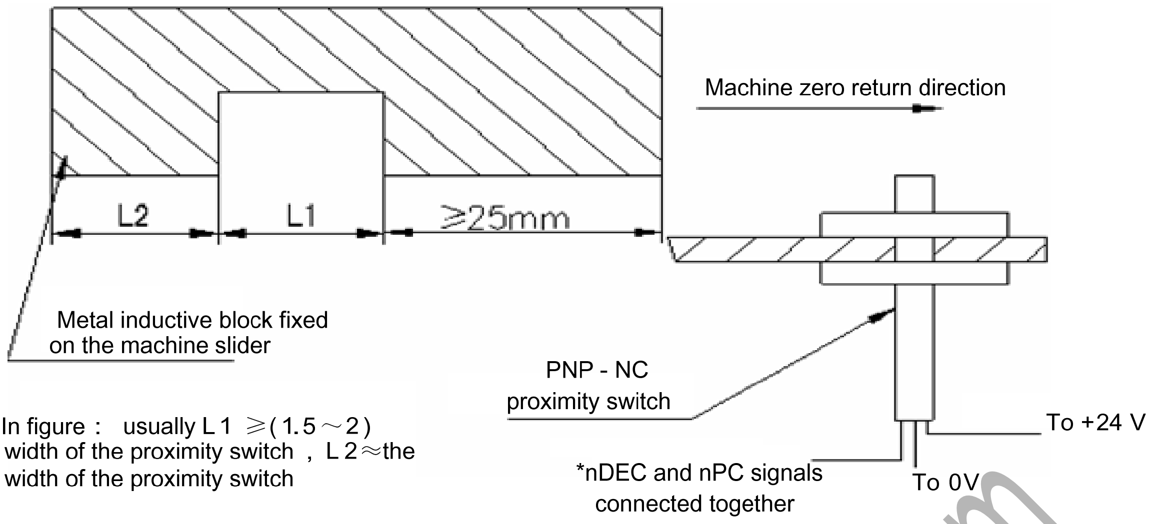


Fig. 4-3

When matching the stepper motor, the parameter settings are as follows:

Bit parameter No.006 BIT5(ZMOD) =0
Bit parameter No.007 BIT2(ZPLS) =0

Data parameter No.033=200
Data parameter No.183 BIT0(MZRX), BIT1(MZRZ), BIT2(MZRY), BIT3(MZR4), BIT4(MZR5) =0

4.6 Spindle adjustment

4.6.1 Spindle encoder

Encoder with the pulses 100~5000p/r is needed to be installed on the machine for threading. The pulses are set by data parameter No.70. The transmission ratio(spindle gear teeth/encoder gear teeth) between encoder and spindle is 1/255～255. The spindle gear teeth are set by CNC data parameter No.110, and the encoder gear teeth are set by data parameter No.111.

4.6.2 Spindle brake

After M05 code is executed, proper spindle brake time should be set to stop the spindle promptly in order to enhance the machining efficiency. If the brake is employed with energy consumption type, too long braking time may burn out the motor.

Data parameter No.087: delay from spindle stop(M05) to spindle brake output

Data parameter No.089: spindle braking time

4.6.3 Switch volume control of spindle speed

When the machine is controlled by a multi-speed motor, the motor speed codes are S01～S04. The relevant parameters are as follows:

State parameter No.001 Bit4=0: select spindle speed switch control;

4.6.4 Analog voltage control of spindle speed

This function can be obtained by the parameter setting of CNC. By interface outputting 0V～10V analog voltage to control frequency inverter, the stepless shift can be obtained. And the related parameters needed to be adjusted are:

Bit parameter No.001 Bit4=1: for spindle speed analog voltage control;

Data parameter No.021: offset value as spindle speed code voltage is 10V;

Data parameter No.036: offset value as spindle speed code voltage is 0V;

Data parameter No.037～ No.040: for max. speed clamping of spindle gear 1～4; it defaults the spindle gear 1 when CNC power on.

Basic parameters are needed to adjust the inverter:

CW or CCW code mode selection: it is determined by terminal VF;

Frequency setting mode selection: it is determined by terminal FR;

If the speed by programming is not consistent with that detected by the encoder, it can be adjusted to be consistent with the actual one by adjusting the data parameter No.037～No.040.

Speed adjustment method: select the corresponding spindle gear, determine the data parameter is 9999 as for this system gear, set the spindle override for 100%. Input spindle run command in MDI mode to run the spindle: M03/M04 S9999, view the spindle speed shown on the right bottom of the screen, then input the speed value displayed into the corresponding system parameter.

When entering S9999 code, the voltage should be 10V, S0 for 0V. If there is a voltage error, adjust bit parameter No.021 and No.036 to correct the voltage offset value (corrected by manufacturer, usually not needed).

For the current max. speed gear, if the analog voltage output by CNC is not 10V, set it for 10V by adjusting the data parameter No.021; when the input speed is 0, if the spindle still slowly rotates, it means the analog voltage output by CNC is higher than 0V, so set a smaller value for data parameter No.036.

If the machine is not fixed with an encoder, the spindle speed can be detected by a speed sensing instrument, input S9999 in MDI mode to set the speed value displayed by the instrument into the data parameter No.037～ No.040.

4.7 Backlash Offset

The X axis backlash offset value is input by diameter, Z axis backlash offset value is input by the actual backlash which can be measured by a dial-indicator, a micrometer or a laser detector. Because the backlash offset can improve the machining precision only by accurate compensation, it is not recommended to measure it in MPG or Step mode, but the following method is suggested:

- Program editing (taking example of Z):

    O0001；

    N10 G01 W10 F800 ；

N20 W15;

N30 W1;

N40 W-1;

N50 M30.

- Set the backlash error offset to 0 before measuring:
- Run the program by single blocks, search the measuring benchmark after 2 positioning operations, record the current data, move 1mm in the same direction, then move 1mm to point B reversely, read the current data.

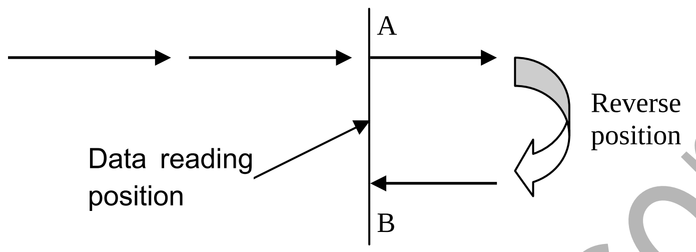


Fig. 4-4 Schematic map of backlash measuring method

- Backlash error offset value= | data of point A –data of point B |; then input its outcome to the data parameter No.034(BKLX) ,No.035(BKLZ) ,No.180(BKLY) ,No.181(BKL4) ,No.182(BKL5) (multiply X axis data to 2 and input the outcome to data parameter No.034).

Data A: dial-indicator data at point A

Data B: dial-indicator data at point B

Note 1: The backlash offset mode can be set by Bit7 of CNC parameter No.011; the backlash frequency can be set by Bit6 of parameter No.011 and Bit4, Bit3, Bit2, Bit1, Bit0 of bit parameter No.010.

Note 2: Check the machine backlash every 3 months.

4.8 Tool Post Debugging

C1000T supports various kinds of tool post, and the parameter settings are based on the machine manual. The parameter settings for the tool post running are as follows:

BIT2(TSGN) of K parameter No.011: high/low level selection of tool post in-position signal, when the signal is low level active, a parallel pull-up resistor is needed.

Bit3 (CTCP) of K parameter No.011: check/do not check tool post lock signal in tool change;

Bit4 (TCPS)of K parameter No.011: tool post lock signal HIGH/LOW selection;

Bit5 (CHET) of K parameter No.11: check/do not check tool signal;

Combinations and functions of tool change mode selection Bit1(CHTB), Bit0(CHTA) of K parameter No.11 are referred to **Tool Change Control**.

Data parameter No.078: Upper limit time for changing one tool

Data parameter No.082: Delay time from tool post CCW stop to CW clamping start

Data parameter No.084: Total tools number

Data parameter No.085: Delay of tool post CW clamping

If the tool post doesn't rotate at first power on for tool change, the phase connection of the 3-phase power of the tool post motor may be incorrect, it needs to press the RESET key immediately and cutoff the power, then check the wiring; if the fault is caused by this, exchange two phases of the

3-phase power.

The CW clamping duration setting should be proper, it should be neither longer nor shorter, longer delay may damage the motor, shorter delay may cause the tool post not to be completely clamped. The method to check the tool post clamping is: approach the dial-indicator to the tool post, turn the tool post manually, and the pointer floating of the dial-indicator should not be over 0.01mm.

During debugging, every tool, max. tools change should be performed to check the correctness of the tool change, time parameter setting.

4.9 Step/MPG Adjustment

The  key on the panel can be used to select the Step mode or MPG mode, which is set by the BIT3 of bit parameter No.001.

Bit3 =1: MPG mode active, Step mode inactive;

=0: Step mode active, MPG mode inactive;

4.10 Other adjustment

K	1	2					CCHU	NYQP	SLSP	SLQP

SLQP =1：chuck control function is valid;

=0：chuck control function is invalid.

SLSP =1：when the chuck function is valid, the system does not check whether the chuck is clamped;

=0：when the chuck function is valid, the system checks whether the chuck is clamped; when the chuck is not clamped and the spindle cannot be stated, the system alarms.

NYQP =1：in outer mode, NQPJ is outer chuck release signal, WQPJ is outer clamp signal;

=0：in inner mode, NQPJ is outer chuck clamp signal, WQPJ is outer release signal.

CCHU =1：the system checks the chuck in-position signal, No.002 Bit7 is inner chuck clamp/outer release signal NQPJ, BIT6 is outer clamp/inner release signal WQPJ, the spindle gear shifting in-position signal M411, M421 is invalid.

=0：the system does not check the chuck in-position signal.

K	1	3							SPTW	SLTW

SLTW =1：tailstock control function is valid;

=0：tailstock control function is invalid.

SPTW=1: the spindle rotation and the tailstock forward/backward does not interlock, the tailstock can execute the tailstock forward/backward no matter what the spindle is in any states;

=0：the spindle rotation and the tailstock forward/backward interlock. When the spindle rotates, the tailstock does not go backward; when it does not go forward, the spindle must not be started.

1	7	2								MST	MSP

MST =0：external cycle start signal (ST) is valid;

=1：external cycle start signal (ST) is invalid,

MSP =0：external pause signal (SP) is valid. At the moment, the system must be connected with the external pause switch, other it alarms “PAUSE”;

=1：external pause signal(SP) is invalid.

CHAPTER 5 DIAGNOSIS MESSAGE

Diagnosis messages for C1000T system are described in this chapter.

5.1 CNC diagnosis

The part is used to check the CNC interface signals and internal running and it can't be modified.

5.1.1 I/O status and data diagnosis message

0 0 0	ESP	***	***	DEC5	DEC4	DECZ	DECY	DECX
Pin	CN61.6			CN61.34	CN61.33	CN61.12	CN61.32	CN61.4
PLC fixed address	X0.5			X2.5	X2.4	X1.3	X2.3	X0.3

DECX, DECY, DECZ, DEC4, DEC5：machine zero return signal of X, Y, Z, 4th, 5th
ESP: emergency stop signal

0 0 1	***	***	***	***	***	***	***	SKIP
Pin								CN61.42
PLC fixed address								X3.5

SKIP：skip signal

5.1.2 CNC motion state and data diagnosis message

0 0 3	***	***	***	RDY5	RDY4	RDYZ	RDYY	RDYX

RDY5～RDYX：X, Y, Z, 4th, 5th ready signal

0 0 4	***	***	***	EN5	EN4	ENZ	ENY	ENX

EN5～ENX：enabling signal

0 0 5	***	***	***	SET5	SET4	SETZ	SETY	SETX

SET5～SETX：pulse prohibit signal

0 0 6	***	***	***	DRO5	DRO4	DROZ	DROY	DROX

DRO5～DROX：X, Y, Z, 4th, 5th motion direction output

0 0 9	***	***	***	ALM5	ALM4	ALMZ	ALMY	ALMX

ALM5～ALMX：X, Y, Z, 4th, 5th alarm signal

0	9	0	X output pulse quantity
0	9	1	Z output pulse quantity
0	9	2	Y output pulse quantity
0	9	3	4TH output pulse quantity
0	9	4	5TH output pulse quantity
1	4	0	MPG count value
1	4	4	Spindle encoder count value

5.1.3 Diagnosis keys

DGN.016～DGN.022 are the diagnosis messages of edit keypad keys; When pressing a key in the operation panel, the corresponding bit displays “1”, and “0” after releasing this key. If it displays reversely, it means there is a fault in the keypad circuit.

0 1 0	9	8	7	P/Q	G	N	O	RST
Key	9	8	7	P Q	G *	N #	O ␣	RESET

0 1 1	6	5	4	W	U	Z	X	PGU
Key	6	5	4	W	U	Z	X	

0 1 2	3	2	1	R	K	J	I	PGD
Key	3	2	1	R V	K C	J B	I A	

0 1 3	-	0	.	T	S	M	RIGHT	CRU
Key	-+ >	0	. / <	T Y	S]	M [	⇨	⇧

0 1 4	ALT	INS	EOB	F/E	D/L	H	LEFT	CRD
Key	ALT	INS	EOB	F E	L D	H =	⇦	⇩

0 1 5	PLC	DGN	PAR	SET	ALM	OFT	PRG	POS
Key	PLC	DGN	PAR	SET	ALM	OFT	PRG	POS

0 1 6	***	***	***	DEL	CAN	CHG	OUT	IN
Key				DEL	CAN	CHG	OUT	IN

Volume III Connection

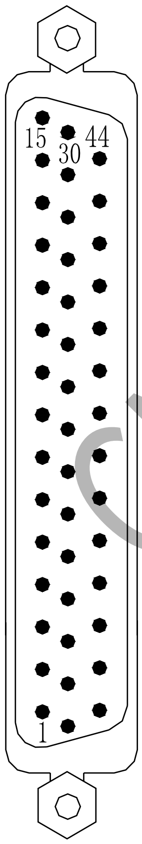
5.1.4 Others

1 4 5	PLC execution time(ms)
1 4 6	Execution all time (h)

5.2 PLC state

This part of diagnosis is used to detect the signal state of machine→PLC(X), PLC→machine(Y), CNC→PLC (F), PLC→CNC (G) and alarm address A, and internal relay (R, K) states.

5.2.1 X address (machine→PLC , defined by standard PLC ladders)

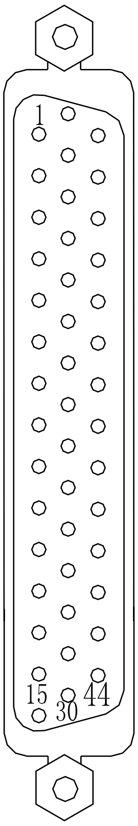


(CN61)

PIN	Address	Function	Explain
21 ~ 24	0V	Power on	Power 0V
18 ~ 20 25 ~ 28	***	***	***
1	X0.0	SAGT	safety door check signal
2	X0.1	SP	external pause
3	X0.2	DIQP	chuck control input
4	X0.3	DECX	X deceleration signal
5	X0.4	DITW	tailstock control input
6	X0.5	KYP	emergency stop signal
7	X0.6	LIMU	Release overtravel input signal
8	X0.7	T05	tool signal T05
9	X1.0	T06	tool signal T06
10	X1.1	T07	tool signal T07
11	X1.2	T08	tool signal T08
12	X1.3	DECZ	Z deceleration signal
13	X1.4	ST	external cycle start
14	X1.5	M41I	auto shifting 1-gear in-position
15	X1.6	M42I	auto shifting 2-gear in-position
16	X1.7	T01	tool signal T01
29	X2.0	T02	tool signal T02
30	X2.1	T03	tool signal T03
31	X2.2	T04	tool signal T04
32	X2.3	DECY	Y deceleration signal
33	X2.4	DEC4	4TH deceleration signal
34	X2.5	DEC5	5TH deceleration signal
35	X2.6	TCP	tool post clamp signal
36	X2.7	AEY/BDT	external skip
37	X3.0	LMIX	X overtravel input

38	X3.1	LMIY	Y overtravel input
39	X3.2	LMIZ	Z overtravel input
40	X3.3	WQPJ	inner chuck release/outer clamp in-position
41	X3.4	NQPJ	outer chuck release/inner clamp in-position
42	X3.5	SKIP	G31 skip signal
43	X3.6	AEX	G36 skip signal
44	X3.7	AEZ	G37 skip signal

5.2.2 Y address (PLC→machine, defined by standard PLC ladders)



(CN62)

PIN	Address	function	Explain
17～19 26～28	0V	Power on	Power 0V
20～25	+24V	Power on	Power+ 24V
1	Y0.0	COOL	cooling output
2	Y0.1	M32	lubricating output
3	Y0.2		Retain
4	Y0.3	M03	spindle rotation(CCW)
5	Y0.4	M04	spindle rotation(CW)
6	Y0.5	M05	spindle stop
7	Y0.6	SCLP	spindle clamped
8	Y0.7	SPZD	spindle brake
9	Y1.0	S1/M41	spindle mechanical 1-gear
10	Y1.1	S2/M42	spindle mechanical 2-gear
11	Y1.2	S3/M43	spindle mechanical 3-gear
12	Y1.3	S4/M44	spindle mechanical 4-gear
13	Y1.4	DOQPJ	chuck clamp
14	Y1.5	DOQPS	chuck release
15	Y1.6	TL+	TL+tool post rotation(CCW)
16	Y1.7	TL-	TL-tool post rotation (CW)
29	Y2.0	TZD	tool post worktable brake
30	Y2.1	INDXS	pregraduation coil
31	Y2.2	CLPY	three-color lamp-yellow
32	Y2.3	CLPG	green
33	Y2.4	CLPR	red
34	Y2.5	DOTWJ	tailstock going forward
35	Y2.6	DOTWS	tailstock going backward
36	Y2.7	VP2	the 2nd speed/position switch output
37	Y3.0	SVF	spindle servo OFF(reduce excitation)
38	Y3.1	HPST	hydraulic control output
39	Y3.2	TAP2	the 2nd gain selection signal
40	Y3.3	M63	the 2nd spindle CW
41	Y3.4	M64	the 2nd spindle CCW
42	Y3.5		Retain
43	Y3.6		Retain
44	Y3.7		Retain

5.2.3 F address(CNC→PLC)

F000	OP	SA	STL	SPL	***	***	***	

OP: Auto run signal
SA: Servo ready signal
STL: Cycle start indicator signal
SPL: Feed hold indicator signal

F001	MA	***	TAP	ENB	DEN	***	RST	AL

MA: CNC ready signal
TAP: Tapping signal
ENB: Spindle enable signal
DEN: Designation end signal
RST: Reset signal
AL: Alarm signal

F002	MDRN	CUT	MSTOP	SRNMV	THRD		RPDO	AL

MDRN: Dry run detection signal
CUT: Cutting feed signal
MSTOP: Select stop detection signal
SRNMV: Program start signal
THRD: Threading signal
RPDO: Rapid feed signal

F003	***	MEDT	MMEM	MRMT	MMDI	MJ	MH	MINC

MEDT: Memory edit selection detection signal
MMEM: Auto run selection detection signal
MRMT: Run selection detection signal
MMDI: MDI selection detection signal
MJ: JOG selection detection signal
MH: MPG selection detection signal
MINC: Increment feed detection signal

F004	***	MPST	MREF	MAFL	MSBK	MABSM	MMLK	MBDT

MPST: Program beginning return detection signal
MREF: Manual reference return detection signal
MAFL: MST lock detection signal
MSBK: Single block detection signal
MABSM: JOG absolute detection signal
MMLK: All machine axes lock detection signal
MBDT: Optional block skip detection signal

F007	***	***	***	***	TF	SF	***	MF

TF: Tool function strobe signal
SF: Spindle speed strobe signal
MF: MST function strobe signal

F009	DM00	DM01	DM02	DM30	***	***	***	***

DM00: M decoding signal
DM01: M decoding signal
DM02: M decoding signal
DM30: M decoding signal

F010	MB07	MB06	MB05	MB04	MB03	MB02	MB01	MB00

MB07: Miscellaneous function code M07
MB06: Miscellaneous function code M06
MB05: Miscellaneous function code M05
MB04: Miscellaneous function code M04
MB03: Miscellaneous function code M03
MB02: Miscellaneous function code M02
MB01: Miscellaneous function code M01
MB00: Miscellaneous function code M00

F014							DRUN	PDBG

PDBG: PLC enter debug mode
DRUN: No switching signal

F015				EN5T	EN4T	ENY		

EN5T: 5TH axis selection
EN4T: 4TH axis selection
ENY: Y axis selection

F018.	AAR07	AAR06 A	AR05 A	AR04	AAR03 A	AR02 A	AR01	AAR00

AR07:Actual speed of spindle AR07
AR06:Actual speed of spindle AR06
AR05:Actual speed of spindle AR05
AR04:Actual speed of spindle AR04
AR03:Actual speed of spindle AR03
AR02:Actual speed of spindle AR02
AR01:Actual speed of spindle AR01
AR00:Actual speed of spindle AR00

F0198	AAR15	AAR14 A	AR13 A	AR12	AAR11 A	AR10 A	AR09	AAR08

AR15:Actual speed of spindle AR15
AR14:Actual speed of spindle AR14
AR13:Actual speed of spindle AR13
AR12:Actual speed of spindle AR12
AR11:Actual speed of spindle AR11
AR10:Actual speed of spindle AR10
AR09:Actual speed of spindle AR09
AR08:Actual speed of spindle AR08

F020							BCLP	BUCLP

BCLP: 4TH axis indexing table clamp signal
BUCLP: 4TH axis indexing table release signal

F022	SB07	SB06	SB05	SB04	SB03	SB02	SB01	SB00

SB07: Spindle speed code signal S07
SB06: Spindle speed code signal S06
SB05: Spindle speed code signal S05
SB04: Spindle speed code signal S04
SB03: Spindle speed code signal S03
SB02: Spindle speed code signal S02
SB01: Spindle speed code signal S01
SB00: Spindle speed code signal S00

F026	TB07	TB06	TB05	TB04	TB03	TB02	TB01	TB00

TB07: Tool code signal T07
TB06: Tool code signal T06
TB05: Tool code signal T05
TB04: Tool code signal T04
TB03: Tool code signal T03
TB02: Tool code signal T02
TB01: Tool code signal T01
TB00: Tool code signal T00

F030	R08O	R07O	R06O	R05O	R04O	R03O	R02O	R01O

R08O: S12 bit code signal R08O
R07O: S12 bit code signal R07O
R06O: S12 bit code signal R06O
R05O: S12 bit code signal R05O
R04O: S12 bit code signal R04O
R03O: S12 bit code signal R03O
R02O: S12 bit code signal R02O
R01O: S12 bit code signal R01O

F031	***	***	***	***	R12O	R11O	R10O	R09O

R12O: S12 bit code signal R12O
R11O: S12 bit code signal R11O
R10O: S12 bit code signal R10O
R09O: S12 bit code signal R09O

F032	X1000	X100	X10	X1			RGSPM	RGSPP

X1000: Step X1000 softkey.
X100: Step X100 softkeyy
X10: Step X10 softkeyy
X1: Step X1 softkeyy
RGSPM: The reversal in rigid tapping
RGSPP: Rigid tapping spindle is in turn

F033								RTAP

RTAP: Rigid tapping mode signal

F034	SSTOP	SCW	Z-	Z+	X-	X+		

SSTOP: Spindle stop softkey

SCW: Rotating softkey

Z-: Z- softkey

Z+: Z+ softkey

X-: X- softkey

X+: X+ softkey

F035	SCCW	MSTOP	AFLO	BDTO	SBKO	MLKO	DRNO	QFAST

SCCW: Spindle counterclockwise softkey

MSTOP: Choose to stop softkey

AFLO: The auxiliary function lock key

BDTO: Hop key program

SBKO: Single program softkey

MLKO: Machine lock key

DRNO: Dry run softkey

QFAST: Fast moving softkey

F036	S-	S+	FAST-	FAST+			FEED-	FEED+

S-: Rate reduction

S+: Rate increase

FAST-: Fast rate reduction

FAST+: Fast rate increase

FEED-: Feed rate reduction

FEED+: Feed rate increase

F037				ZP5	ZP4	ZP3	ZP2	ZP1

ZP5: Program zero return end signal ZP5

ZP4: Program zero return end signal ZP4

ZP3: Program zero return end signal ZP3

ZP2: Program zero return end signal ZP2

ZP1: Program zero return end signal ZP1

F038				MV5	MV4	MV3	MV2	MV1

MV5: Axis move signal MV5

MV4: Axis move signal MV4

MV3: Axis move signal MV3

MV2: Axis move signal MV2

MV1: Axis move signal MV1

F039				**MVD5**	**MVD4**	**MVD3**	**MVD2**	**MVD1**

MVD5: Axis move direction signal MVD5

MVD4: Axis move direction signal MVD4

MVD3: Axis move direction signal MVD3

MVD2: Axis move direction signal MVD2

MVD1: Axis move direction signal MVD1

F040				**ZRF5**	**ZRF4**	**ZRF3**	**ZRF2**	**ZRF1**

ZRF5: Reference point creation signal ZRF5

ZRF4: Reference point creation signal ZRF4

ZRF3: Reference point creation signal ZRF3

ZRF2: Reference point creation signal ZRF2

ZRF1: Reference point creation signal ZRF1

F041				**ZP15**	**ZP14**	**ZP13**	**ZP12**	**ZP11**

ZP15: 5TH Reference point return end signal

ZP14: 4TH Reference point return end signal

ZP13: Y Reference point return end signal

ZP12: Z Reference point return end signal

ZP11: X Reference point return end signal

F042				**PRO5**	**PRO4**	**PRO3**	**PRO2**	**PRO1**

PRO5: Program zero return end signal PRO5

PRO4: Program zero return end signal PRO4

PRO3: Program zero return end signal PRO3

PRO2: Program zero return end signal PRO2

PRO1: Program zero return end signal PRO1

F043								**MSPHD**

MSPHD: Spindle jog detection signal

F044				**SIMSPL**			**FSCSL**	

SIMSPL: Analog spindle active

FSCSL: Cs contour control switch end signal

F047	Total tool number							
F048		**MST**	**MSP**		**MESP**			

MST: Shield external cycle start signal

MSP: Shield external feed hold signal

MKYP: Shield external emergency stop sign

F051				**VAL5**	**VAL4**	**VAL3**	**VAL2**	**VAL1**

VAL5：5TH axis direction selection

VAL4：4TH axis direction selection

VALY：Y axis direction selection

VALZ：Z axis direction selection

VALX：X axis direction selection

F054	UO07	UO06	UO05	UO04	UO03	UO02	UO01	UO00

UO07: Macro output signal UO07

UO06: Macro output signal UO06

UO05: Macro output signal UO05

UO04: Macro output signal UO04

UO03: Macro output signal UO03

UO02: Macro output signal UO02

UO01: Macro output signal UO01

UO00: Macro output signal UO00

F0055	UO15	UO14	UO13	UO12	UO11	UO10	UO09	UO08

UO15: Macro output signal UO15

UO14: Macro output signal UO14

UO13: Macro output signal UO13

UO12: Macro output signal UO12

UO11: Macro output signal UO11

UO10: Macro output signal UO10

UO09: Macro output signal UO09

UO08: Macro output signal UO08

F057				**ZP25**	**ZP24**	**ZP23**	**ZP22**	**ZP21**

ZP25：5TH second reference point return end signal

ZP24：4TH second reference point return end signal

ZP23：Y second reference point return end signal

ZP22：Z second reference point return end signal

ZP21：X second reference point return end signal

F058				ZP35	ZP34	ZP33	ZP32	ZP31

ZP35: 5TH third reference point return end signal
ZP34: 4TH third reference point return end signal
ZP33: Y third reference point return end signal
ZP32: Z third reference point return end signal
ZP31: X third reference point return end signal

F059				ZP45	ZP44	ZP43	ZP42	ZP41

ZP45: 5TH fourth reference point return end signal
ZP44: 4TH fourth reference point return end signal
ZP43: Y fourth reference point return end signal
ZP42: Z fourth reference point return end signal
ZP41: X fourth reference point return end signal

F060								TLIFE

TLIFE: In the same group, the life of all the cutting tools has arrived.

F061								ESEND

KYEND: Required parts to arrive signal

5.2.4 G address(PLC→CNC)

G004					FIN			

FIN: MST function end signal

G005	LEDT	AFL		LAXIS				

LEDT: Edit lock signal
AFL: MST lock signal
LAXIS: All axis interlock sianal

G006		SKIPP		OVC		ABSM	MSTOP	SRN

SRN: Program restart signal
ABSM: Manual absolute signal
OVC: Feedrate override cancel signal
SKIPP: Skip signal
MSTOP: Selective stop signal

G007						ST		

ST: Cycle start signal

G008	ERS	RRW	SP	ESP				

ESP: Emergency stop signal
SP: Feed hold signal
RRW: Reset and cursor return signal
ERS: External reset signal

G009						M12	M32	COOL

M12: Chuck signal
M32: Lubricating signal
COOL: Cooling signal

G0010	JV07	JV06	JV05	JV04	JV03	JV02	JV01	JV00

JV00: JOG override signal JV00
JV01: JOG override signal JV01
JV02: JOG override signal JV02
JV03: JOG override signal JV03
JV04: JOG override signal JV04
JV05: JOG override signal JV05
JV06: JOG override signal JV06
JV07: JOG override signal JV07

G0011	JV15	JV14	JV13	JV12	JV11	JV10	JV09	JV08

JV08: JOG override signal JV08
JV09: JOG override signal JV09
JV10: JOG override signal JV10
JV11: JOG override signal JV11
JV12: JOG override signal JV12
JV13: JOG override signal JV13
JV14: JOG override signal JV14
JV15: JOG override signal JV15

G0012	FV07	FV06	FV05	FV04	FV03	FV02	FV01	FV00

FV00: Feedrate override signal FV00
FV01: Feedrate override signal FV01
FV02: Feedrate override signal FV02
FV03: Feedrate override signal FV03
FV04: Feedrate override signal FV04
FV05: Feedrate override signal FV05
FV06: Feedrate override signal FV06
FV07: Feedrate override signal FV07

G0014	RV8	RV7	RV6	RV5	RV4	RV3	RV2	RV1

RV1: Rapid feedrate override signal RV1
RV2: Rapid feedrate override signal RV2
RV3: Rapid feedrate override signal RV3
RV4: Rapid feedrate override signal RV4
RV5: Rapid feedrate override signal RV5
RV6: Rapid feedrate override signal RV6
RV7: Rapid feedrate override signal RV7
RV8: Rapid feedrate override signal RV8

G016				SAR				

SAR: Spindle speed arrival

G017					DECA	DECY	DECZ	DECX

DECA: 4TH axis back to zero deceleration signal
DECY: Y axis back to zero deceleration signal
DECZ: Z axis back to zero deceleration signal
DECX: X axis back to zero deceleration signal

G018					H4TH	HY	HZ	HX

H4TH: 4TH MPG feed selection signal
HY: Y MPG feed selection signal
HX: X MPG feed selection signal

Volume III Connection

G019	RT		MP2	MP1				

RT: Manual rapid feed selection signal
MP2: MPG override signal MP2
MP1: MPG override signal MP1

G021	SOV7	SOV6	SOV5	SOV4	SOV3	SOV2	SOV1	SOV0

SOV7: Spindle override signal SOV7
SOV6: Spindle override signal SOV6
SOV5: Spindle override signal SOV5
SOV4: Spindle override signal SOV4
SOV3: Spindle override signal SOV3
SOV2: Spindle override signal SOV2
SOV1: Spindle override signal SOV1
SOV0: Spindle override signal SOV0

G022	R08I	R07I	R06I	R05I	R04I	R03I	R02I	R01I

R01I: Spindle motor speed code signal R01I
R02I: Spindle motor speed code signal R02I
R03I: Spindle motor speed code signal R03I
R04I: Spindle motor speed code signal R04I
R05I: Spindle motor speed code signal R05I
R06I: Spindle motor speed code signal R06I
R07I: Spindle motor speed code signal R07I
R08I: Spindle motor speed code signal R08I

G023	SIND	SGN			R12I	R11I	R10I	R09I

R09I: Spindle motor speed code signal R09I
R10I: Spindle motor speed code signal R10I
R11I: Spindle motor speed code signal R11I
R12I: Spindle motor speed code signal R12I
SGN: Spindle motor code polarity selection signal
SIND: Spindle motor speed code selection signal

G024	MRDYA							

MRDYA: Machine ready signal

G025			SRVB	SFRB				

SRVB: Spindle reverse signal
SFRB: Spindle forward signal

G026	CON							

CON: Cs contour control switch signal

G027					+J4	+J3	+J2	+J1

+J4:Feed axis and direction selection signal +J4
+J3:Feed axis and direction selection signal +J3
+J2:Feed axis and direction selection signal +J2
+J1: Feed axis and direction selection signal +J1

G028					-J4	-J3	-J2	-J1

-J4:Feed axis and direction selection signal -J4
-J3:Feed axis and direction selection signal -J3
-J2:Feed axis and direction selection signal -J2
-J1: Feed axis and direction selection signal -J1

G030					+L4	+L3	+L2	+L1

+L4: Axis overtravel signal +L4
+L3: Axis overtravel signal +L3
+L2: Axis overtravel signal +L2
+L1: Axis overtravel signal +L1

G031					-L4	-L3	-L2	-L1

-L4: Axis overtravel signal -L4
-L3: Axis overtravel signal -L3
-L2: Axis overtravel signal -L2
-L1: Axis overtravel signal -L1

G036	BEUCL	BECLP						SPD

BEUCL: Indexing table release signal
BECLP: Indexing table clamp signal
SPD: Spindle point function signal

G037	NT07	NT06	NT05	NT04	NT03	NT02	NT01	NT00

NT07: Current tool No. NT07
NT06: Current tool No. NT06
NT05: Current tool No. NT05
NT04: Current tool No. NT04
NT03: Current tool No. NT03
NT02: Current tool No. NT02
NT01: Current tool No. NT01
NT00: Current tool No. NT00

Volume III Connection

5.2.5 Address A (message display requiery signal, defined by standard PLC ladders)

Address	Alarm No.	Content
A0000.0	1200	Tool change too long
A0000.1	1201	Tool post not in-position alarm as tool change ends
A0000.2	1202	Tool change unfinished alarm
A0000.3	1203	Tool post clamping signal not received
A0000.4	1204	Recheck clamping signal, and clamping signal inactive as tool change ends
A0000.5	1205	Tool change execution is mistaken before power off
A0001.0	1208	M10 and M11 codes disabled for tailstock function inactive
A0001.1	1209	Run-out disabled in spindle running
A0001.3	1211	Spindle start unallowed as tailstock advancing not detected
A0001.4	1212	Cycle unallowed for cycle start disabled
A0001.5	1213	Spindle start unallowed for spindle enable off
A0002.0	1216	Alarm of safe door not closed
A0002.1	1217	Alarm of chuck low pressure
A0002.3	1219	Chuck released unallowed in spindle running
A0002.4	1220	Clamping in-position signal inactive alarm in spindle running
A0002.5	1221	Spindle start unallowed if chuck clamping in-position signal inactive
A0002.6	1222	Spindle start unallowed for chuck releasing
A0003.0	1224	M12/M13 code disabled as chuck inactive
A0003.1	1225	Has not checked the chuck clamped/released start signal
A0003.7	1031	Total tools is more than 4, and the external override cannot be connected
A0004.0	1232	Illegal M code
A0004.1	1233	Spindle jog disabled in non-analog spindle mode
A0004.2	1234	M03, M04 designation error
A0004.4	1236	Spindle gear change time is too long
A0004.5	1237	Spindle speed/position control switch time is too long
A0005.1	1241	Alarm for the abnormal spindle servo or frequency converter for abnormality
A0007.1	1257	Safety door has been opened
A0007.3	1259	Alarm for the tool pot unclocked

CHAPTER 6 MEMORIZING PITCH ERROR COMPENSATION

6.1 Function description

There are more or less precision errors in the pitch of machine axes lead screw, and it will definitely affect the parts machining precision. This C1000T CNC system has the memorizing pitch error offset function that it can accurately compensate the pitch error of the lead screw.

6.2 Specification

1) The offset is concerned with the offset origin, offset intervals, offset point, mechanical moving direction etc.;

2) After performing the machine zero return, take this reference point as the offset origin, and set the offset value into the parameters according to axes offset intervals;

3) Points to be compensated: 256 points for each axis

4) Axis compensated: X, Y, Z ,4th,5th axis

5) Offset range: 0～±127μ m for each offset point

6) Offset interval: 1000～9999999μm;

7) Offset of point N(N=0,1,2,3,…255) is determined by the mechanical error between point N and point N-1;

8) The setting is the same as the CNC parameters input, see **Volume Ⅱ Operation.**

6.3 Parameter setting

6.3.1 Pitch compensation

Bit parameter

0	0	3				PCOMP					

Bit5=1: Pitch error offset active;

Bit5=0: Pitch error offset inactive;

6.3.2 Pitch error origin

A position which the pitch error offset starts from in the offset list, which is determined from the machine zero, is called pitch error offset origin (reference point). This position may be set from 0 to 255 in each axis by data parameter No.098, No.099, depending on the mechanical requirement.

Data parameter

0	9	8	X axis pitch error reference position No.

0	9	9	Z axis pitch error reference position No.

1	8	6	Y axis pitch error reference position No.
1	8	7	4th axis pitch error reference position No.
1	8	8	5th axis pitch error reference position No.

6.3.3 Offset interval

Pitch offset interval: No.102, No.103,No.183,No.184,No.185;

Input unit: metric machine mm; Inch machine inch

Setting range: 1～9999999

State parameter

1	0	2	Pitch error interval of X axis

1	0	3	Pitch error interval of Z axis

1	8	3	Pitch error interval of Z axis
1	8	4	Pitch error interval of 4th axis
1	8	5	Pitch error interval of 5th axis

Note: The offset value is input by diameter

6.3.4 Offset value

The axes pitch offset values are set according to the parameter No. in the following table. The offset value is input by diameter with the input unit 0.001mm, which is irrelevant to diameter or radius programming.

Offset No.	X	Z	Y
000	…	…	…
001	5	-2	3
002	-3	4	-1
…	…	…	…
255	…	…	…

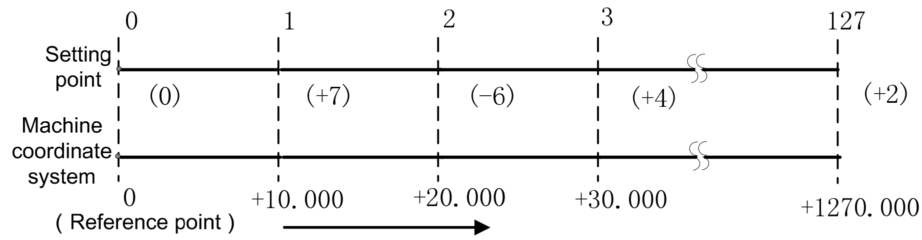
6.4 Notes of offset setting

①The setting and alteration of pitch offset can only be done at the authority of password level 2.

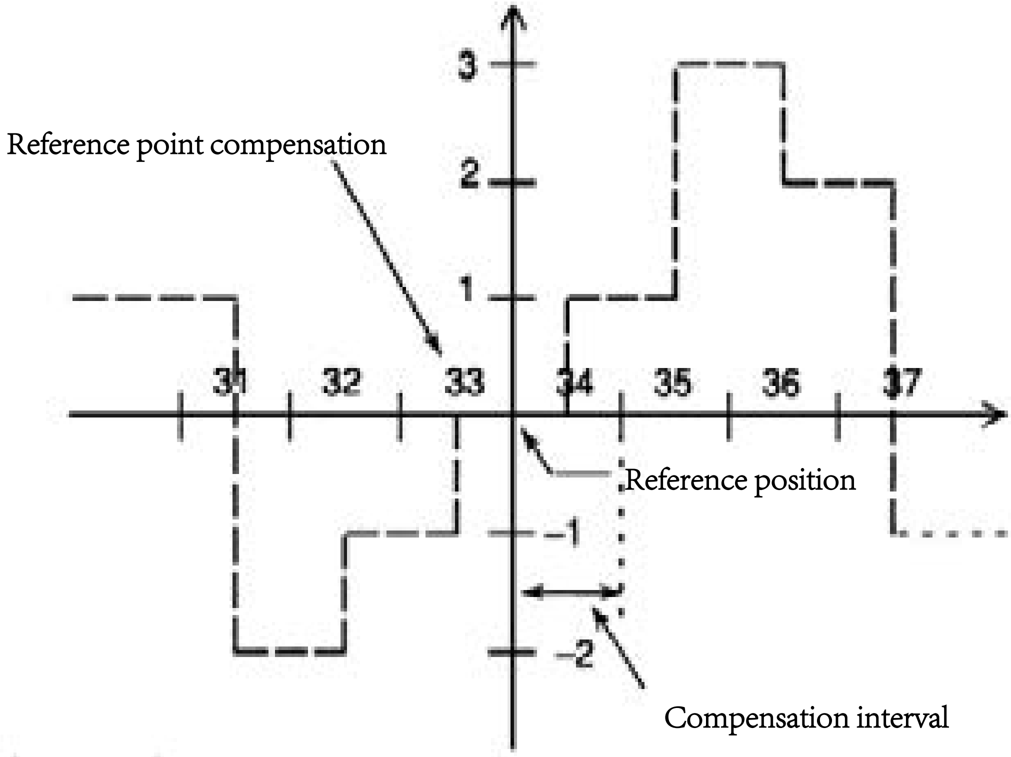
②After the parameter of pitch offset is set, only the machine zero is returned could the offset be done.

6.5 Setting examples of offset parameters

①Data parameter No.99(pitch error origin) =33, Data parameter No.103 (offset interval)=10.000mm When the pitch error origin is set to 33:



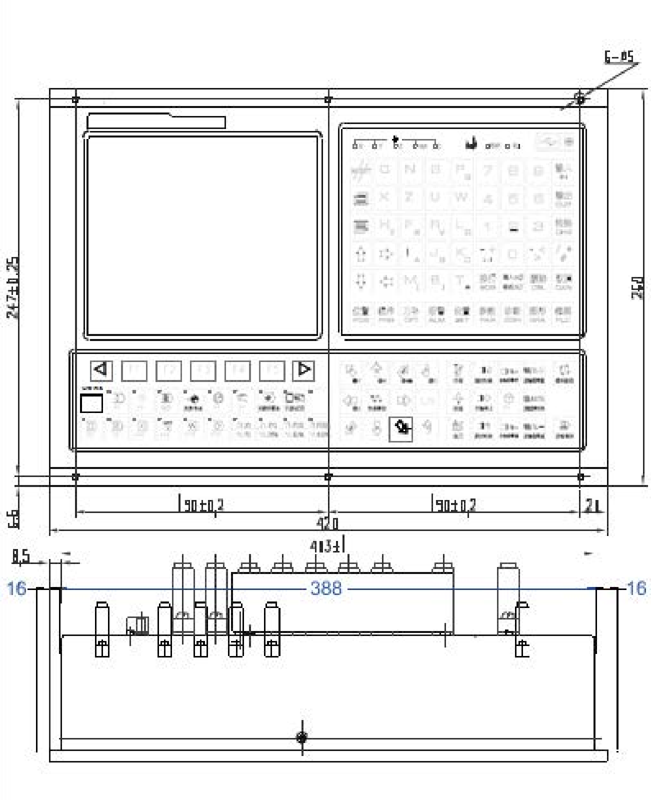
Pitch error compensation (absolute value)

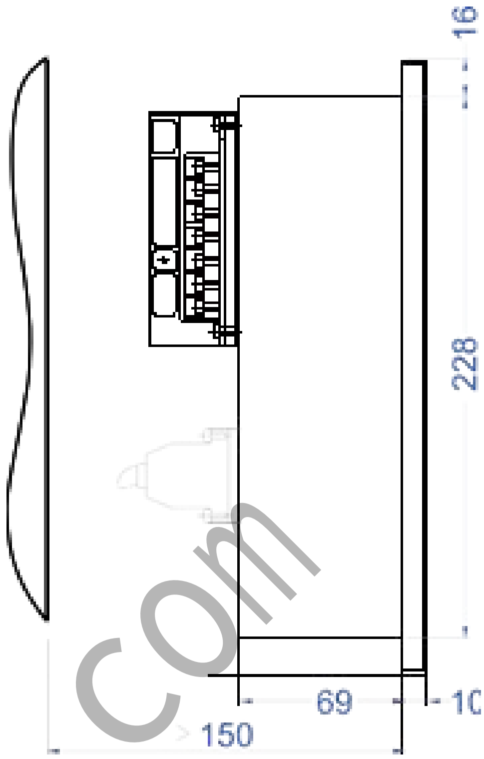


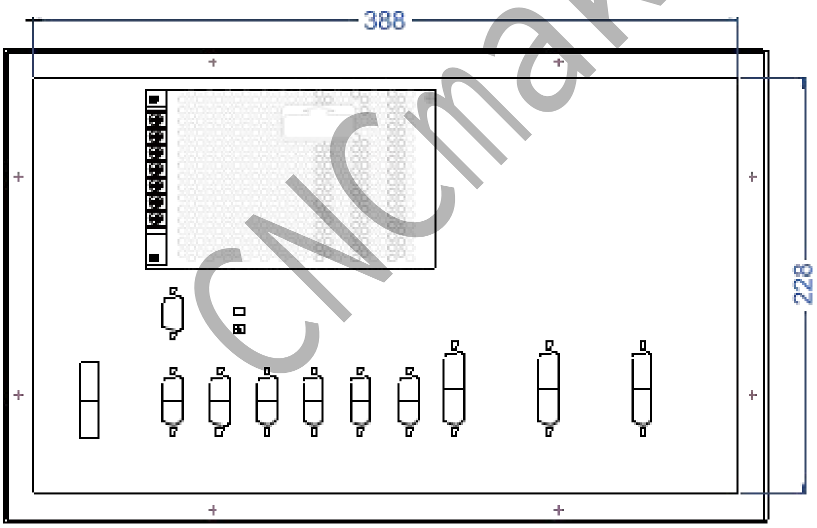
Compensation point	31	32	33	34	35	36	37
Compensation value	+3	−1	−1	+1	+2	−1	−3

Appendix

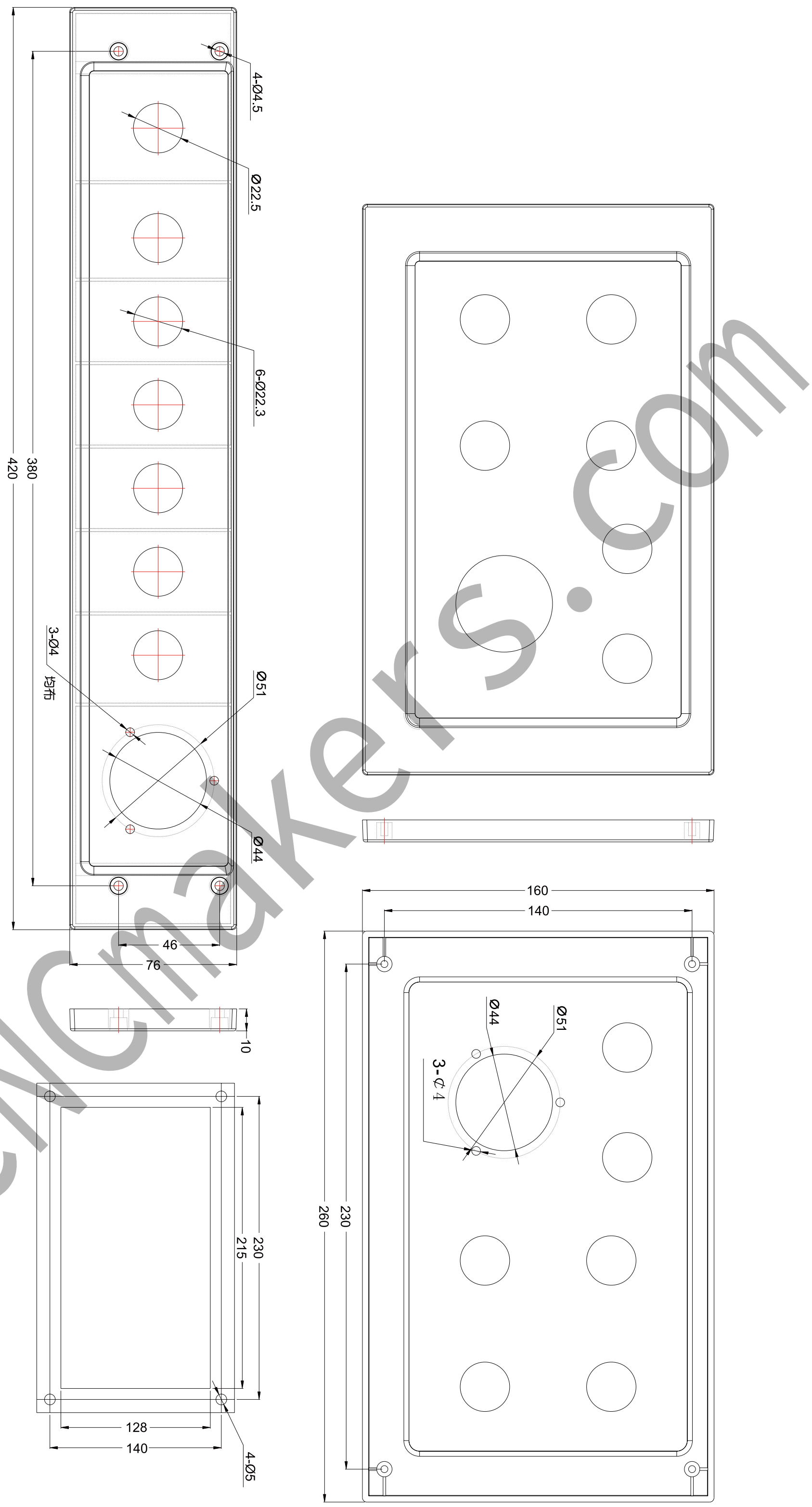
Appendix 1 C1000T contour dimension







Appendix two. Additional panel dimensions



1、CNC Alarm

NO.	Content	Remark
0000	Please power off!	
0001	Fail opening file	
0002	Edited data exceeding limit	
0003	Copy or rename program No. existing.	
0004	No searched address	
0005	No data behind address	
0006	illegal minus	
0007	illegal decimal point	
0008	File too capacity not be loaded.	
0009	Input illegal address	
0010	Incorrect G codes	
0011	No feedrate instruction	
0012	Insufficient disc.	
0013	Too many Files	
0014	Not command G95, spindle not support	
0015	Command too axes	
0016	Cur pitch error comp. point too many!	
0017	No right to alert!	
0018	Not permit to alert	
0019	Cann't use scale!	
0020	Exceed radius tolerance	
0021	Command illegal plane axis	
0022	R and IJK is 0 in arc	
0023	IJK and R specified simultaneously in arc	
0024	Screw interpolation chamfer is 0	
0025	G12 cann't specify with other G code	
0026	Format not supported by system.	
0027	Offset can't share a block with G92.	
0028	illegal plane selection	

0029	illegal offset value	
0030	illegal comp. number	
0031	illegal P commanded in G10	
0032	illegal comp. value in G10	
0033	No intersection in C	
0034	Cann't start or cancel tool comp. in arc	
0035	Not cancel C offset before M99	
0036	Not command G31	
0037	Not change plane in C	
0038	interference in arc block	
0039	Tool nose position error in C	
0040	Workpiece coordinate changed in comp. C	
0041	interference in C	
0042	Over 10 non-traverse instructions in comp. C	
0043	Unauthorized	
0044	No permitting G27~G30 in canned cycle	
0045	Address Q not found (G73/G83)	
0046	illegal reference point return instruction	
0047	Executing machine zero return before it	
0048	Z plane should be higher than R	
0049	Z plane should be lower than R	
0050	Should traverse pos before chang fixed cycle	
0051	Mistaken traverse after chamfer	
0052	Mirror disabled in grooving cycle	
0053	Over address instruction	
0054	DNC carry setting error	
0055	Mistaken traversing value in chamfer or R	
0056	M99 can't share a block with macro	
0057	Save failed.	
0058	Not found end point	
0059	Not found program number	
0060	Not found sequence number	
0061	X axis not in reference point	

Appendix

0062	Y axis not in reference point	
0063	Z axis not in reference point	
0064	4TH axis not at reference point	
0065	5TH axis not at reference point	
0066	Cancel fixed cycle before executing G10	
0067	Setting format not supported by G10.	
0068	PARA SWITCH hasn't turned on	
0069	Need close "U" disk interface as cnc runing	
0070	Memory capacity insufficiency	
0071	Not found data	
0072	Over program quantities	
0073	Program number used	
0074	illegal program number	
0075	Protection	
0076	Address P no defined	
0077	Mistaken subprogram embedding	
0078	Not found sequence number	
0079	System expired.	
0080	Improper input data	
0082	Command H in G37	
0083	illegal axis instruction in G37	
0084	Key overtime or short circuit	
0085	Communication error	
0087	X axis reference point return unfinished	
0088	Z axis reference point return unfinished	
0089	Y axis reference point return unfinished	
0090	4TH axis reference point return unfinished	
0091	5TH axis reference point return unfinished	
0092	Axis not in reference point	
0094	Not permit P type (coordinates)	
0095	Not permit P type (EXT OFS CHG)	
0096	Not permit P type (WRK OFS CHG)	
0097	Not permit P type(automatically execute)	

0098	Found G28 in sequence return	
0099	Not permit executing MDI after searching	
0100	Valid parameter write	
0101	Power-off memory data confused	
0110	Data overflow.	
0111	PC data overflow	
0112	Divided by zero	
0113	Mistaken instruction	
0114	Mistaken G39 format	
0115	illegal variable	
0116	Write protection variable	
0118	Mistaken big brackets embed	
0119	M00~M02,M06,M98,M99,M30 can't at the same block with other block	
0122	Fourfold macro mold-calling	
0123	Not use macro instruction in DNC	
0124	Program illegal completion	
0125	Mistaken macro program format	
0126	illegal cycle number	
0127	NC & macro instruction in the same block	
0128	Sequence number of illegal macro instruction	
0129	illegal independent variable address	
0130	illegal axis operation	
0131	Over external alarm information	
0132	Not found alarm number	
0133	System not support axis instruction	
0134	Axis more than 3 can not use rigid tapping	
0135	illegal angle instruction	
0136	illegal axis instruction	
0139	Can't change PLC control axis	
0142	illegal proportional rate	
0143	Scaling motion data overflow	
0144	illegal plane selection	
0148	illegal data setting	

0149	Format error in G10L3	
0150	illegal tool group number	
0151	Not found tool group number	
0152	Tool data not memorize	
0153	Not cancle C before changing tool	
0154	Tool in unused tool life	
0155	Illegal T code in M06	
0156	Not found P/L instruction	
0157	Over tool group	
0158	illegal tool life data	
0159	Tool data setting unfinished	
0160	Arc only use R prg in polar coordinates mode	
0161	Not execute the instr in polar coordinates	
0162	Have used G70~G76 instructions in MDI mode	
0163	Not execute the instruction in rotation mode	
0164	Not execute the instruction in scaling mode	
0165	Specify the instruction in sole block	
0166	Axis not specified in reference point return	
0167	Coordinates in intermediate point too big	
0168	Min. dwell time should smaller than max.4	
0170	Not cancel comp. in entering or Esc subprg	
0172	P not int or less than 0 in calling subprg	
0173	Subprogram calling times less than 9999	
0175	G17 executed only in canned cycle	
0176	Spindle rotate speed not set	
0177	Not support spindle oriented function	
0178	Spindle rotate speed not set before canned cycle	
0181	illegal M code	
0182	illegal S code	
0183	illegal T code	
0184	Selected tool exceeding limit	
0185	L too small	
0186	L too large	

0187	Tool radius too large	
0188	U too large	
0189	U smaller than tool radius	
0190	V too small or V has not defined	
0191	W too small or W has not defined	
0192	Q too small or Q has not defined	
0193	I has not define or I is zero	
0194	J has not define or J is zero	
0195	D has not define or D is zero	
0198	Illegal axis selection	
0199	Macro instruction not defined	
0200	illegal S mode instruction	
0201	Not found feedrate in rigid tapping	
0202	Position LSI overflow	
0203	Program error in rigid tapping	
0204	Illegal axis operation	
0205	Rigid mode DI signal closed	
0206	Not change plane (rigid tapping)	
0207	Tapping data error	
0208	Cann’t exe. the instruction in G10 .	
0212	illegal plane selection	
0224	Reference point return	
0231	illegal format in G10 L50 or L51	
0232	Commanded spiral interpolation axes too many	
0233	Device busy	
0235	Error completion	
0236	Program restart parameter error	
0237	No decimal point	
0238	Address repetition error	
0239	Parameter 0	
0240	No permitting G41/G42 in MDI	
0241	MPG abnormal	
0243	Spindle plus abnormal	

0244	Thread process speed exceed upper limit value
0245	Spindle rotate speed fluctuation beyond range while thread processing
0251	Emergency stop alarm
0255	Can't specify spindle rotate speed in thread block
0256	Thread lead beyond range
0257	Have used T instruction in the block specify by G71~G73
0258	Have specified M98,M99 or M30 in two block specified by adress P or Q
0259	Have specified adress Z(W)/X(U) in P block in G71/G72 instruction
0260	Name of axis is repeated.Please alter parameters NO.225~227
0261	Tool offset No. beyond range(0~32)
0262	Tool No. beyond range set by data parameter No.084
0263	Tool group No. beyond range(1~32) in mangement of tool life
0264	Can't execute T instruction in C,please revoca C
0265	G70~G76,G90,G92,G94 can only used in G18 panel
0266	Can't execute panel convert instruction G17~G19
0267	Program lacks of G11 or G13.1
0268	There isn't any tool in tool group in the mangement of tool life
0269	Current tool group haven't be defineed in the mangement of tool life
0270	The life of all tools in the same group have reached
0271	Tool life mangement function is invalid,can't use G10 L3 instruction
0272	G11 can't specify before G10
0273	The movement in the X direction doesn't equal 0 in G33 tapping
0274	The number of thread index head is bigger than 65535
0275	R absolute value is greater than U/2 absolute value in G90,G92 instruction
0276	R absolute value is greater than W absolute value in G94 instruction
0277	Finish machining block exceeds 31 in G70~G73 instruction
0278	The sequence of Ns and Nf error in finish machining block in G70~G73
0279	Cycle block No. Ns and Nf isn't exit in G70~G73
0280	Not input cycle start and cycle end block No. in G70~G73
0281	Have call subprogram in G70~G73
0282	Not specify G00 or G01 in cycle start block in G70~G73
0283	Have used prohibitive G instructions in cycle start block in G70~G73
0284	Have used prohibitive G instructions in cycle start block in G70~G73

0285	Have used G70~G73 instruction in MDI mode
0286	The coord variation finish machining block isn't monotonous in G71~G72
0287	Single feed toolamount beyond range in G71 or G72
0288	Single retract tool amount beyond range in G71 or G72
0289	Have specified Z or W in the first block in G71
0290	Have specified X or U in the first block in G72
0291	The total cutting amount of G73 beyond range
0292	The cycle time of G73 is less than 1 or greater than 9999
0293	Single retract tool amount of G74 or G75 beyond range
0294	Single retract tool amount of cutting to terminal in G74 or G75 is negative
0295	Single cutting amount in the direction of X or Z in G74 or G75 beyond range
0296	Not input Z value in G74
0297	Q value is 0 or not input in G74
0298	Not input X value in G75
0299	P value is 0 or not input in G75
0300	Start point in G76 process cone thread is between start point and end point
0301	Min. cut-in amount in G76 beyond range
0302	Finish process margin in G76 beyond range
0303	Tooth height is less than finsih process margin or less than 0 in G76
0304	Cycle time in G76 beyond range
0305	Thead chamfer width in G76 beyond range
0306	Tool nose angle in G76 beyond range
0307	The movement of X or Z axis is zero in G76
0308	Not specify thread tooth height P value in G76
0309	Not specify 1st cutting depth Q value or Q value is 0 in G76
0310	Starting point in the closed area of the start and end of the locus
0311	Thread pitch is less than 0 in variational thread pitch thread cutting
0312	Tooth height is less than X axis movement in G76
0320	Not chamfer function in append axis instruction
0321	Use WHILE,END instruction in MDI mode
0322	Maro format error
0323	DO,END label is not 1,2,3 in macro statement
0324	DO,END format error in macro statement

0325	Bracket doesn't match or format error in macro statement
0326	Divisor can't be 0 in macro statement
0327	Arc tangent ATAN format error in macro statement
0328	Anti-logarithm of LN is 0 or less than 0 in macro statement
0329	The square of the macro statement can not be negative.
0330	Result of Tangent TAN is infinity in macro statement
0331	Operand of ASIN or ACOS beyond range(-1~1) in macro statement
0332	Macro variable or variable value is illegal in macro statement
0451	X axis driver alarm.
0452	Z axis driver alarm.
0453	Y axis driver alarm.
0454	4TH axis driver alarm.
0455	5TH axis driver alarm.
0456	Spindle driver alarm.
0500	Software limit overtravel:-X
0501	Software limit overtravel:+X
0502	Software limit overtravel:-Z
0503	Software limit overtravel:+Z
0504	Software limit overtravel:-Y
0505	Software limit overtravel:+Y
0506	Software limit overtravel:-4Th
0507	Software limit overtravel:+4Th
0508	Software limit overtravel:-5Th
0509	Software limit overtravel:+5Th
0510	Hardware limit overtravel:-X
0511	Hardware limit overtravel:+X
0512	Hardware limit overtravel:-Z
0513	Hardware limit overtravel:+Z
0514	Hardware limit overtravel:-Y
0515	Hardware limit overtravel:+Y
0516	Hardware limit overtravel:-4TH
0517	Hardware limit overtravel:+4TH
0518	Hardware limit overtravel:-5TH

0519	Hardware limit overtravel:+5TH
0740	Rigid tapping alarm: overproof
0741	Rigid tapping alarm: overproof
0742	Rigid tapping alarm:LSI 溢出
0751	Check the first spindle alarm(AL-XX)
0754	Abnormal torque alarm
1001	Address of relay or coil not set
1002	Input code inexistence
1003	COM/COME used by mistake .
1004	ladder exceeding max. linage or step .
1005	Error in END1/END2.
1006	illegal output in NET.
1007	Hardware failure or system interrupt error causes PLC to communicate
1008	Not connected correctly.
1009	Network horizon not connected .
1010	Network missing for power-off in edit ladder .
1011	Address data not input correctly .
1012	Symbol undefined or data exceeding limit .
1013	Defined illegal characters .
1014	CTR adress is repeated .
1015	JMP/LBL deal err or exceeding its capacity .
1016	Network struct is incomplete .
1017	Network struct isn’t supported .
1019	TMR address repeat .
1020	No parameter in function instruction .
1021	PLC execution timeout, the system automatically stops PLC.
1022	Function instruction name lost .
1023	Functional address or constant overflow .
1024	Unnecessary relay or coil exist .
1025	Coil not correctly output .
1026	Line number of network connection overflow .
1027	One symbol name defined in another place.
1028	Ladder format error .

1029	Ladder being used lost .	
1030	Incorrect vertical line in NET .	
1031	Data full, reducing COD instr. data capacity .	
1032	First grade of ladder too big .	
1033	SFT instruction exceeding max. capacity.	
1034	DIFU/DIFD used mistakenly.	
1035	Current opened ladder convert failed	
1036	PLC emergency stop alarm	
1037	Opened and data para setting ladder isn't same	
1039	Instruction or network not within range	
1040	CALL/SP/SPE used mistakenly .	
1041	Horizonal line parallels to node net.	
1042	PLC parameter file not loaded	

Appendix 10 Operation list

Type	Function	Operation	Opera-tion mode	Display window	Password level	Program switch	Para-meter switch	Remark
Clear	X incremental coordinate clear	U , CANCEL		Incremental coordinate				Volume II Section 1.3.1

Type	Function	Operation	Opera-tion mode	Display window	Password level	Program switch	Para-meter switch	Remark
	Z incremental coordinate clear	W, CANCEL		Incremental coordinate				
	Workpiece clear	CANCEL + N		Incremental coordinate or absolute coordinate				
	Cutting time clear	CANCEL + T						
	X tool offset clear	X, DATA INPUT		Tool offset	2-level, 3-level, 4- level			Volume II Section 7.4.4
	Z tool offset clear	Z, DATA INPUT		Tool offset	2- level, 3- level, 4 -level			Volume II Section 7.4.4
Data setting	state parameter	parameter value, DATA INPUT	MDI mode	state parameter	2-level,3-level,		ON	Volume II Section 10.1.3
	Data parameter	parameter value, DATA INPUT	MDI mode	data parameter	2-level,3-level		ON	
	X pitch parameter input	X, compensation value, DATA INPUT	MDI mode	pitch compensation parameter	2-level		ON	Volume II Section 10.1.3
	Z pitch parameter input	Z, compensation value, DATA INPUT	MDI mode	pitch compensation parameter	2-level		ON	Volume II Section 10.1.3
	Macro variable	macro variable value, DATA INPUT		macro variable	2-level,3-level, 4-level			Volume II Section 1.3.3
	X tool offset incremental input	U, offset increment		tool offset	2-level,3-level, 4-level			Volume II Section 7.4.2
	Z tool offset incremental input	W, offset increment		tool offset	2-level,3-level, 4-level			Volume II Section 7.4.2
Search	Search downward from the cursor's current position	character, ⇩	EDIT mode	program content	2-level,3-level, 4-level	ON		Volume II Section 6.1.3
	Search upward from the cursor's current position	character, ⇧	EDIT mode	program content	2-level,3-level, 4-level	ON		Volume II Section 6.1.3

Type	Function	Operation	Opera-tion mode	Display window	Password level	Program switch	Para-meter switch	Remark
	Search downward from the current program	O, ⇩	EDIT mode or AUTO mode	program content program contents or program state	2-level,3-level, 4-level			Volume Ⅱ Section 6.4.1
	Search upward from the current program	O, ⇧			2-level,3-level, 4-level			Volume Ⅱ Section 6.4.1
	Search specified program	O ,program name, ⇩			2-level,3-level, 4-level			Volume Ⅱ Section 6.4.2
	Search bit parameters, data parameters or pitch compensation parameters	P Q, parameter number, DATA INPUT		Corresponding page of data				Volume Ⅱ Section 10.1.3
	PLC state, PLC data search	P Q ,address number, DATA INPUT		PLC state PLC data				Volume Ⅱ Section 1.3.7
Delete	character deletion at the cursor	DELETE	EDIT mode	program content	2-level,3-level, 4-level	ON		Volume Ⅱ Section 6.1.6
		CANCEL	EDIT mode	program content	2-level,3-level, 4-level	ON		
	Single block deletion	Move the cursor to the head, DELETE	EDIT mode	program content	2-level,3-level, 4-level	ON		Block No. in block，Volume Ⅱ Section 6.1.7
	Blocks deletion	CHANGE, N ,sequence number, DELETE	EDIT mode	program content	2-level,3-level, 4-level	ON		Volume Ⅱ Section 6.1.8
	Segment deletion	CHANGE ,character, DELETE	EDIT mode	program content	2-level,3-level, 4-level	ON		Volume Ⅱ Section 6.1.9
	Single program deletion	O ,program name, DELETE	EDIT mode	program content	2-level,3-level, 4-level	ON		Volume Ⅱ Section 6.3.1
	All programs deletion	O, - + 999, DELETE	EDIT mode	program content	2-level,3-level, 4-level	ON		Volume Ⅱ Section 6.3.2

Type	Function	Operation	Opera-tion mode	Display window	Password level	Program switch	Para-meter switch	Remark
Rename	Program rename	O ,program name, INSERT ALTER	EDIT mode	program content	2-level,3-level, 4-level	ON		Use 2-level when the program number is more than or equal to 9000 Volume Ⅱ Section 6.6
Copy	Program copy	O ,program name, CHANGE	EDIT mode	program content	2-level,3-level, 4-level	ON		Use 2-level when the program number is more than or equal to 9000 Volume Ⅱ Section 6.7
CNC, CNC（send）	tool offset	DATA OUTPUT	EDIT mode	tool offset	2-level,3-level		ON	Volume Ⅱ Section 11.6
	state parameter	DATA OUTPUT	EDIT mode	state parameter	2-level,3-level		ON	
	data parameter	DATA OUTPUT	EDIT mode	data parameter	2-level,3-level		ON	
	pitch compensation parameter	DATA OUTPUT	EDIT mode	pitch compensation parameter	2-level		ON	
	Send a program	O ,program name, DATA OUTPUT	EDIT mode	program content	2-level,3-level, 4-level	ON		
	Send all programs	O , -+ 999, DATA OUTPUT	EDIT mode	program content	2-level,3-level, 4-level	ON		
CNC→CNC（receive）	tool offset		EDIT mode		2-level,3-level, 4-level		ON	Volume Ⅱ Section 11.6
	state parameter		EDIT mode		2-level,3-level		ON	
	data parameter		EDIT mode		2-level,3-level		ON	
	pitch compensation parameter		EDIT mode		2-level		ON	
	Part program		EDIT mode		2-level,3-level, 4-level	ON		
CNC→PC (upload)	tool offset	DATA OUTPUT	EDIT mode	tool offset	2-level,3-level 4-level		ON	Volume Ⅱ Section 11.5.3
	state parameter	DATA OUTPUT	EDIT mode	state parameter	2-level,3-level 4-level		ON	Volume Ⅱ Section 11.5.4
	data parameter	DATA OUTPUT	EDIT mode	data parameter	2-level,3-level		ON	

Type	Function	Operation	Opera-tion mode	Display window	Password level	Program switch	Para-meter switch	Remark
	pitch compensation parameter	DATA OUTPUT	EDIT mode	pitch compensation parameter	2-level		ON	
	Send a part program	O ,program name, DATA OUTPUT	EDIT mode	program content	2-level,3-level 4-level	ON		Volume II Section 11.5.1
	Send all part programs	O , -+ 999, DATA OUTPUT	EDIT mode		2-level,3-level, 4-level	ON		Volume II Section 11.5.2
PC, CNC (down-load)	tool offset		EDIT mode		2-level,3-level, 4-level		ON	Volume II Section 11.4.2
	state parameter		EDIT mode		2-level,3-level		ON	Volume II Section 11.4.3
	data parameter		EDIT mode		2-level,3-level		ON	
	pitch compensation parameter		EDIT mode		2-level		ON	Volume II Section11.4.3 , 2-level
	Part program		EDIT mode		2-level,3-level, 4-level	ON		Volume II Section 11.4.1, use 2-level when the program number is more than or equal to 9000
Switch setting	Parameter switch ON	L		Switch setting	2-level,3-level			Volume II Section10.1.1
	Program switch ON	L		Switch setting	2-level,3-level, 4-level			
	Automatic sequence number ON	L		Switch setting				
	Parameter switch OFF	W		Switch setting	2-level,3-level			
	Program switch OFF	W		Switch setting	2-level,3-level, 4-level			
	Automatic sequence number OFF	W		Switch setting				

Note 1: "," in "Operation" indicates that the two operations are successive, "+" indicates that the two operations are executed at the same time.

Example: " U , CANCEL " indicates that we firstly press U and then press CANCEL ; " CANCEL + N " indicates these two keys are pressed simultaneously.

Note 2: The blanks in Operation Mode, Display Window, Password Level, Program Switch and Parameter Switch column indicate that the corresponding switches are not related to their items correspondingly.

Appendix

CNCmakers Limited
Address: No.168, Xiadu Road, Haizhu District, Guangzhou, China 510300
Email: info@CNCmakers.com Website: www.CNCmakers.com Tel: +86-138-24444158 Fax: +86-20-84185336