
 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.

 This user manual is the property of KY980Tc CNC Equipment Co., Ltd. All rights are reserved. It is against the law for any organization or individual to publish or reprint this manual without the express written permission of KY980Tc and the latter reserves the right to ascertain their legal liability.

FOREWORD

Dear user,

We are really grateful for your patronage and purchase of this **KY-980Tc CNC system** made by KY CNC Equipment Co., Ltd.

The user manual describes the programming, operation, installation and connection of this **KY-980Tc 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 KY'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 can'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

KY-980Tc CNC Operation Use

Volume III Installation and Connection

KY-980Tc 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

CHAPTER 1 PROGRAMMING	3
1.1 980Tc introduction	3
1.1.1 Product introduction.....	3
1.1.2 Technical specification	4
1.1.3 Environment and conditions.....	6
1.1.4 Power supply	7
1.1.5 Guard.....	7
1.2 CNC system of machine tools and CNC machine tools	7
1.3 Programming fundamentals	9
1.3.1 Coordinates definition	9
1.3.2 Machine coordinate system, Machine Zero and machine reference point	9
1.3.3 Workpiece coordinate system and Program Zero	10
1.3.4 Interpolation function	11
1.3.5 Absolute programming and relative programming	12
1.3.6 Diameter programming and radius programming	12
1.4 Structure of an NC program	13
1.4.1 General structure of a program.....	14
1.4.2 Main program and subprogram.....	17
1.5 Program run.....	18
1.5.1 Sequence of program run	18
1.5.2 Execution sequence of word.....	19
CHAPTER 2 MSTF COMMAND	20
2.1 M (miscellaneous function)	20
2.1.1 End of program M02	20
2.1.2 End of program run M30	20
2.1.3 Subprogram call M98	21
2.1.4 Return from subprogram M99	21
2.1.5 M commands defined by standard PLC ladder diagram	22
2.1.6 Program stop M00.....	23
2.1.7 Program optional stop M01.....	23
2.1.8 Spindle CW, CCW and stop control M03, M04, M05	24
2.1.9 Cooling control M08, M09	24
2.1.10 Tailstock control M10, M11	24
2.1.11 Chuck control M12, M13	24

2.1.12	Spindle position/speed control switch M14, M15	24
2.1.13	Spindle clamped/released M20, M21	25
2.1.14	The 2 nd spindle position/speed switch M24, M25	25
2.1.15	Lubricating control M32, M33	25
2.1.16	Spindle automatic gear change M41, M42, M43, M44	25
2.1.17	The 2 nd spindle rotation CCW, rotation CW , stop M63, M64, M65	26
2.2	Spindle function	26
2.2.1	Spindle speed switching value control	26
2.2.2	Spindle speed analog voltage control	27
2.2.3	Constant surface speed control G96, constant rotational speed control G97	27
2.2.4	Spindle override	28
2.3	Tool function	30
2.3.1	Tool control	30
CHAPTER 3 G COMMANDS		34
3.1	Commands	34
3.1.1	Modal, non-modal and initial mode	35
3.1.2	Omitting words	36
3.1.3	Related definitions	36
3.2	Rapid traverse movement G00	37
3.3	Linear interpolation G01	38
3.4	Circular interpolation G02, G03	40
3.5	Plane selection G17 ~ G19	43
3.6	Chamfering function	43
3.6.1	Linear chamfering	44
3.6.2	Circular chamfering	45
3.6.3	Special cases	47
3.7	Dwell G04	49
3.8	Machine Zero function	49
3.8.1	Machine 1st reference point G28	49
3.8.2	Machine 2nd, 3rd, reference point G30	50
3.9	Skip interpolation G31	52
3.10	Floating workpiece coordinate system G50	55
3.11	Workpiece coordinate system G54~G59	56
3.12	Fixed cycle command	58
3.12.1	Axial cutting cycle G90	60
3.12.2	Radial cutting cycle G94	61
3.12.3	Caution of fixed cycle commands	63
3.13	Multiple cycle commands	64
3.13.1	Axial roughing cycle G71	64

3.13.2	Radial roughing cycle G72.....	69
3.13.3	Closed cutting cycle G73.....	73
3.13.4	Finishing cycle G70	77
3.13.5	Axial grooving multiple cycle G74	78
3.13.6	Radial grooving multiple cycle G75.....	81
3.14	Thread cutting commands	84
3.14.1	Thread cutting with constant lead G32.....	85
3.14.2	Thread cutting with variable lead G34.....	88
3.14.3	Z thread cutting G33	90
3.14.4	Thread cutting cycle G92.....	92
3.14.5	Multiple thread cutting cycle G76.....	94
3.15	Constant surface speed control G96, constant rotational speed control G97	98
3.16	Feedrate per minute G98, feedrate per rev G99.....	98
3.17	Macro commands	99
3.17.1	MACRO variables	99
3.17.2	Operation and jump command G65	104
3.17.3	Program example with macro command	107
3.18	Metric/Inch Switch.....	108
3.18.1	Functional summary	109
3.18.2	Function command G20/G21.....	109
3.18.3	Notes	109
CHAPTER 4 TOOL NOSE RADIUS COMPENSATION (G41, G42)		110
4.1	Application	110
4.1.1	Overview.....	110
4.1.2	Imaginary tool nose direction.....	111
4.1.3	Compensation value setting.....	114
4.1.4	Command format.....	115
4.1.5	Compensation direction	116
4.1.6	Notes	117
4.1.7	Application	118
4.2	Tool nose radius compensation offset path.....	119
4.2.1	Inner and outer side.....	119
4.2.2	Tool traversing when starting tool	120
4.2.3	Tool traversing in Offset mode	121
4.2.4	Tool traversing in Offset canceling mode	126
4.2.5	Tool interference check.....	127
4.2.6	Commands for canceling compensation vector temporarily	129
4.2.7	Particulars.....	131

Volume II Operation

CHAPTER 1 OPERATION MODE AND DISPLAY INTERFACE	133
1.1 Panel division.....	134
1.1.1 State indication	134
1.1.2 Edit keypad.....	135
1.1.3 Menu display	135
1.1.4 Machine panel	135
1.2 Summary of operation mode	138
1.3 Display interface	139
1.3.1 POS interface	141
1.3.2 PRG interface	143
1.3.3 TOOL OFFSET&WEAR, MACRO, TOOL-LIFE MANAGEMENT interfaces	145
1.3.4 ALARM interface	147
1.3.5 Setting interface.....	147
1.3.6 BIT PARAMETER, DATA PARAMETER, SCREW-PITCH COMP interfaces	150
1.3.7 CNC DIAGNOSIS, PLC STATE, PLC VALUE, TOOL PANEL, VERSION MESSAGE interfaces	152
CHAPTER 2 POWER ON/OFF AND PROTECTION	154
2.1 System power on	154
2.2 System power off	154
2.3 Overtravel protection	154
2.3.1 Hardware overtravel protection.....	154
2.4 Emergency operation.....	156
2.4.1 Reset	156
2.4.2 Emergency stop.....	156
2.4.3 Feed hold.....	156
2.4.4 Power-off	156
CHAPTER 3 MANUAL OPERATION	157
3.1 Coordinate axis move	157
3.1.1 Manual feed.....	157
3.1.2 Manual rapid traverse.....	158
3.1.3 Speed tune	158
3.2 Other manual operations	159
3.2.1 Spindle CCW, CW, stop control	159
3.2.2 Spindle jog	159
3.2.3 Cooling control.....	160
3.2.4 Lubricating control	160
3.2.5 Manual tool change	161
3.2.6 Spindle override.....	161
CHAPTER 4 MPG/STEP OPERATION	162
4.1 Step feed	162
4.2 MPG(handwheel) feed	163
4.2.2 Moving axis and direction selection	164
4.2.3 Other operations	164
4.2.4 Explanation items.....	165

CHAPTER 5 MDI OPERATION	166
5.1 Code words input	166
5.2 Code words execution.....	166
5.3 Parameter setting	167
5.4 Data alteration	164
5.5 Other operations.....	165
CHAPTER 6 PROGRAM EDIT AND MANAGEMENT	166
6.1 Program creation.....	169
6.1.1 Creating a block number.....	169
6.1.2 Inputting a program.....	169
6.1.3 Movement of cursor.....	170
6.1.5 Inserting a character.....	172
6.1.6 Deleting a character.....	172
6.1.7 Altering a character.....	172
6.1.8 Deleting a single block.....	172
6.1.9 Deleting blocks.....	172
6.1.10 Deleting a segment.....	173
6.1.11 Copying a single block.....	173
6.1.12 Copying blocks.....	173
6.1.13 Deleting a segment.....	173
6.1.14 Pasting a single block.....	173
6.2 Deleting program.....	174
6.3 Selecting a program.....	174
6.4 Renaming a program.....	176
6.5 Copy a program.....	176
CHAPTER 7 TOOL OFFSET AND SETTING	178
7.1 Tool positioning setting	178
7.2 Trial tool setting	179
7.3 Tool setting by machine zero return	180
7.4 Setting and altering the offset value.....	182
7.4.1 Offset setting	182
7.4.2 Offset alteration	183
7.4.3 Clearing the offset values	183
7.4.4 Setting and altering the tool wear	183
7.4.5 No.0 tool offset moving workpiece coordinate system	183
CHAPTER 8 AUTO OPERATION	184
8.1 Automatic run.....	184
8.1.1 Selection of the program to be run	184
8.1.2 Start of the automatic run.....	185
8.1.3 Stop of the automatic run.....	185
8.1.4 Automatic run from an arbitrary block.....	186
8.1.5 Adjustment of the feedrate, rapid rate	186
8.1.6 Spindle speed adjustment.....	187

8.2	Running state.....	187
8.2.1	Single block execution.....	187
8.2.2	Dry run.....	188
8.2.3	Machine lock.....	188
8.2.4	MST lock.....	189
8.2.5	Block skip	189
8.3	Other operations	189
CHAPTER 9	ZERO RETURN OPERATION	190
9.1	Program zero return.....	190
9.1.1	Program Zero	190
9.1.2	Program zero return steps.....	190
9.2	Machine Zero return	191
9.2.1	Machine Zero (machine reference point).....	191
9.2.2	Machine Zero return steps.....	191
9.3	Other operations in zero return.....	192
CHAPTER 10	DATA SETTING, BACKUP and RESTORE	193
10.1	Data setting.....	193
10.1.1	Switch setting	193
10.1.2	Graphic display.....	193
10.1.3	Parameter setting	194
10.2	Data recovery and backup.....	199
10.3	Password setting and alteration.....	200
10.3.1	Operation level entry	201
10.3.2	Altering the password	202
10.3.3	Operation grade degradation.....	203
CHAPTER 11	U OPERATION FUNCTION	205
11.1	File catalog window.....	205
11.2	Copying File.....	205
CHAPTER 12	MACHINING EXAMPLES	206
12.1	Programming	207
12.2	Program input	208
12.2.1	View a saved program	208
12.2.2	Creating a new program	209
12.3	Checkout a program.....	210
12.3.1	Graphic setting.....	210
12.3.2	Program check.....	210
12.4	Tool setting and running.....	211

Volume III Connection

CHAPTER 1	INSTALLATION LAYOUT.....	215
1.1	KY980Tc system connection.....	215
1.1.1	KY980Tc back cover interface layout.....	215
1.1.2	Interface explanation	216
1.2	980Tc installation	216
1.2.1	external dimensions	216
1.2.2	Preconditions of the cabinet installation.....	216
1.2.3	Measures against interference	216
CHAPTER 2	DEFINITION & CONNECTION OF INTERFACE SIGNALS.....	218
2.1	Connection to drive unit	218
2.1.1	Drive interface definition	218
2.1.2	Code pulse and direction signals.....	218
2.1.3	Drive unit alarm signal nALM.....	218
2.1.4	Axis enable signal nEN.....	219
2.1.5	Pulse disable signal nSET	219
2.1.6	Zero signal nPC.....	219
2.1.7	Connection to a drive unit.....	221
2.2	Being connected with spindle encoder	222
2.2.1	Spindle encoder interface definition.....	222
2.2.2	Signal explanation	222
2.2.3	Being connected with spindle encoder interface.....	222
2.3	Being connected with MPG (Manual Pulse Generator)	223
2.3.1	MPG interface definition	223
2.3.2	Signal explanation	223
2.4	Spindle interface	224
2.4.1	Spindle interface definition.....	224
2.4.2	Connection to inverter	224
2.5	980Tc being connected with PC	225
2.5.1	Communication interface definition.....	225
2.5.2	Communication interface connection.....	225
2.6	Power interface connection	225
2.7	I/O interface definition.....	225
2.7.1	Input signal	227
2.7.2	Output signal	228
2.8	I/O function and connection	230
2.8.1	Stroke limit and emergency stop.....	230
2.8.2	Tool change control	233
2.8.3	Machine zero return.....	235
2.8.4	Spindle control.....	238
2.8.5	Spindle switching volume control.....	240
2.8.6	Spindle automatic gearing control.....	240

2.8.7	External cycle start and feed hold	242
2.8.8	Cooling control.....	244
2.8.9	Lubricating control	245
2.8.10	Chuck control.....	246
2.8.11	Tailstock control	246
2.8.12	Safety door detection.....	246
2.8.13	Block skip.....	247
2.8.14	CNC macro variables.....	250
2.8.15	Tri-colour indicator	250
2.8.19	Block skip	250
2.9	Commonly use symbol of electricity drawing	253
CHAPTER 3 PARAMETERS		254
3.1	Parameter description (by sequence)	254
3.1.1	Bit parameter	254
3.1.2	Data parameter.....	264
CHAPTER 4 MACHINE DEBUGGING METHODS AND MODES		293
4.1	Emergency stop and limit.....	293
4.2	Drive unit configuration	293
4.3	Gear ratio adjustment	293
4.4	ACC&DEC characteristic adjustment.....	294
4.5	Mechanical (machine) zero adjustment	296
4.6	Spindle adjustment	299
4.6.1	Spindle encoder.....	299
4.6.2	Spindle brake.....	299
4.6.3	Switch volume control of spindle speed.....	300
4.6.4	Analog voltage control of spindle speed	300
4.7	Backlash Offset.....	300
4.8	Tool Post Debugging.....	301
4.9	Step/MPG Adjustment.....	302
4.10	Other adjustment	302
CHAPTER 5 DIAGNOSIS MESSAGE		304
5.1	CNC diagnosis.....	304
5.1.1	I/O status and data diagnosis message.....	304
5.1.2	CNC motion state and data diagnosis message.....	304
5.1.3	Diagnosis keys	305
5.1.4	Others.....	306
5.2	PLC state.....	306
5.2.1	X address (machine→PLC , defined by standard PLC ladders)	306
5.2.2	Y address (PLC→machine, defined by standard PLC ladders)	307
5.2.3	Machine panel	308
5.2.4	F address(CNC→PLC).....	308
5.2.5	G address(PLC→CNC)	315
5.2.6	Address A (message display requiery signal, defined by standard PLC ladders) ..	319

CHAPTER 6	MEMORIZING PITCH ERROR COMPENSATION	320
6.1	Function description.....	320
6.2	Specification	320
6.3	Parameter setting	320
6.3.1	Pitch compensation	320
6.3.2	Pitch error origin	320
6.3.3	Offset interval	321
6.3.4	Offset value	321
6.4	Notes of offset setting	321
6.5	Setting examples of offset parameters.....	321

Appendix

Appendix 1	KY980Tc contour dimension.....	324
Appendix 2	Additional panel dimensions	325

Volume I Programming

Volume I Programming

1.1.1 Product introduction

KY-980Tc can control 5 feed axes(including C axis), 2 analog spindles, 1ms high-speed interpolation, 0.1 μ m control precision, which can obviously improve the machining efficiency, precision and surface quality.



KY980Tc

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

1ms interpolation period, control precision 1 μ m, 0.1 μ m

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 KY100Tc

1.1.2 Technical specification

Controllable axes

Controllable axes: 3 (X, Z, Y)

Link axes : 3

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: $\pm 99999999 \times$ 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 cancel	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	Cancel 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 machine zero point	G80	Rigid tapping state cancel		
G30	2 nd , 3 rd 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

980Tc storage delivery, working environment as follows:

Table 1-2

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

1.1.4 Power supply

KY-980Tc can normally run in the following AC input power supply.

Voltage: within $(0.85 \sim 1.1) \times$ rated AC input voltage (AC 220V);

Frequency: 49Hz ~ 51Hz continuously changing

1.1.5 Guard

KY-980Tc 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.

KY-980Tc 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 GSK980TDb 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 KY-980Tc 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 KY-980Tc 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 GSK980TDb 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.

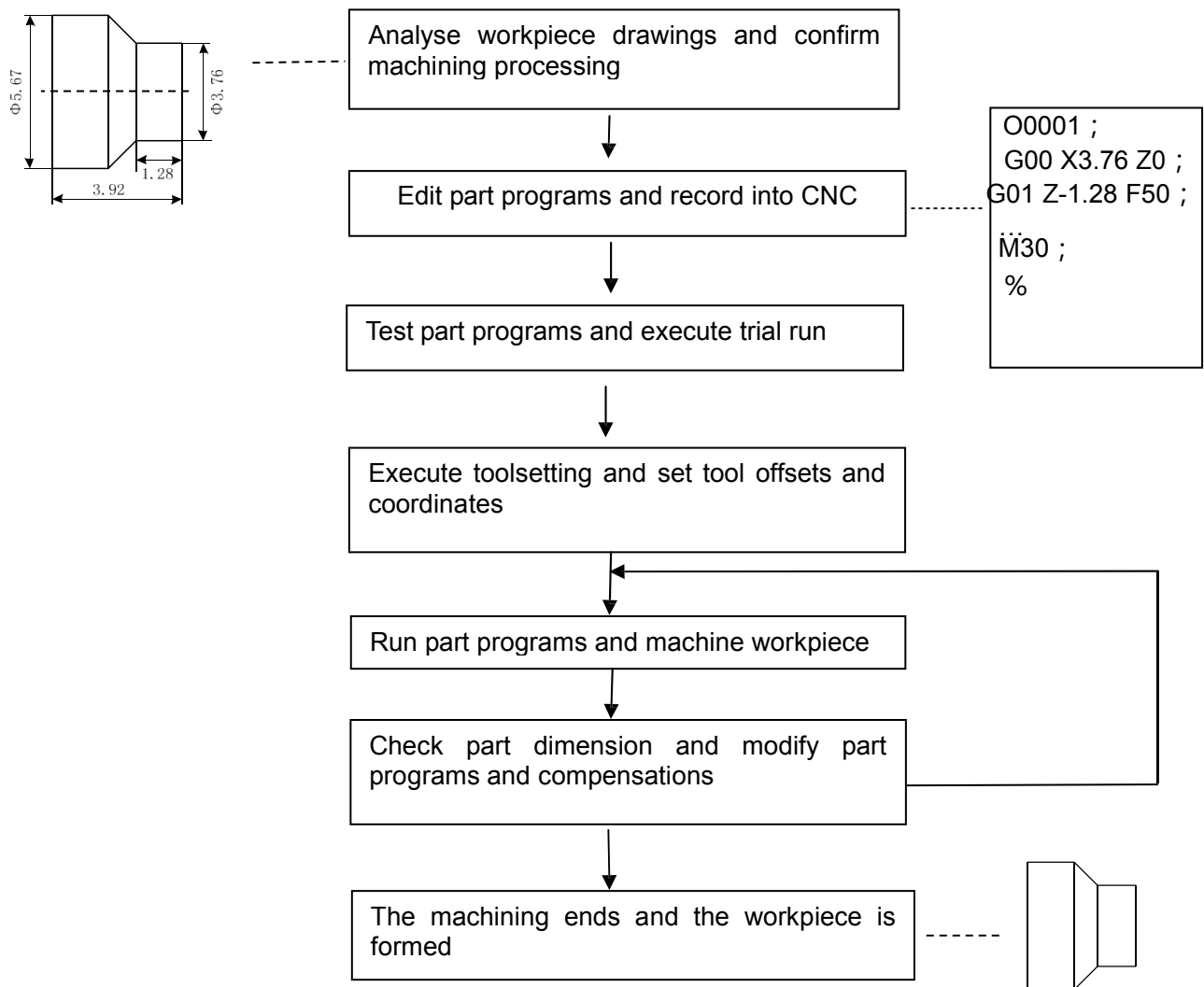


Fig. 1-2

1.3 Programming fundamentals

1.3.1 Coordinates definition

Sketch map of CNC turning machine is as follows:

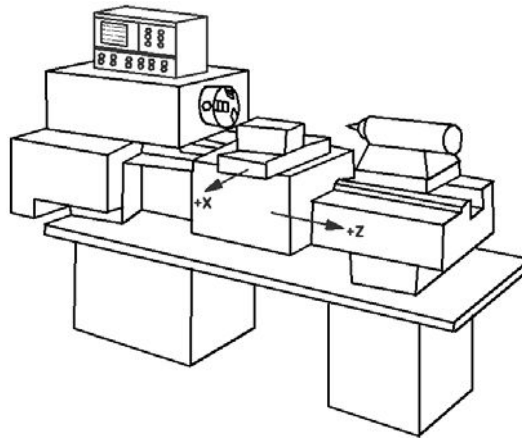


Fig. 1-3

KY-980Tc 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.

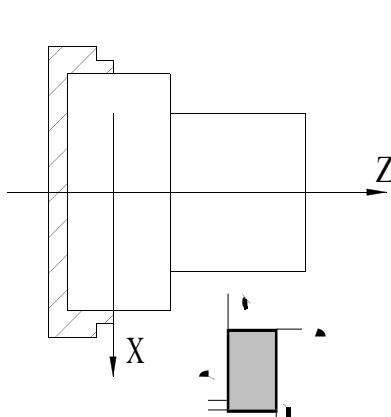


Fig.1-4 Front tool post coordinate system

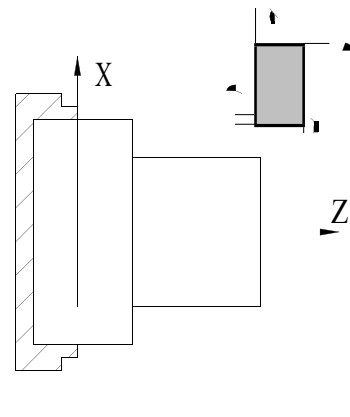


Fig.1-5 Rear tool post coordinate system

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, KY-980Tc 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.

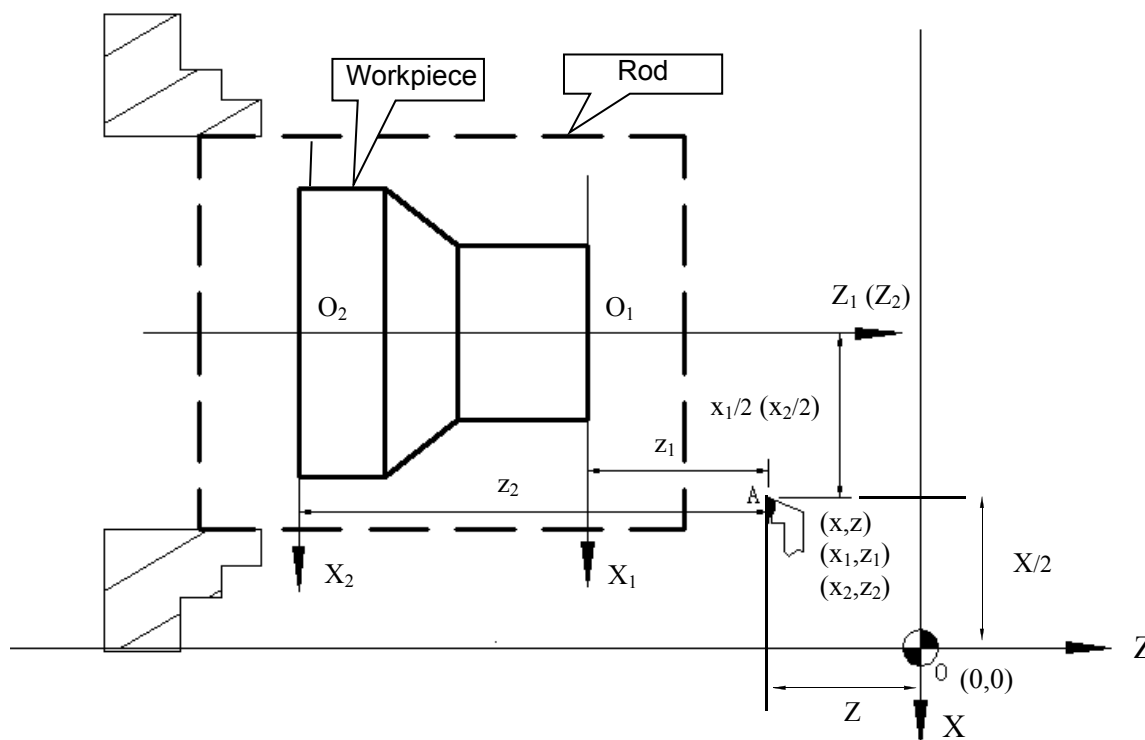


Fig. 1-6

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.

KY-980Tc 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:

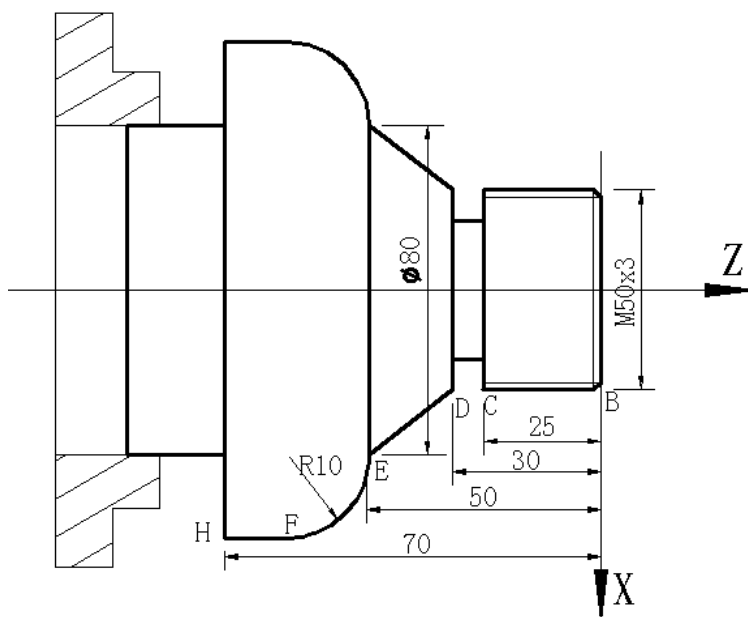


Fig. 1-7

```

...
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;

```

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

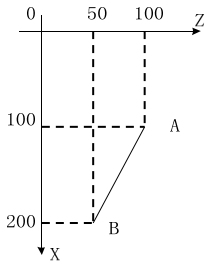


Fig.1-8

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:

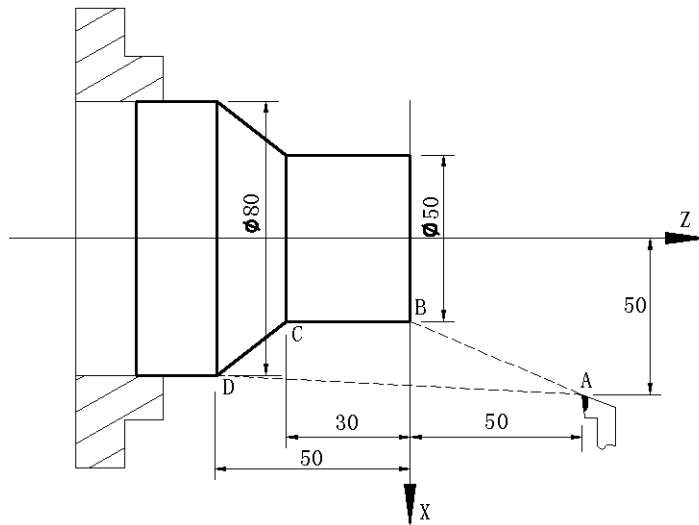


Fig. 1-9

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:

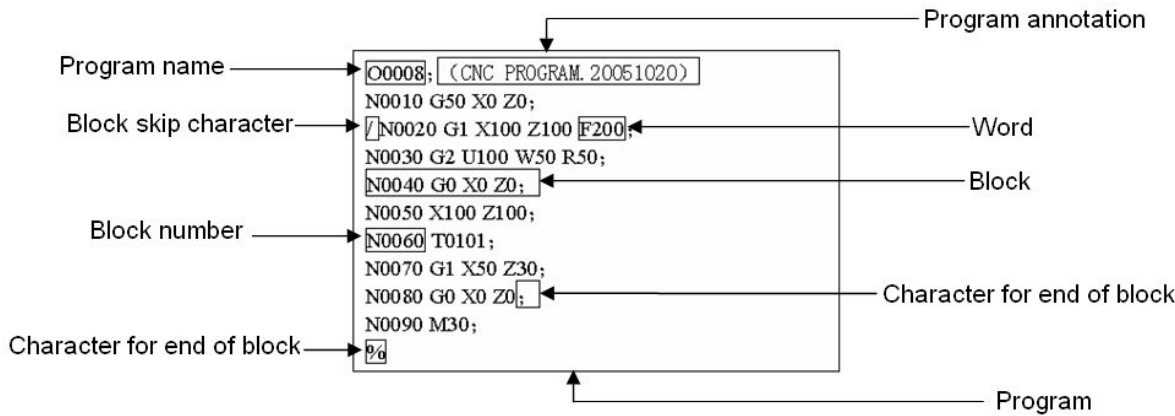
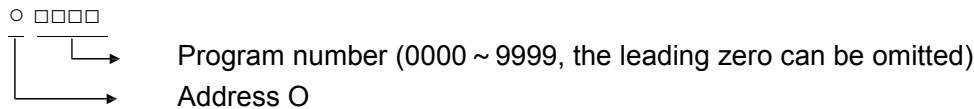


Fig. 1-10 Structure of a program

Program name

There are most 10000 programs stored in GSK980TDb. 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 GSK980TDb are in Table 1-4.

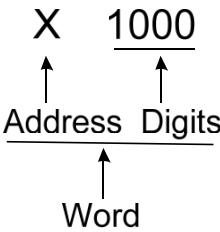


Table 1-4 Word table

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 No.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.

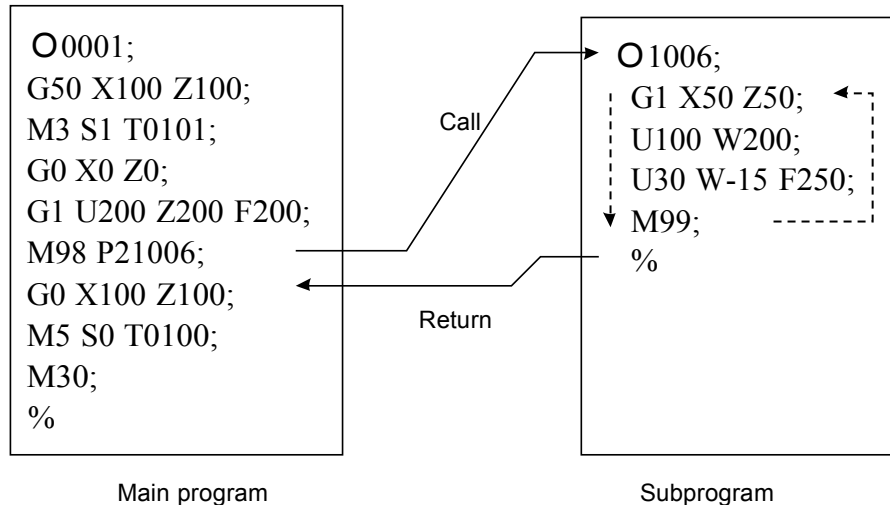









Fig. 1-12 Main program and subprogram

1.5 Program run

1.5.1 Sequence of program run

Running the current open program must be in Auto mode. KY-980Tc 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 ( 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:

- z The program stops running after pressing  or emergent stop button;
- z The program stops running when the system or PLC alarms;
- z 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,  is pressed or external cycle start signal is switched on;
- z 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  is pressed or the external cycle start signal is switched on;
- z The program pauses after pressing  or the external cycle start signal is switched off, and it continuously runs from current position after pressing  or the external cycle start signal is switched on;
- z When Single Block is ON, the program pauses after every block is executed completely, and then it continuously runs from the next block after  is pressed or the external cycle start signal is switched on;
- z Block with “/” in the front of it is not executed when the block skipping switch is ON;
- z The system skips to the target block to run after executing G65;
- z Please see Section Three G Commands about execution sequence of G70~73;
- z 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;
- z 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 KY-980Tc standard PLC program is as follows:

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

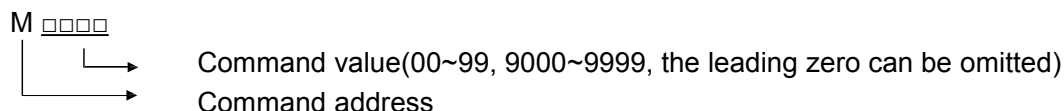
M5, M9, M11, M13, M33 after G codes are executed;

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.



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

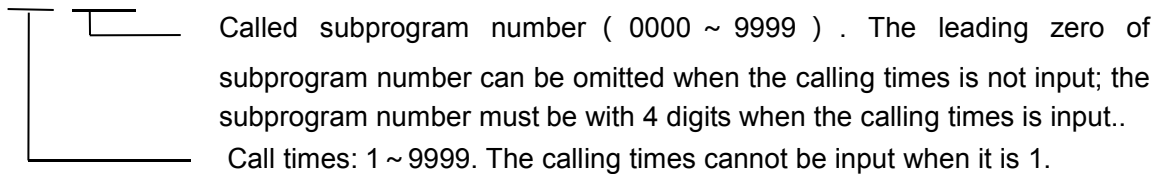
Command function: In Auto mode, after other commands of current block are executed in M30, the automatic run stops, the amount of workpiece is added 1, the tool nose radius compensation is cancelled and the cursor returns to the start of program (whether the cursor return to the start of program or not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

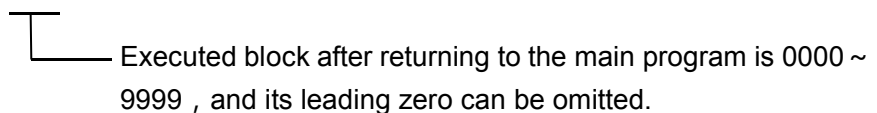
M98 P○○○○□□□□



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○○○○



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.

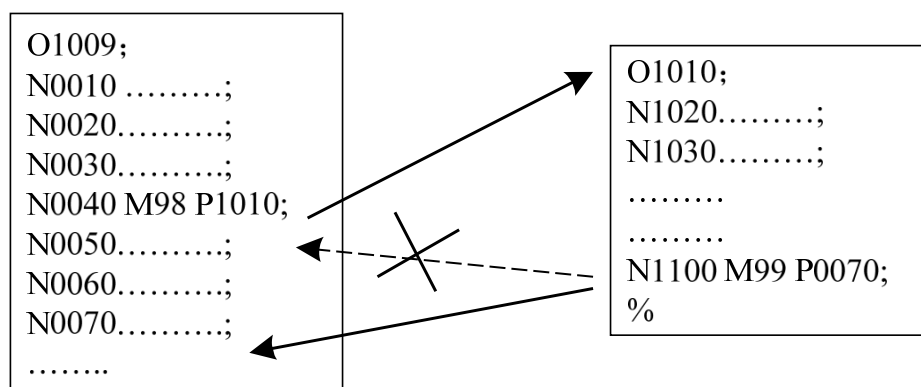


Fig. 2-1 Execution path of calling subprogram (with P in M99)

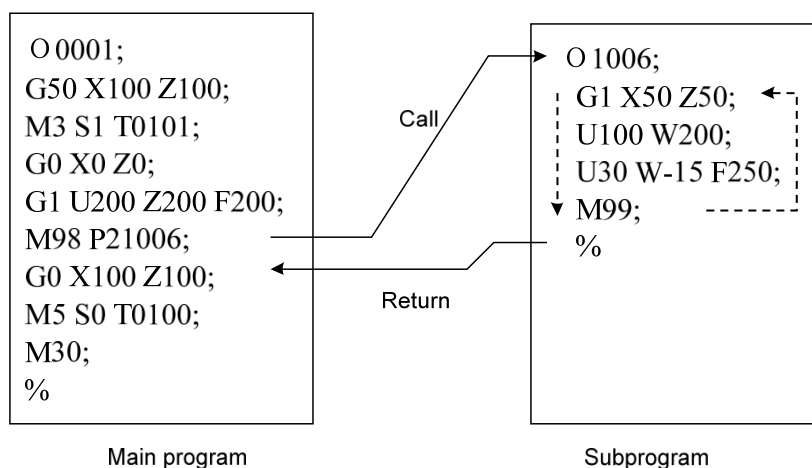


Fig. 2-2 Execution path of calling subprogram (without P in M99)

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

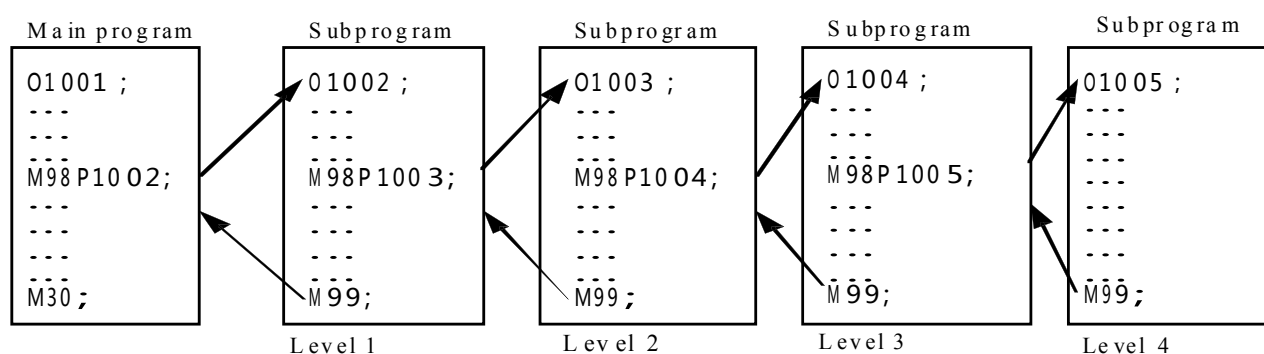


Fig. 2-3 Subprogram nesting

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 KY-980Tc 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 2 nd spindle position control	Functions interlocked and states reserved
*M25	The 2 nd 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 KY-980Tc 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.

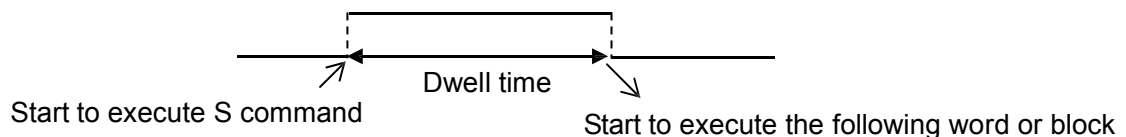
Their executing sequence is defined by PLC when S command and word for moving function are in the same block. Please refer to *User Manual* from machine manufacturer.

When spindle speed is controlled by switching value, KY-980Tc 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 KY-980Tc 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 versa, 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.

$$\text{Surface speed} = \text{spindle speed} \times |X| \times \pi \div 1000 \quad (\text{m/min})$$

Spindle speed: r/min

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

$\pi \approx 3.14$

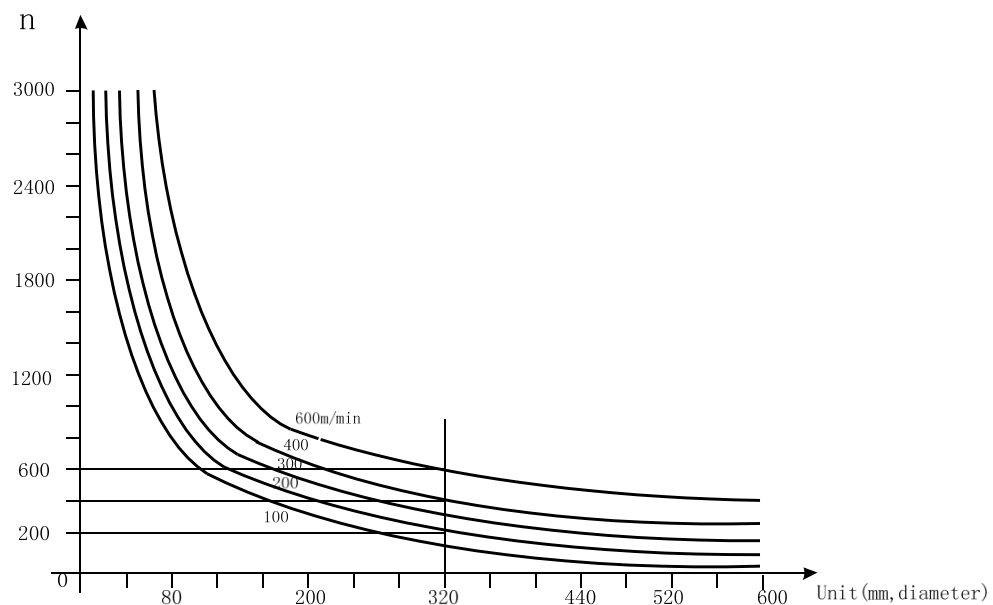


Fig. 2-4

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:

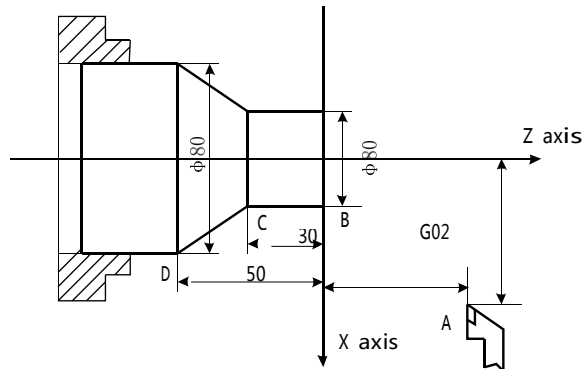


Fig. 2-5

```

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 KY-980Tc standard PLC ladder.

The spindle actual speed specified by KY-980Tc 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**.

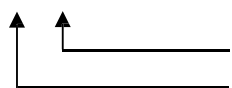
2.3 Tool function

2.3.1 Tool control

T functions of KY-980Tc: 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



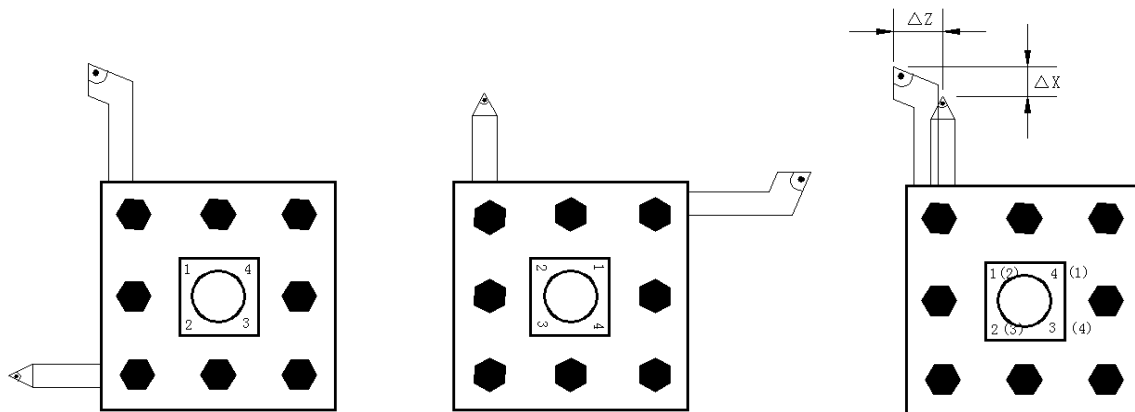
Tool offset number (00-32 , the leading zero cannot be omitted)

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 :

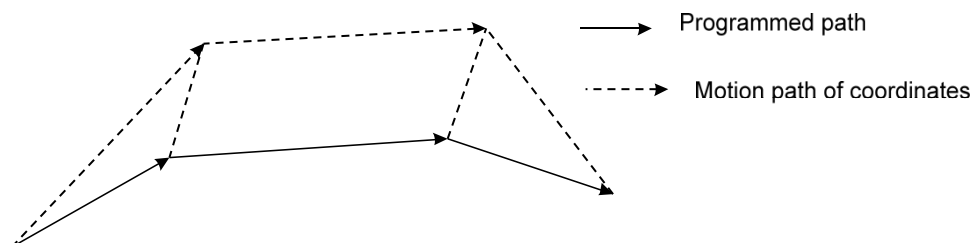


Fig. 2-5 Creation, execution and cancellation of tool length compensation

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

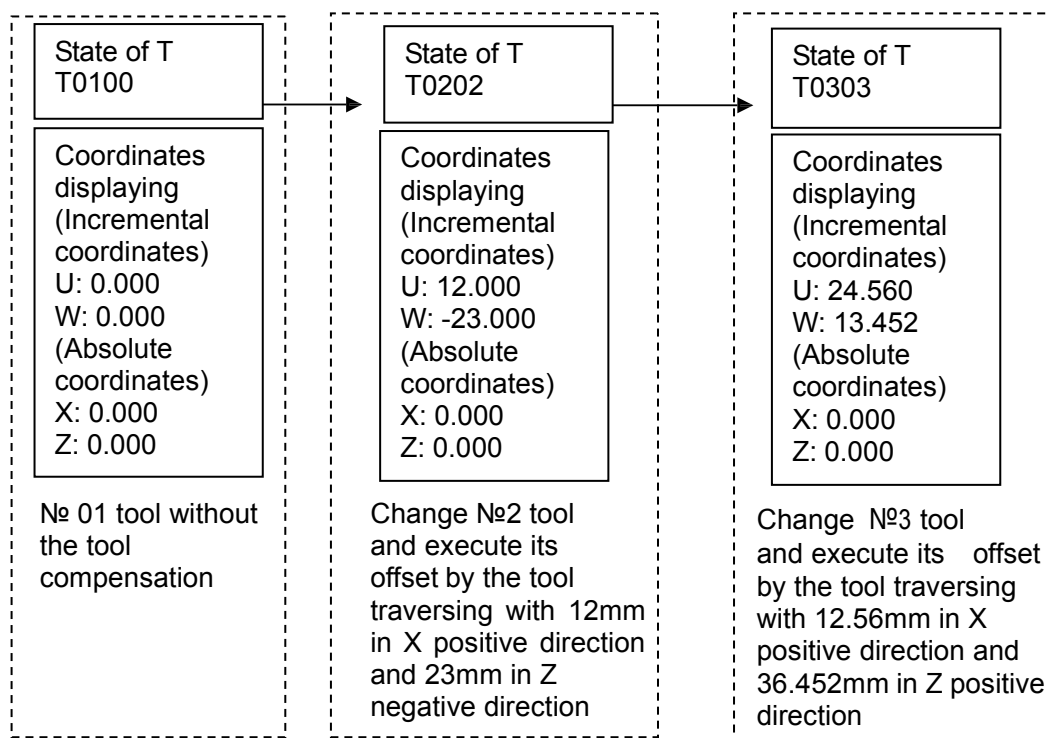


Fig. 2-6 Tool traversing mode

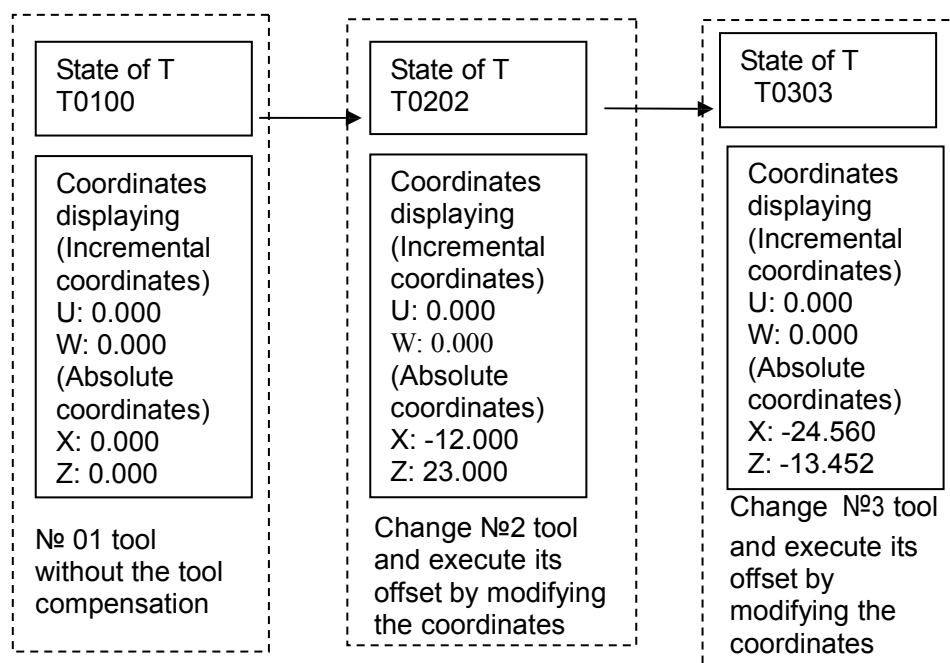


Fig. 2-7 Modifying the coordinates mode

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. GSK980TDb 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.**

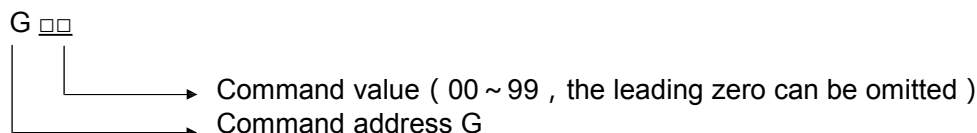
When the system is employed with line-up tool post, No.084 should be set to 1 and different tool number is executed by different tool offset as T0101, T0102, T0103.

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 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 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: $\pm 99999999 \times$ 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:

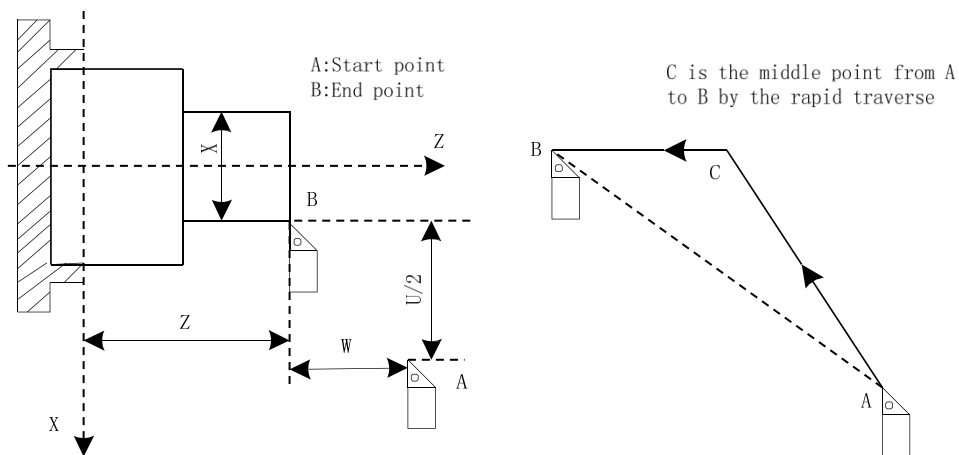


Fig. 3-1

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

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

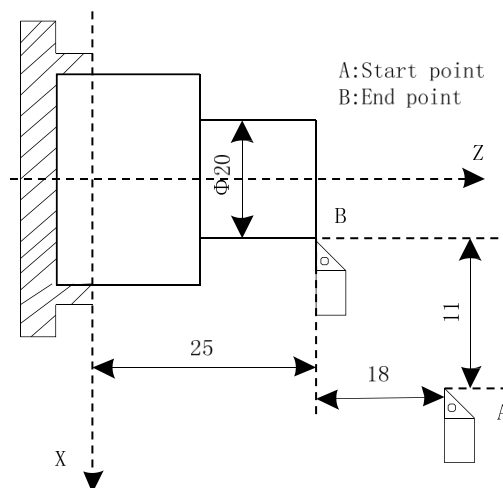


Fig. 3-2

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:

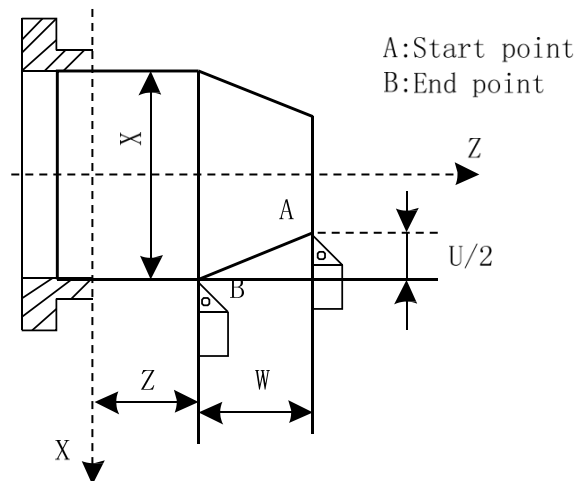


Fig. 3-3

Example: Cutting path from $\Phi 40$ to $\Phi 60$ as Fig.3-4:

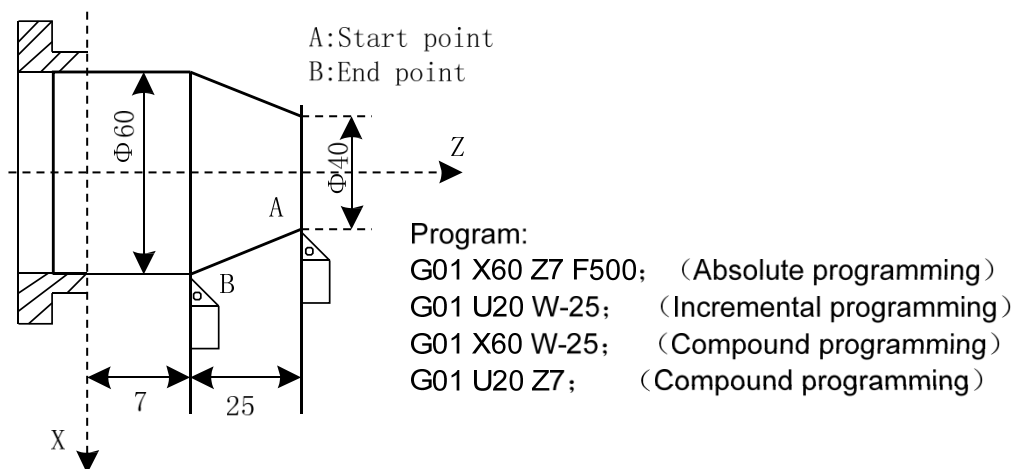


Fig. 3-4

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:

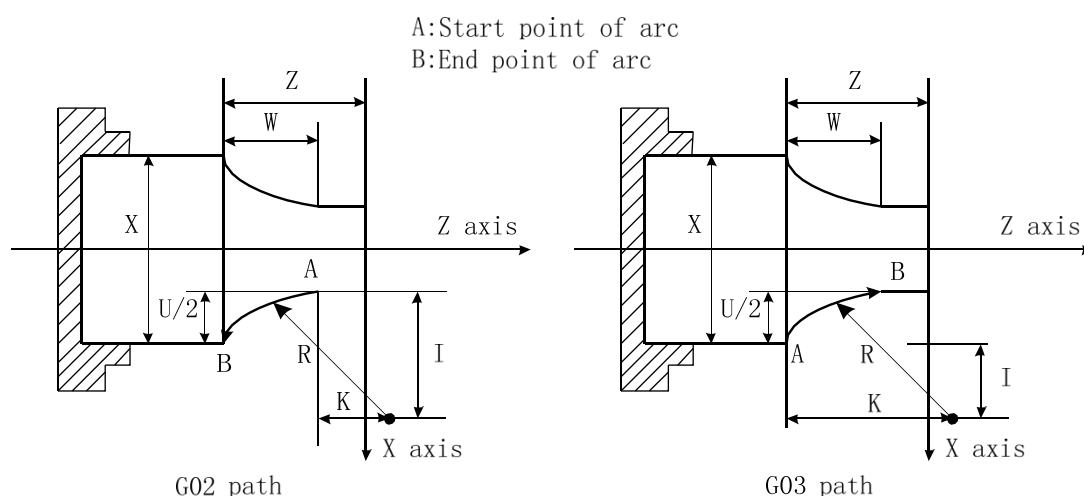


Fig. 3-5 G02 and G03 path

Command specification:

G02, G03 are modal,

R is arc radius, range: $\pm 99999999 \times$ 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.

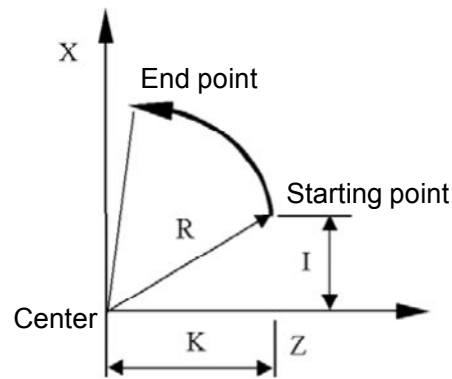


Fig. 3-6

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

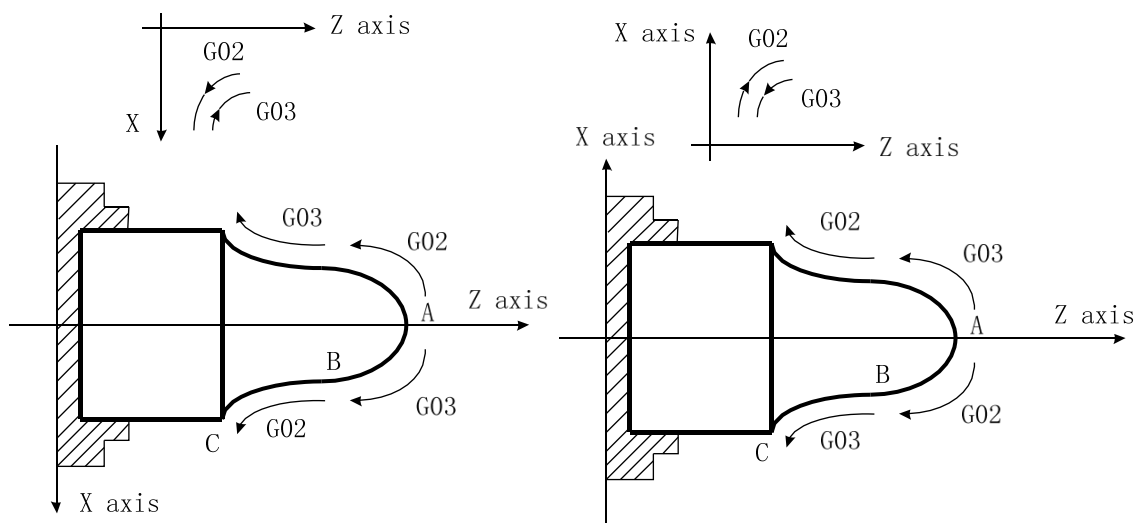
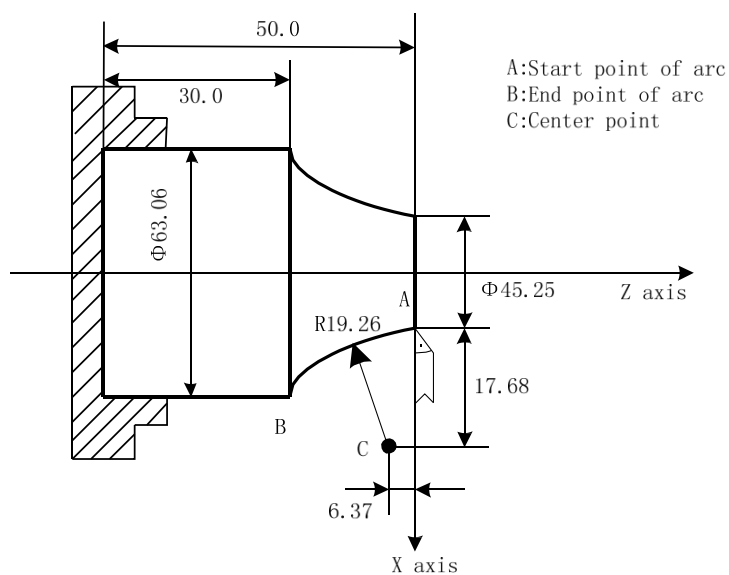


Fig. 3-7

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 $0(0^\circ)$ 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 $\Phi 45.25$ to $\Phi 63.06$ 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:

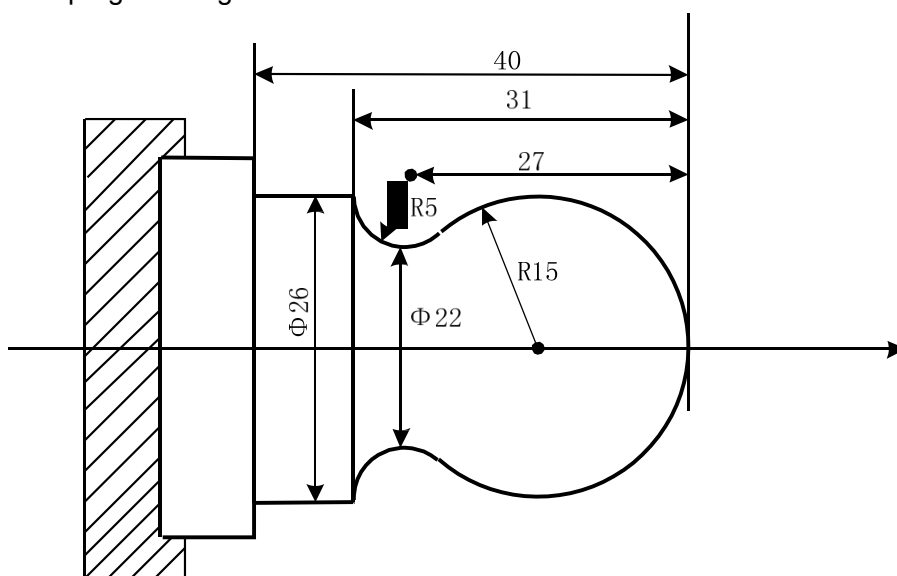


Fig. 3-9 Circular programming example

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 $\phi 26$)
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. GSK980TDb uses the linear and circular chamfering functions.

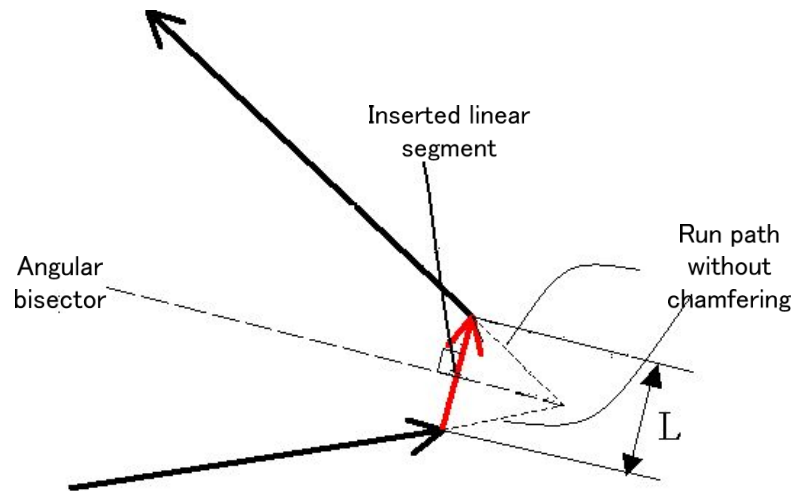
3.6.1 Linear chamfering

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

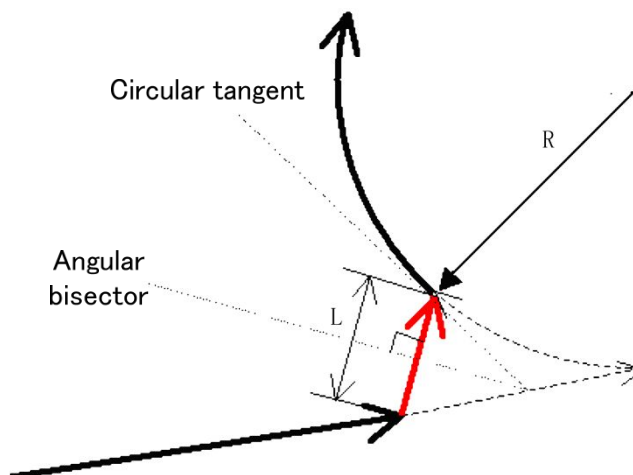
**B. Linear to circular**

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.

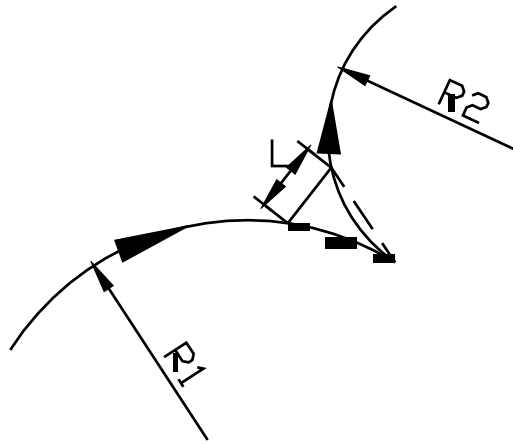
**C. Circular to circular**

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

Or

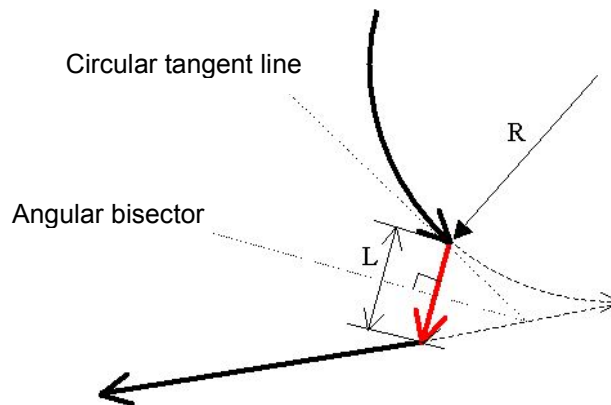
G02/G03 X(U)_ Z(W)_ I_ K_ L_;
G02/G03 X(U)_ Z(W)_ I_ K_;

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.



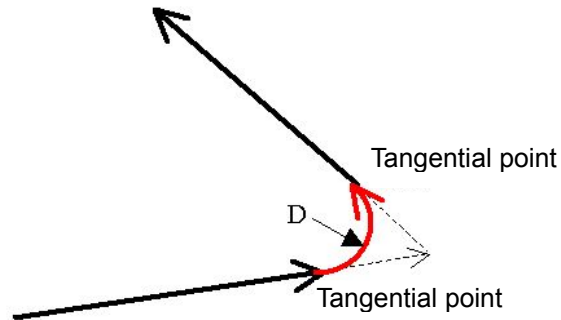
3.6.2 Circular chamfering

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

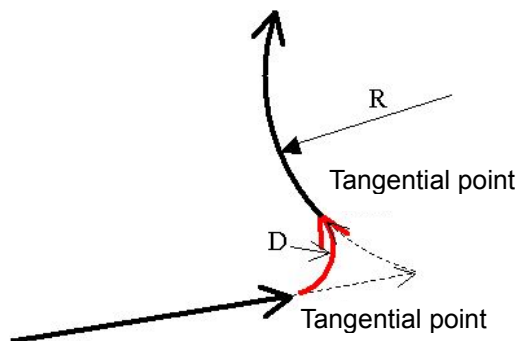
Command format: G01 X(U)_ Z(W)_ D_;
 G01 X(U)_ Z(W)_;

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**

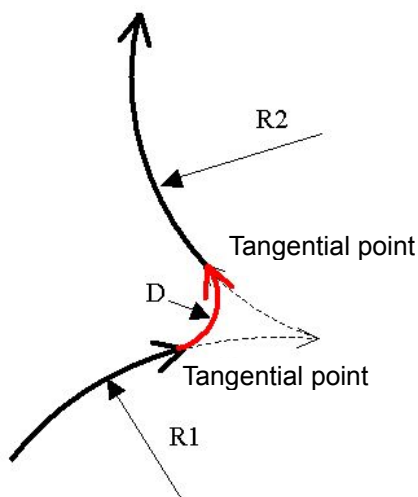
Command format: G01 X(U)_ Z(W)_ D_;
 G02/G03 X(U)_ Z(W)_ R_;
 or
 G01 X(U)_ Z(W)_ D_;
 G02/G03 X(U)_ Z(W)_ I_ K_;

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_;

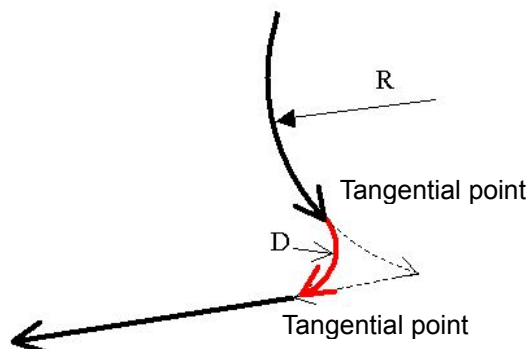
G01 X(U)_ Z(W)_;

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:

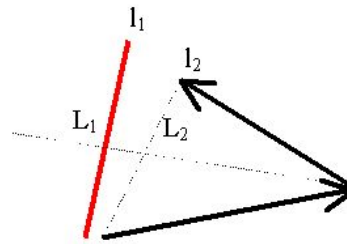
1) Linear chamfering

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



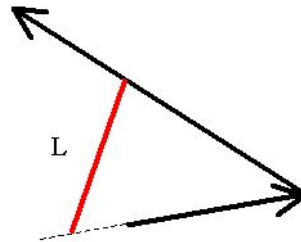
B. CNC alarms when the chamfering linear is too long.

L1 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 calculated chamfering linear is not in the interpolation linear (in the extension line of the interpolation linear).



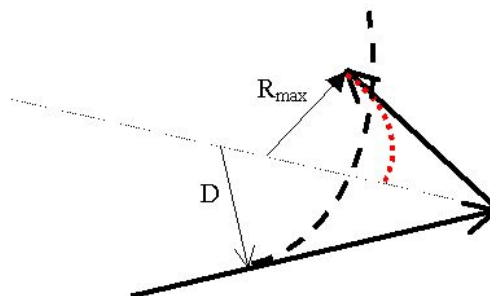
2) Circular chamfering

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.



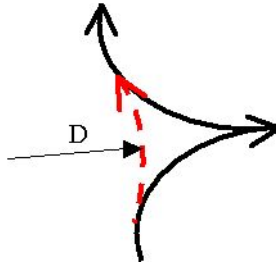
C. The circular chamfering function is invalid when the linear and the circular, or the circular and the linear are tangential.



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:

- z The system exactly stop a block when P, X, U are not input
- z 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.

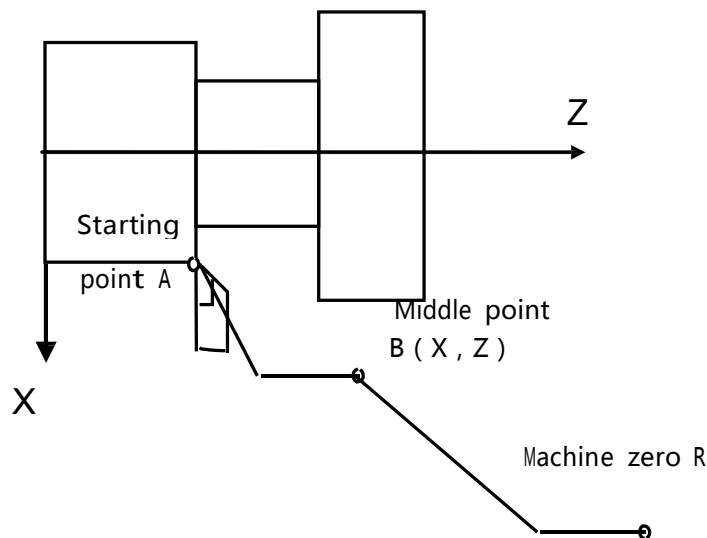


Fig.3-12

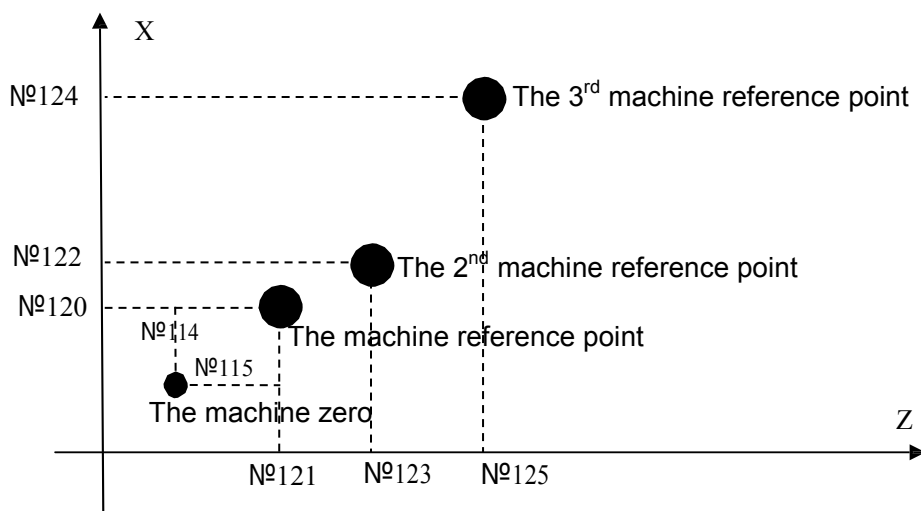
- Note 1:** Machine zero returns in Jog mode and in G28 are the same and their deceleration signals and signals per rev must be detected;
- Note 2:** X and Z move at the respectively rapid traverse speed from A to B and from B to R, and so the path is not always a straight line;
- Note 3:** The system 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 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.

KY-980Tc has machine 2nd, 3rd reference point functions. Use separately No.122 ~ No.125 to set X, Z machine coordinates of the machine 2nd, 3rd reference point.

The relationship between the machine zero, machine reference point, machine 2nd, 3rd 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 2nd, 3rd 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 P _n X/U ____	X returns to the machine nth reference point, Z axis retains
G30 P _n 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 P _n X/U ____ Z/W ____	X and Z return to the machine nth reference point simultaneously

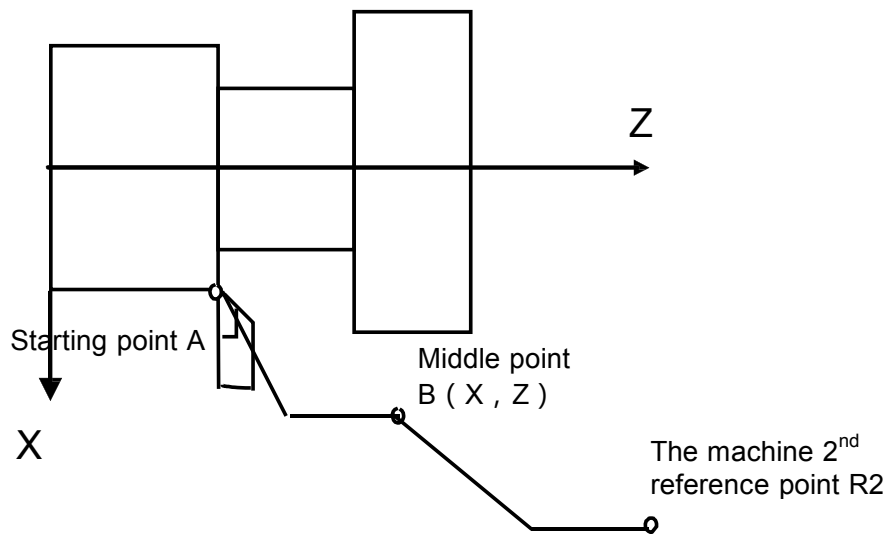
Note 1: n in the above table is 2, 3 or 4;

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

Command operations: (taking example of returning to machine 2nd 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 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 reference point.
- Note 4:** Must not execute G30 (machine 2nd, 3rd 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 and the machine 3rd 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:

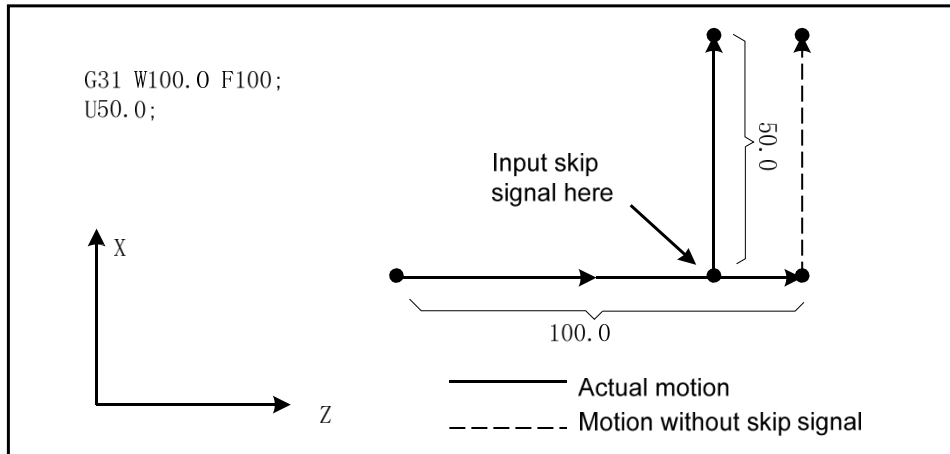


Fig. 3-13

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

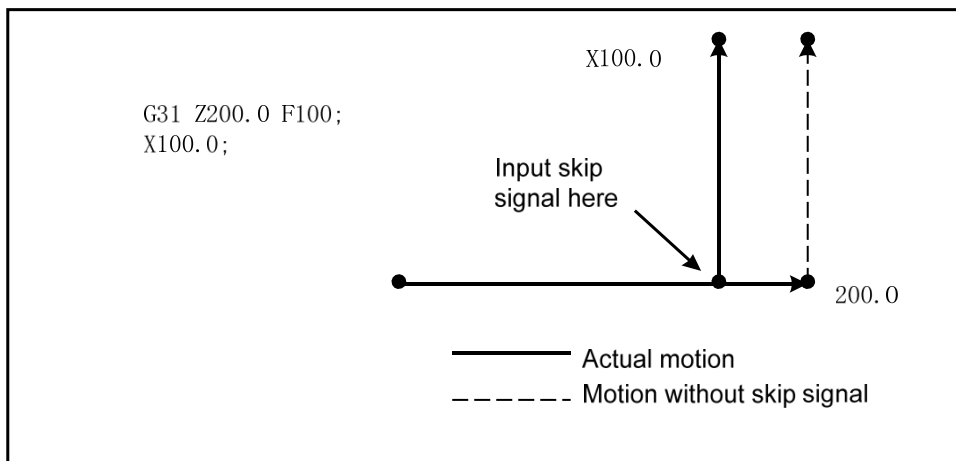


Fig. 3-14

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

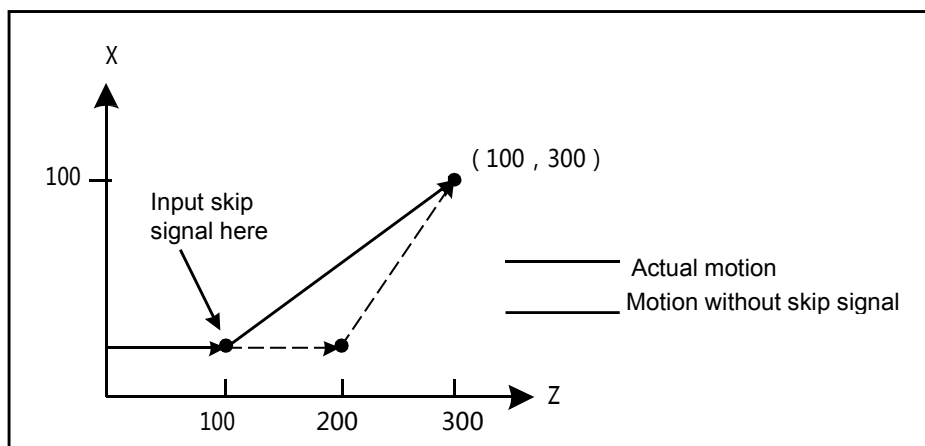


Fig. 3-15

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 ~ #5009 separately corresponds to X, Z, Y) 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:

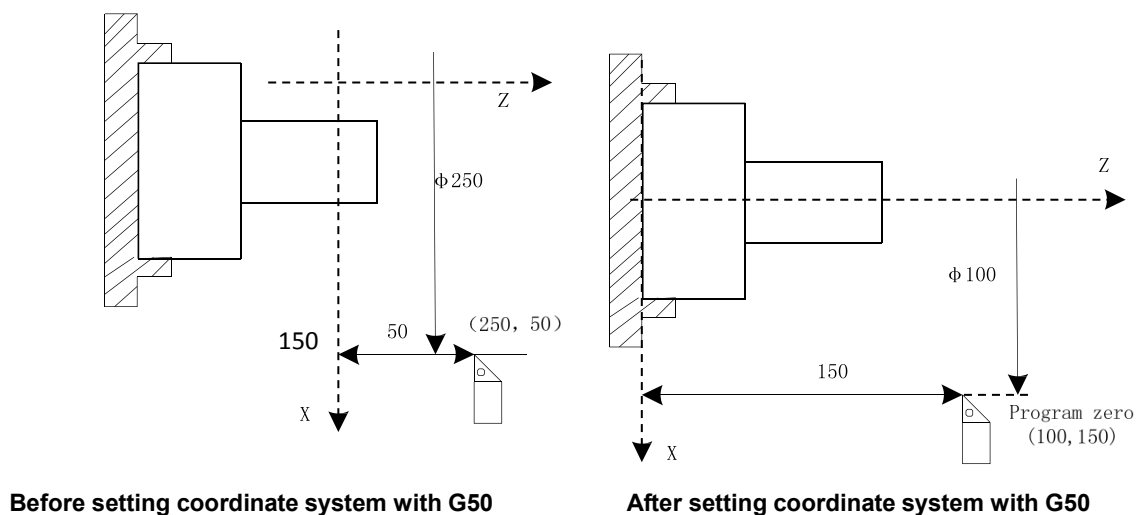


Fig.3-16

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 by G54~G59 instruction.

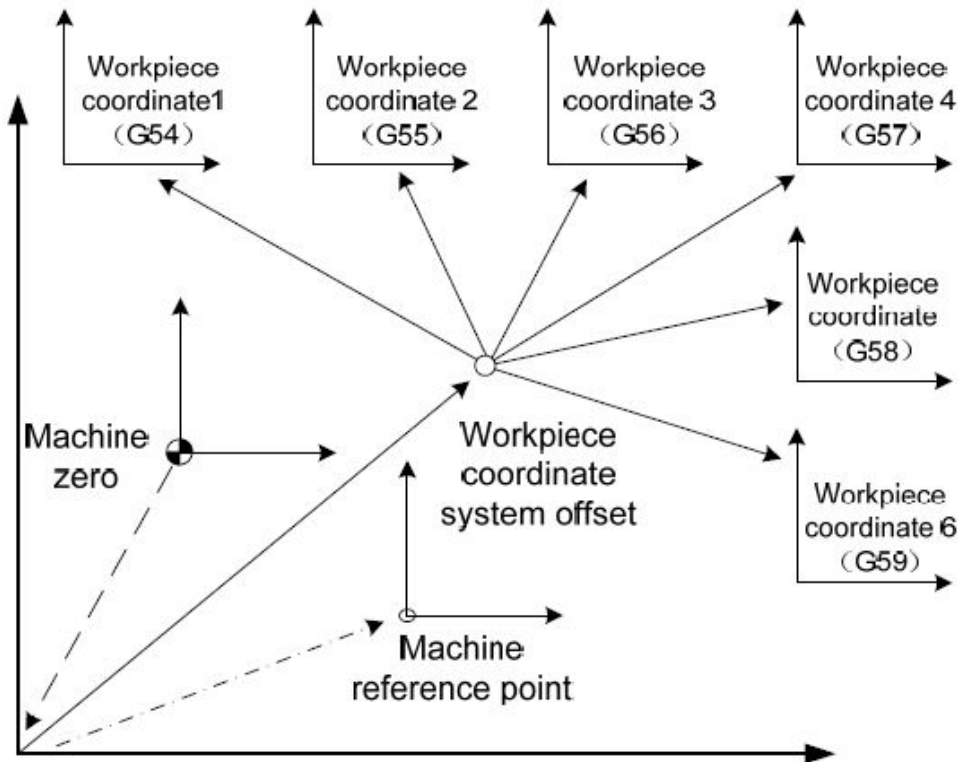
G54 -----	Workpiece coordinate system 1
G55 -----	Workpiece coordinate system 2
G56 -----	Workpiece coordinate system 3
G57 -----	Workpiece coordinate system 4
G58 -----	Workpiece coordinate system 5
G59 -----	Workpiece coordinate system 6

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 G56 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, $R \leq |U/2|$; 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.

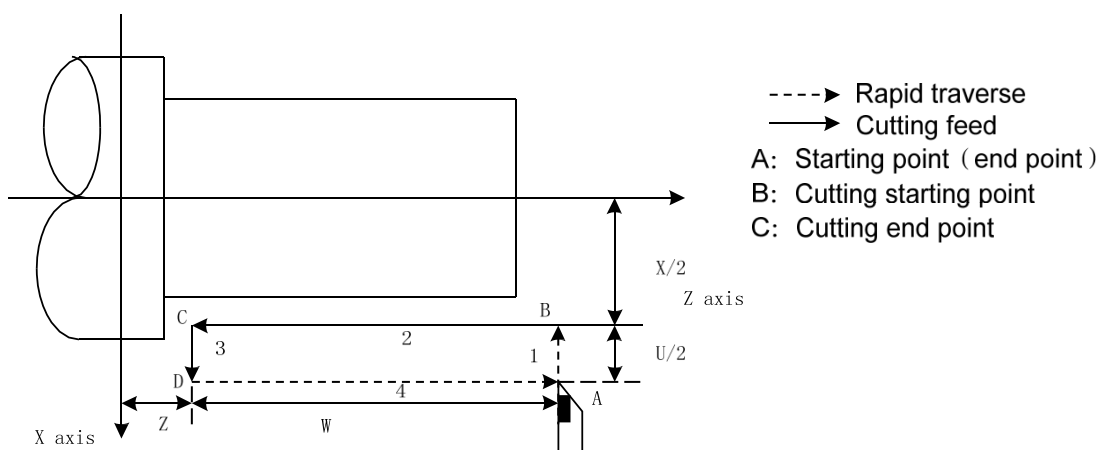


Fig. 3-17

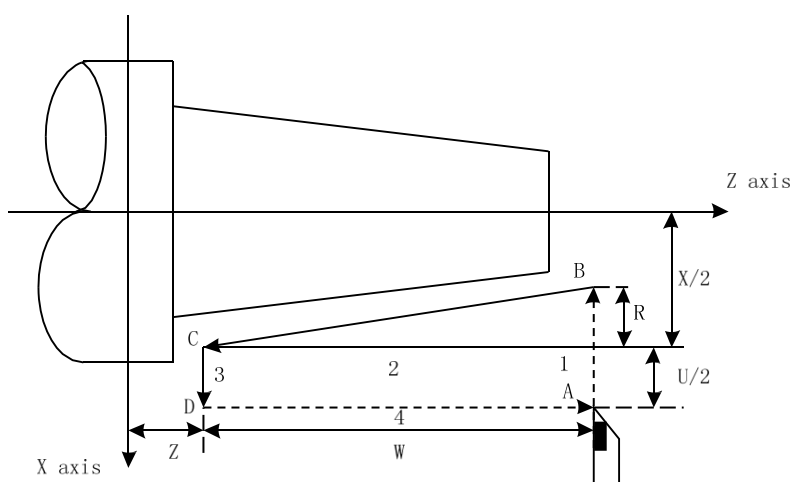
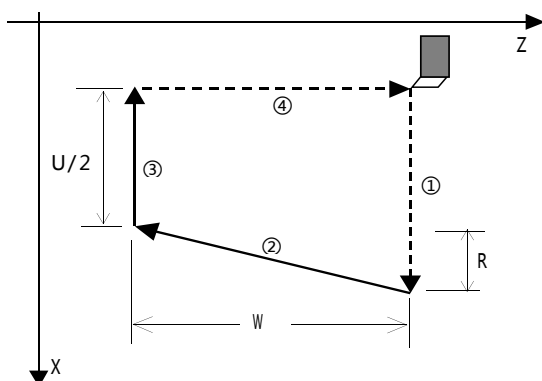


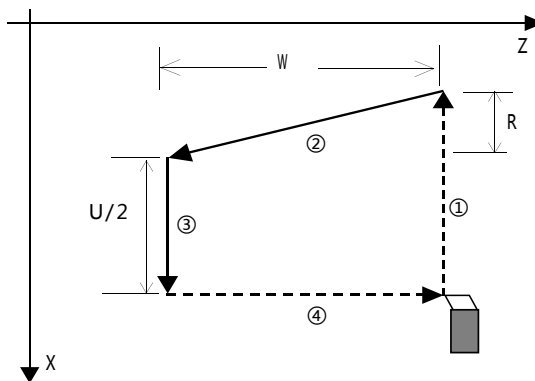
Fig. 3-18

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) $U < 0, W < 0, R < 0$



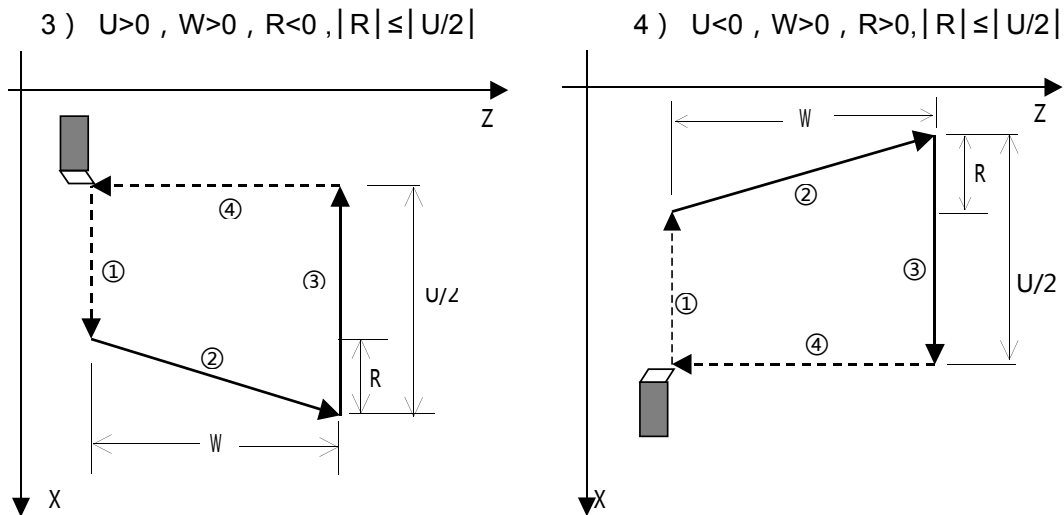


Fig. 3-19

Example: Fig. 3-20 rod $\Phi 125 \times 110$

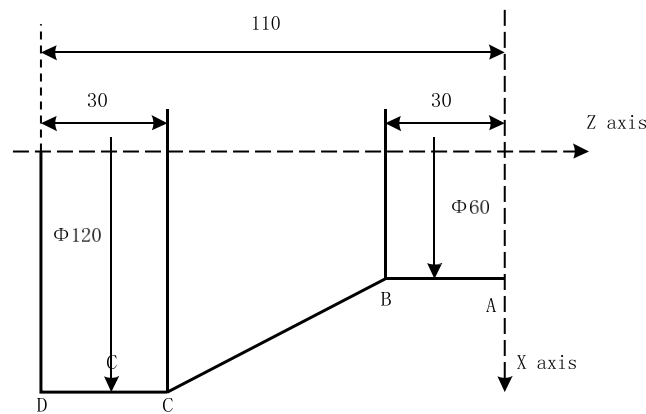


Fig. 3-20

```

Program : O0002;
M3 S300 G0 X130 Z3;
G90 X120 Z-110 F200;      (A→D, cut Φ120)
X110 Z-30;
X100;
X90;
X80;
X70;
X60;
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
M30;

```

(A→B , 6 times cutting cycle Φ60, increment of 10mm)

(B→C , 4 times taper cutting)

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, $R, |R| \leq |W|$.

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.

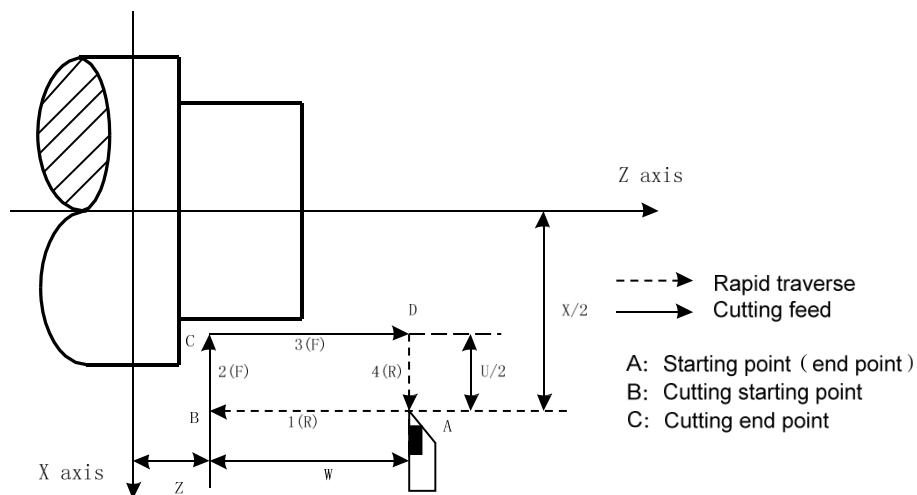


Fig. 3-21

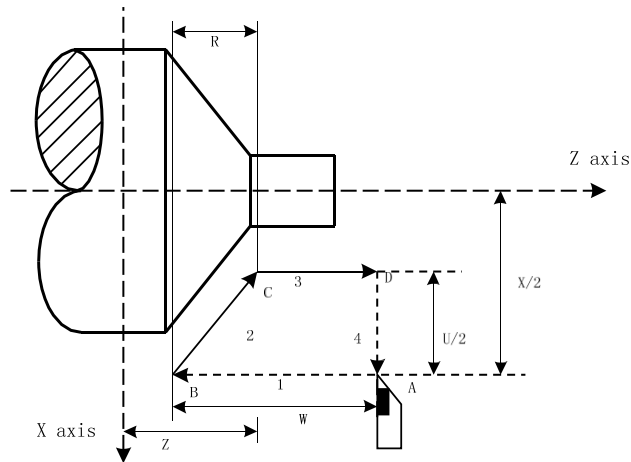
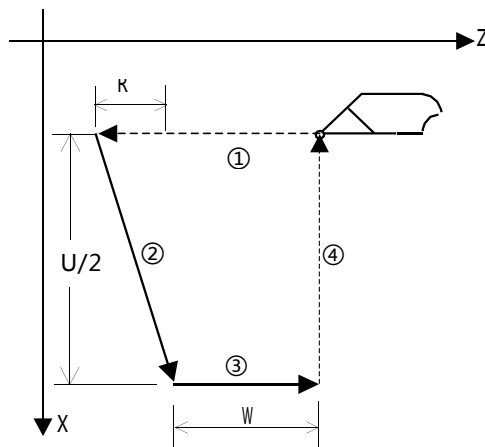


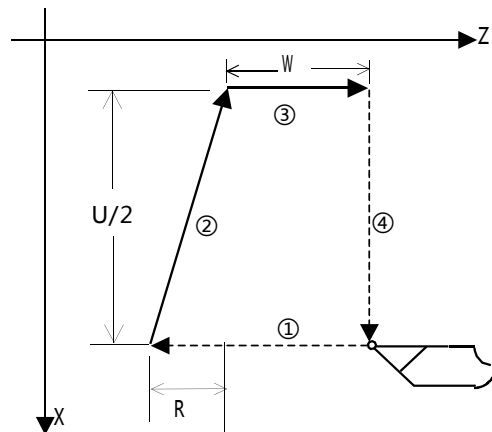
Fig. 3-22

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

1) $U > 0$ $W < 0$ $R < 0$



2) $U < 0$ $W < 0$ $R < 0$



(3) $U > 0$ $W > 0$ $R < 0$ ($|R| \leq |W|$)

4) $U < 0$ $W > 0$ $R < 0$ ($|R| \leq |W|$)

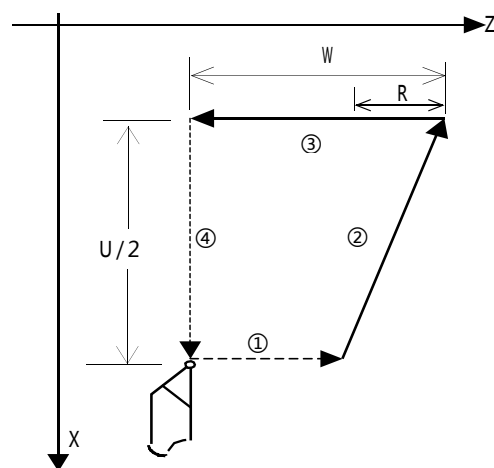
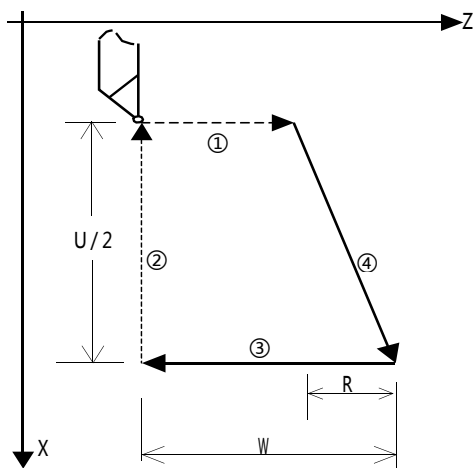


Fig. 3-23

Example: Fig. 3-24, rod $\Phi 125 \times 112$

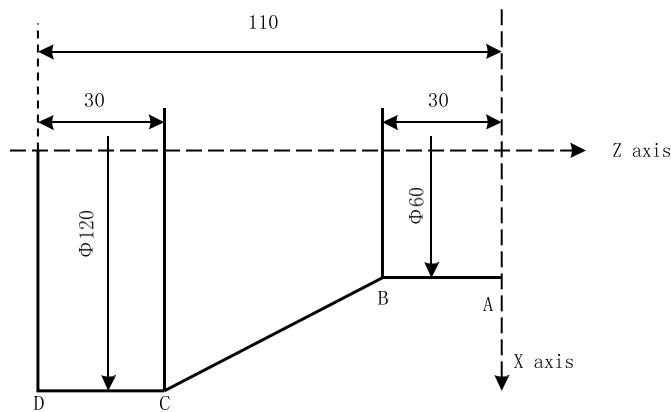


Fig. 3-24

Program: O0003;

G00 X130 Z5 M3 S1;

G94 X0 Z0 F200

X120 Z-110 F300;

G00 X120 Z0

G94 X108 Z-30 R-10

X96 R-20

X84 R-30

X72 R-40

X60 R-50;

M30;

End face cutting
(Outer cutting $\Phi 120$)

(C→B→A , cutting $\Phi 60$)

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 ;
. . . .
N(nf) . . . . . ;

```

(3)

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($\Delta u, \Delta w$) 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 $\Delta d \times 1000$ (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 $e \times 1000$ (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 $\pm 99999.999 \times$ 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 $\Delta u = 0$ when U(Δu) is not input, i.e. there is no finishing allowance in X direction for roughing cycle.

Δw : Z finishing allowance is $\pm 99999.999 \times$ 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 $\Delta w = 0$ 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 tool 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 ($\Delta d + e$) 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 ($\Delta d + e$) again, execute ③; after executing the tool infeed ($\Delta d + e$) 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 block following the block after G71 cycle is ended.

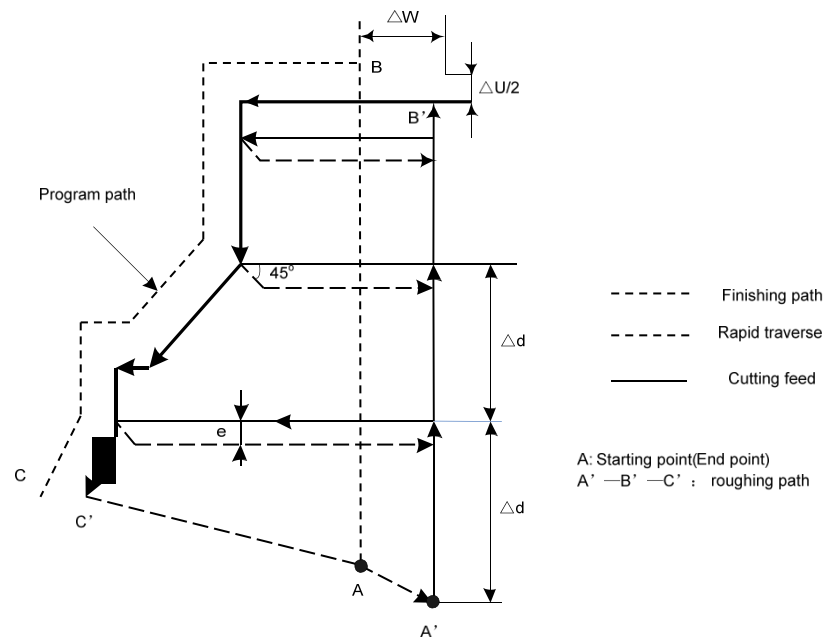


Fig. 3-25 G71 cycle path

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.

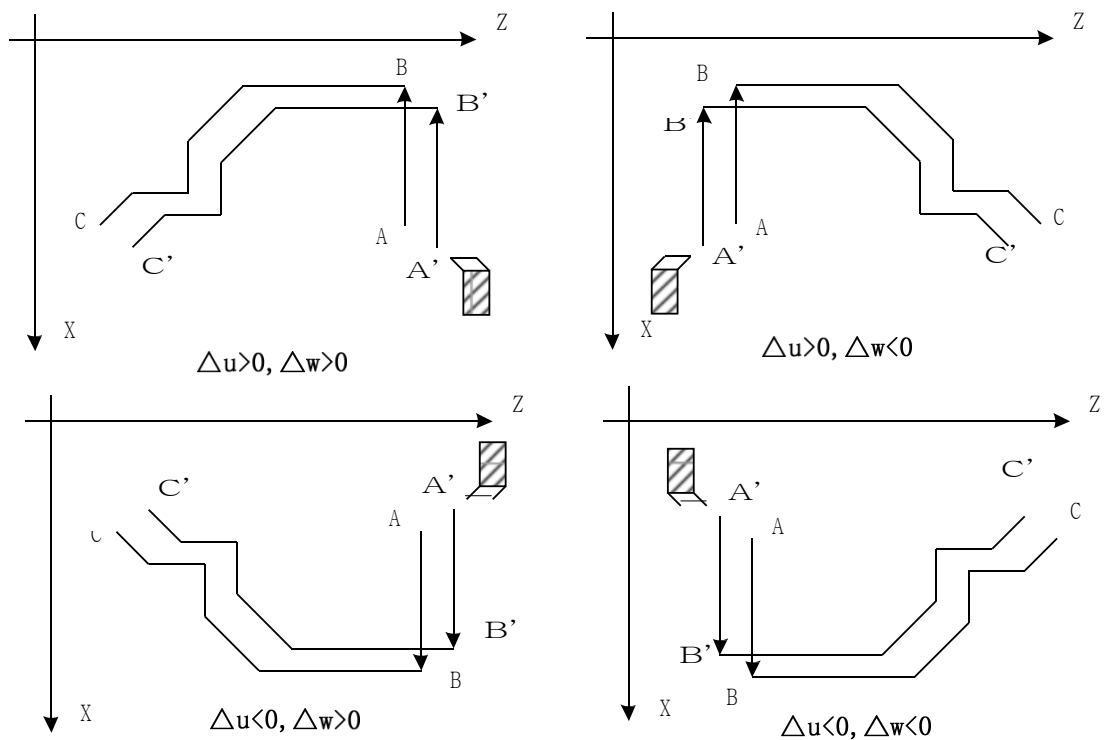


Fig.3-26

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;
- $\Delta d, \Delta u$ 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)

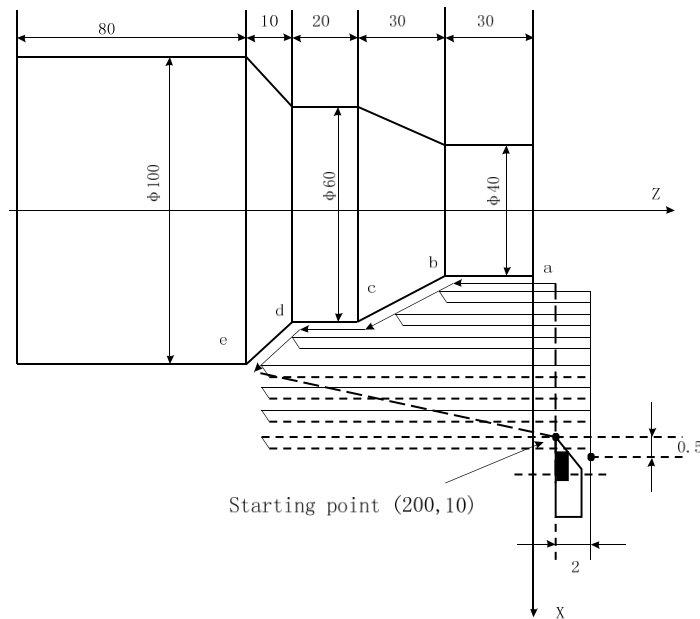


Fig. 3-27

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)
G70 P80 Q120;	(a---e blocks for finishing path)
M30;	(End of block)

a→b→c→d→e blocks for finishing path

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 $W(\Delta d)$ 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: $\pm 99999999 \times \text{least input increment}$ (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: $\pm 99999999 \times \text{least input increment}$ (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 $ns \sim nf$. 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

- 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 G72 feedrate F in G1, and its direction of tool infeed is that of $A \rightarrow B$ point;
- ③ X executes the cutting feeds to the roughing path, and its direction is the same that of X coordinate $B \rightarrow C$ point;
- ④ X, Z execute the tool retraction e (45° straight line) at feedrate, the directions of tool retraction is opposite to that of tool infeed ;
- ⑤ X rapidly retracts at rapid traverse speed to the position which is the same that of Z coordinate;
- ⑥ After Z tool infeed ($\Delta d + e$) 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 ($\Delta d + e$) again, ③ is executed; after the tool infeed ($\Delta d + e$) is executed again, the end point of tool traversing reaches B' point or exceeds the straight line between $A' \rightarrow B'$ point and Z 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 Rapidly traverse to A from C' point and the program jumps to the next block following nf block after G71 cycle is completed.

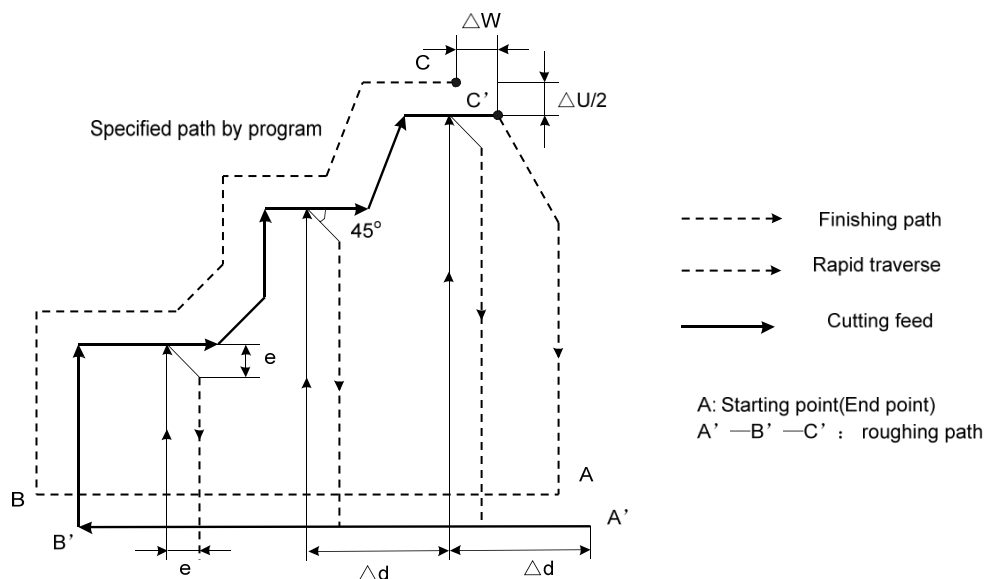


Fig. 3-28

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;
- Δd , Δw 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.

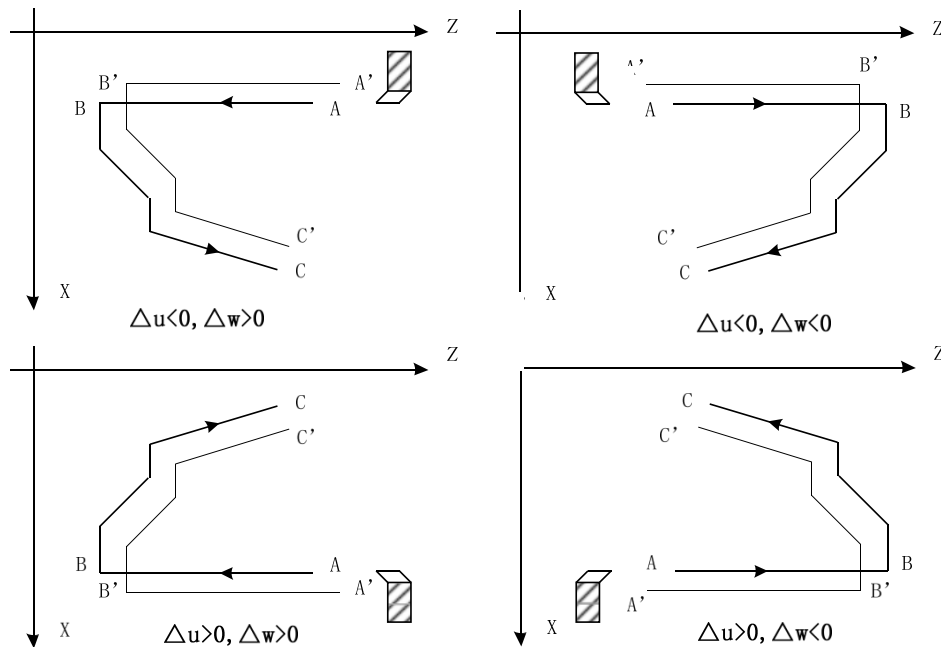


Fig.3-29

Example : Fig. 3-30

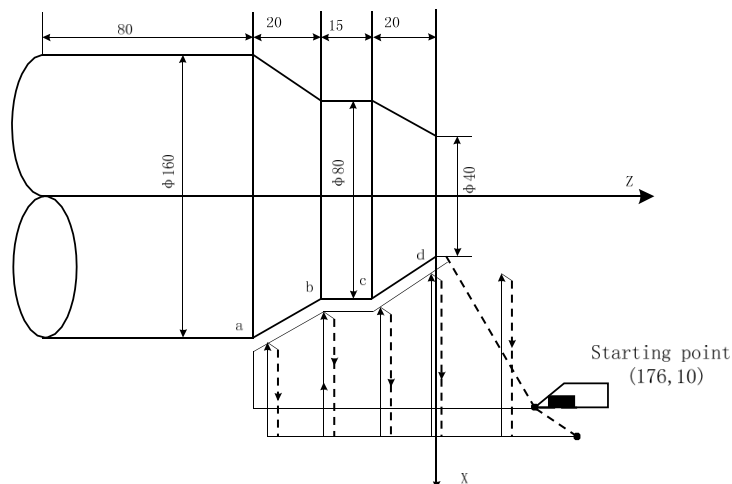


Fig.3-30

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)	}	Blocks for finishing path
G01 X160 F120;	(Infeed to a point)		
X80 W20;	(Machining a—b)		
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; (3)
 S;
 ;
 .
 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 as the 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 as 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 ($\Delta i \times 2 + \Delta u$, $\Delta w + \Delta k$) (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 $\pm 99999.999 \times$ 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 $|\Delta i|$ in roughing, and X cutting direction is opposite to the sign of Δi : $\Delta i > 0$, 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 $U(\Delta i)$ is not input.

Δk : It is Z tool retraction clearance in roughing, and its range is $\pm 99999.999 \times$ 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 $|\Delta k|$ in roughing, and Z cutting direction is opposite to the sign of Δk : $\Delta i > 0$, 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 $W(\Delta k)$ 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 $\pm 99999.999 \times$ 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 A₁ compared to A. $\Delta u > 0$, it is the offset of the last X positive roughing path compared to finishing path. The system defaults $\Delta u = 0$ when $U(\Delta u)$ is not input, i.e. there is no X finishing allowance for roughing cycle.

Δw : It is Z finishing allowance and its range is $\pm 99999.999 \times$ 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 A₁ compared to A. $\Delta w > 0$, it is the offset of the last X positive roughing path compared to finishing path. The system defaults $\Delta w = 0$ when $W(\Delta w)$ 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→A₁: Rapid traverse;

② First roughing A₁→B₁→C₁ :

A₁→B₁: Rapid traverse speed in ns block in G0, cutting feedrate specified by G73 in ns block in G1;

B₁→C₁: Cutting feed.

③ C₁→A₂: Rapid traverse.

④ Second roughing A₂→B₂→C₂ :

A₂→B₂: 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;

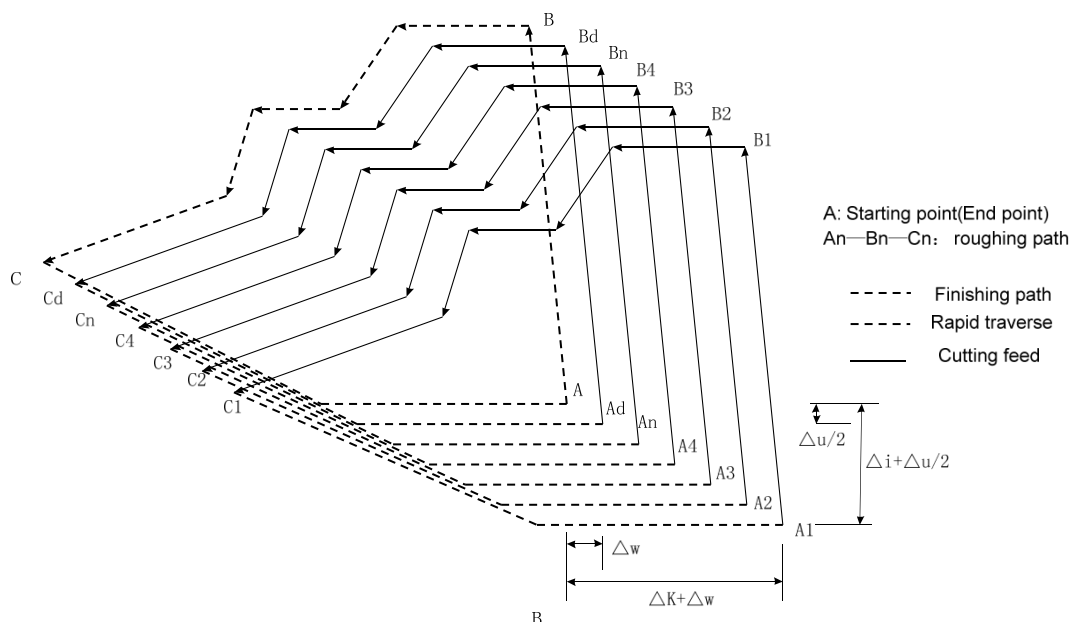


Fig. 3-31 G73 path

Command specifications:

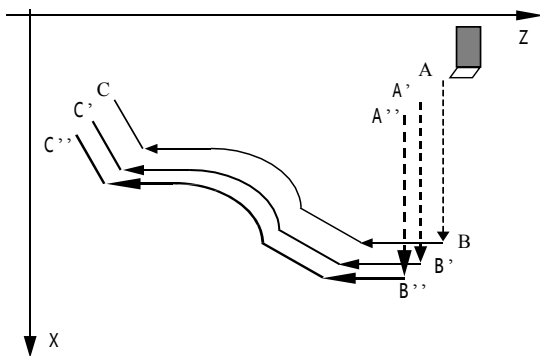
- ns ~ nf blocks in programming must be followed G73 blocks.
- ns ~ nf blocks are used for counting the roughing path and the blocks are not executed when G73 is executed. F, S, T commands of ns ~ nf blocks are invalid when G71 is executed, at the moment, F, S, T commands of G73 blocks are valid. F, 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.
- 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 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.
- Δi , Δu are specified by the same U and Δk , Δw are specified by the same W, 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.

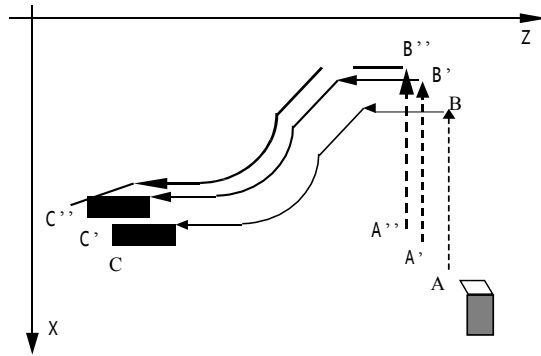
Coordinate offset direction with finishing allowance:

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.

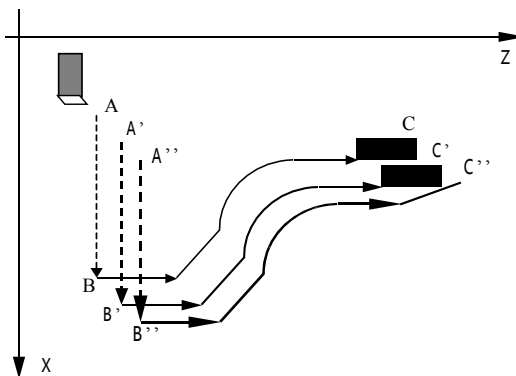
1) $\Delta i < 0$ $\Delta k > 0$, $\Delta u < 0$ $\Delta w > 0$;



2) $\Delta i > 0$ $\Delta k > 0$, $\Delta u > 0$ $\Delta w > 0$;



3) $\Delta i < 0$ $\Delta k < 0$, $\Delta u < 0$ $\Delta w < 0$;



4) $\Delta i > 0$ $\Delta k < 0$, $\Delta u > 0$ $\Delta w < 0$;

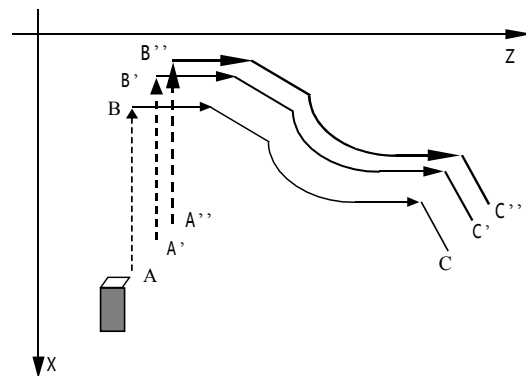


Fig.3-32

Example : Fig. 3-33

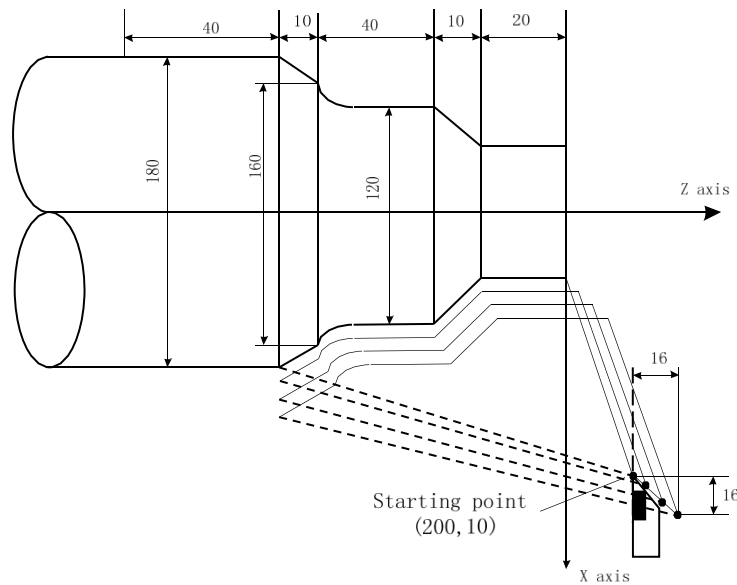


Fig.3-33

```

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 ;
G02 X160 W-20 R20 ;
N19 G01 X180 W-10 ;
G70 P14 Q19 M30;

```

} Blocks for finishing

(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
    .
    .
N__(nf).....
. . .
G70 P(ns) Q(nf) ;
. . .

```

} Blocks for finishing path

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(Δ i) Q(Δ k) R(Δ d) 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 $A_n(n=1,2,3,\dots)$, 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 $B_n(n=1,2,3,\dots)$, 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 $C_n(n=1,2,3,\dots)$, 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 $D_n(n=1, 2, 3,\dots)$, 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: $0 < \Delta i \leq 9999999 \times$ least input increment (diameter value), without sign symbol.

Q(Δk) : radial(Z) cutting for each axial cutting cycle, range: $0 < \Delta k \leq 9999999 \times$ least input increment (diameter value), without sign symbol.

R(Δd) : radial (X) tool retraction after cutting to end point of axial cutting, range: $0 \sim 99999.999 \times$ 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 Δk 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 ($\Delta k + e$) 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 ($\Delta k + e$) again and execute ②; after Z executing the cutting feed ($\Delta k + e$) 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 Δd (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 D_n , 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 ($\Delta d + \Delta i$) (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 ($\Delta d + \Delta i$) (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.

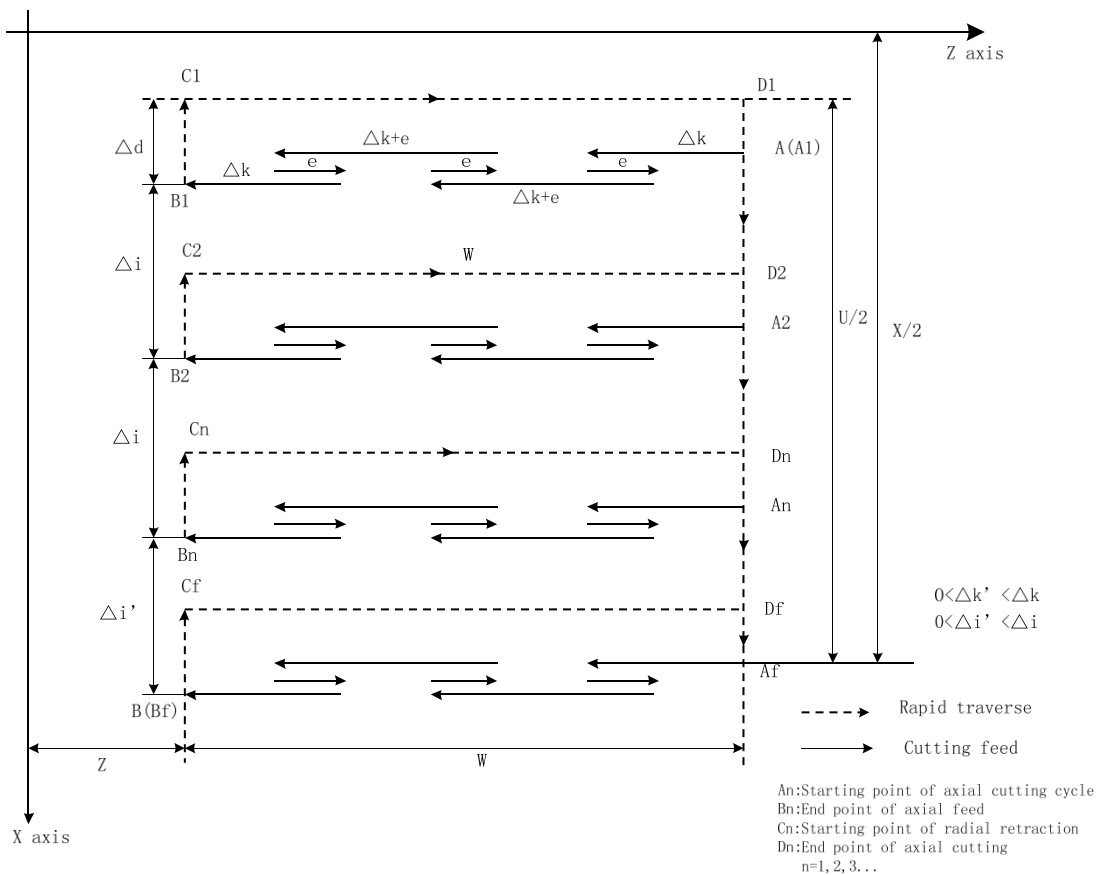


Fig. 3-34 G74 path

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.
- $R(\Delta d)$ 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

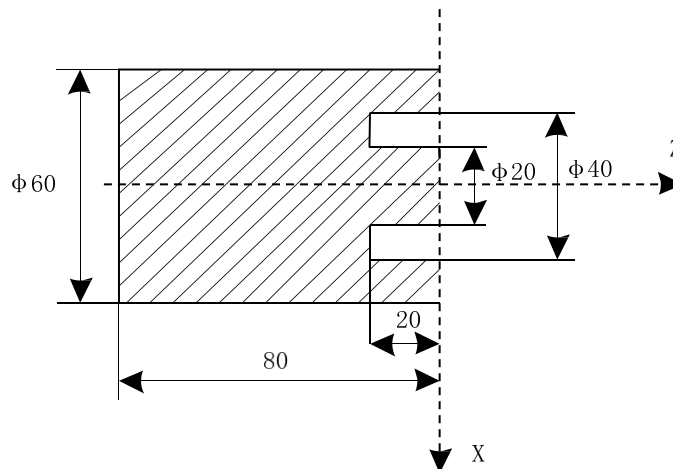


Fig.3-35

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 $A_n (n=1, 2, 3, \dots)$, 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 ($n=1, 2, 3, \dots$), 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 ($n=1, 2, 3, \dots$), 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 ($n=1, 2, 3, \dots$), 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: $0 < \Delta i \leq 9999999$ x least input increment without sign.

Q(Δk) : Axial(Z) discontinuous tool infeed of each radial cutting cycle, its range: $0 < \Delta k \leq 9999999 \times$ 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 \times 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.

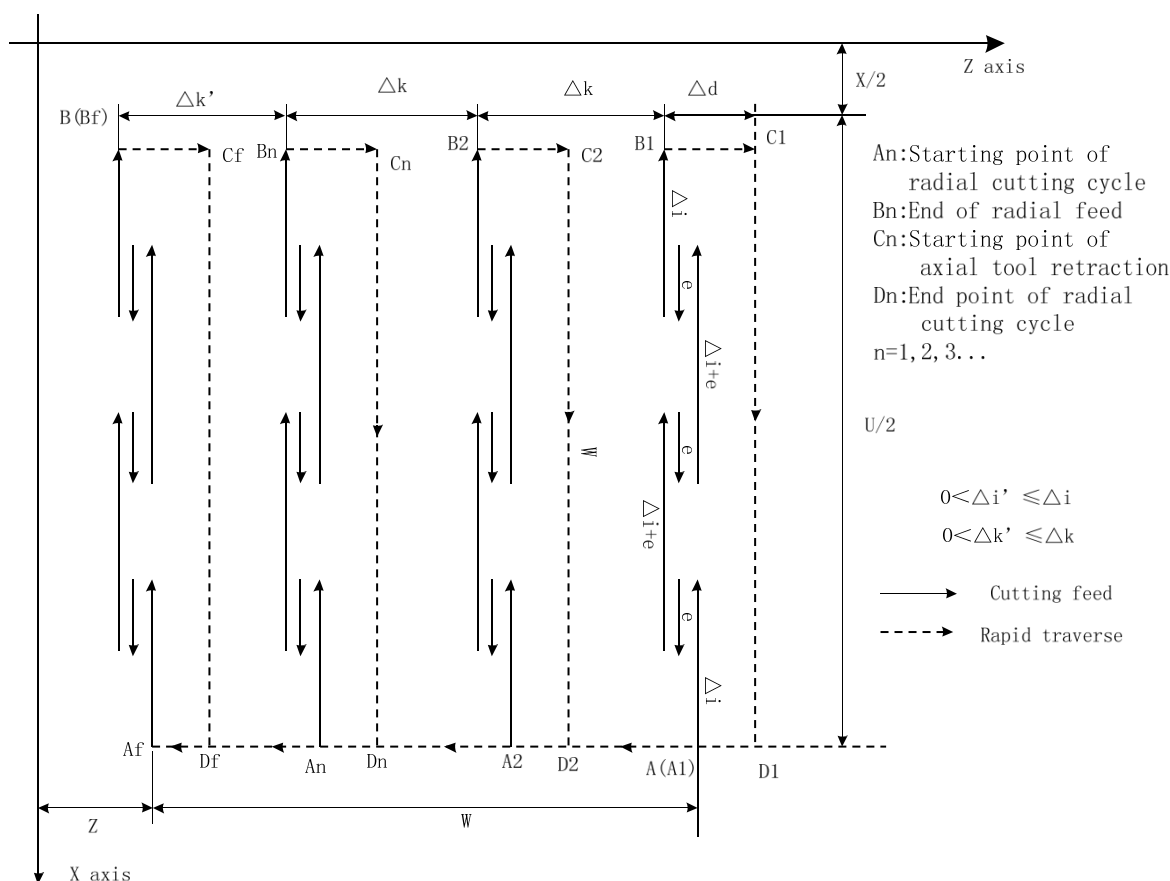


Fig. 3-36 G75 path

Execution process: (Fig. 3-36)

- ① Radial (X) cutting feed Δi 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 $(\Delta k + e)$ 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 $(\Delta i + e)$ again and executes ②; after X cutting feed $(\Delta i + e)$ 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 execute ④;
- ④ Axial(Z) rapid tool retraction Δd (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 D_n , 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 its 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 $(\Delta d + \Delta k)$ (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 $(\Delta d + \Delta k)$, 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

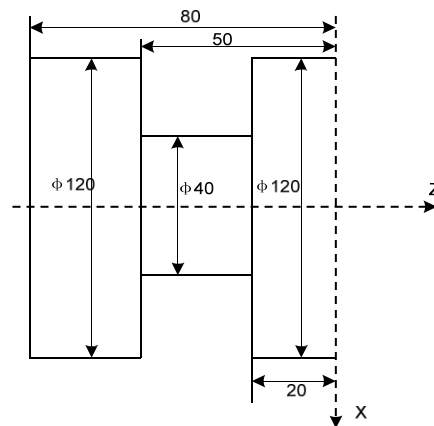


Fig. 3-37 G75 cutting

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

KY-980Tc 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 be 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°.

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° . Q180000 is input in program if it offsets 180° 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

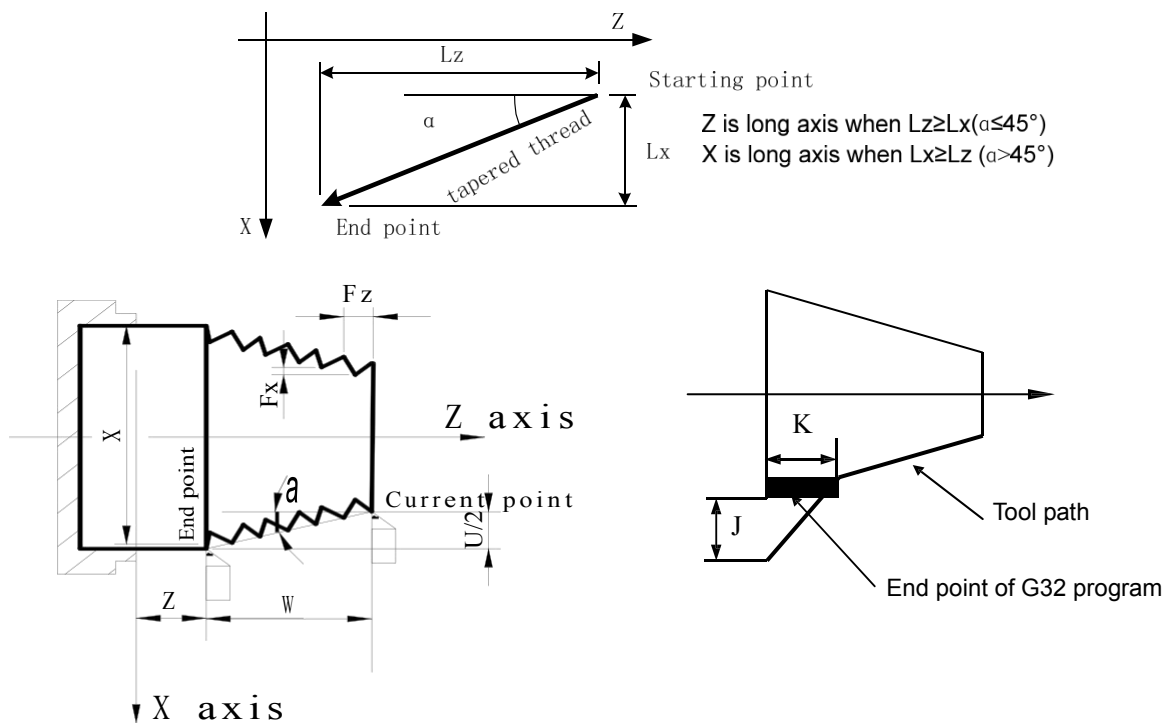


Fig. 3-38 G32 path

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 $J \neq 0, K=0$;
- There is no thread run-out when $J=0$ or $K \neq 0$;
- 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. $\delta 1 = 3\text{mm}$, $\delta 2 = 2\text{mm}$, total cutting depth 2mm divided into two times cut-in.

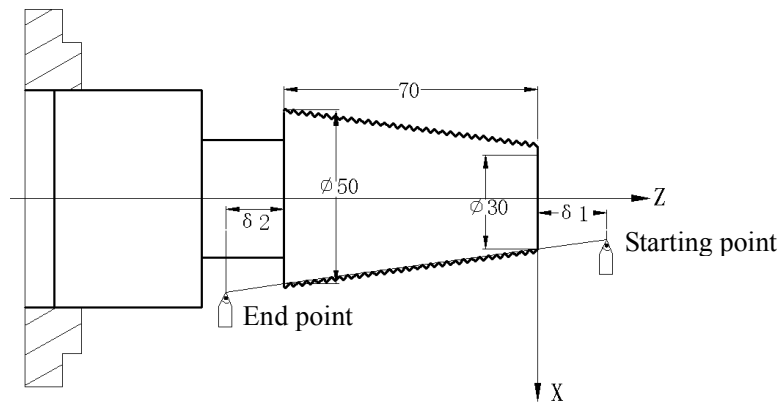


Fig.3-39

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: $\pm 0.001 \sim \pm 500.000$ mm/pitch (metric thread);
 $\pm 0.060 \sim \pm 25400$ 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.

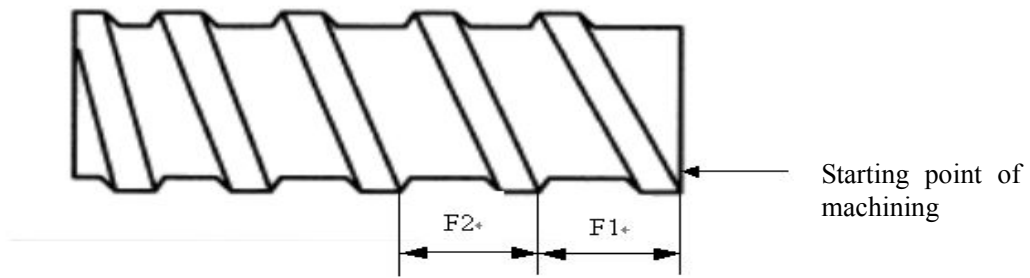


Fig. 3-40 Variable pitch thread

Note: It is the same as that of G32.

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

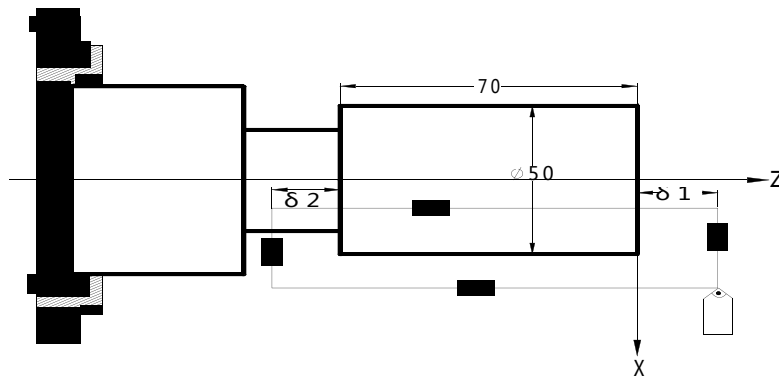


Fig. 3-42 Variable pitch thread machining

```

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;

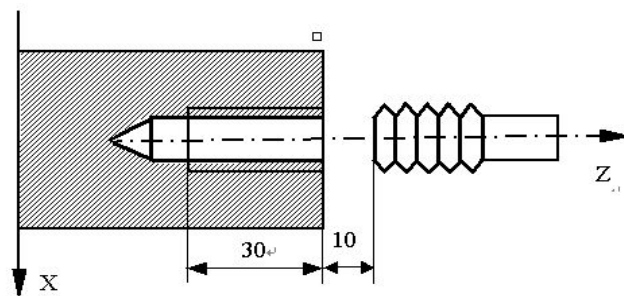
I: Teeth per inch thread 0.06 ~ 25400 teeth/inch; its range is referred to Table 1-4.

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



Program:

Fig. 3-43

```

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).

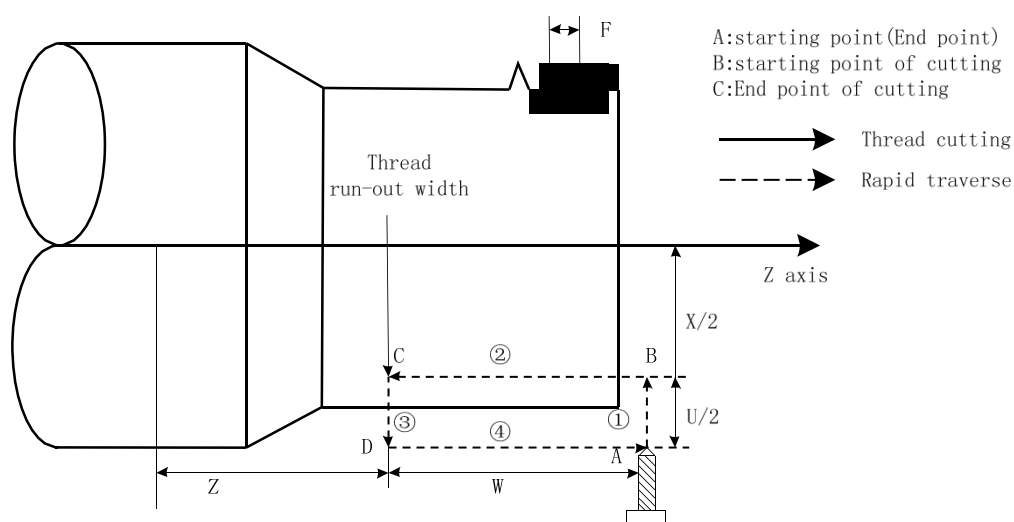


Fig. 3-44

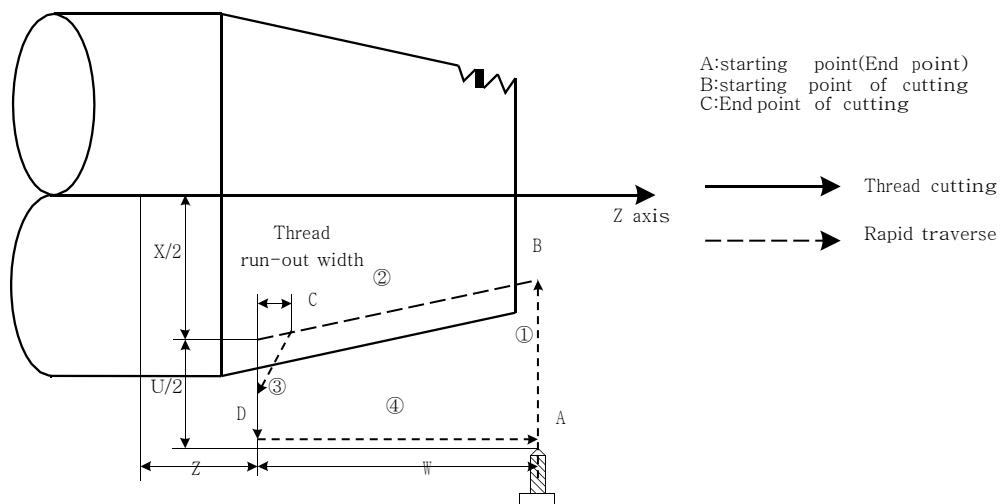


Fig. 3-45

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 N019 when J, K are omitted;
- Length of thread run-out is K in the long direction and is specified by N019 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:

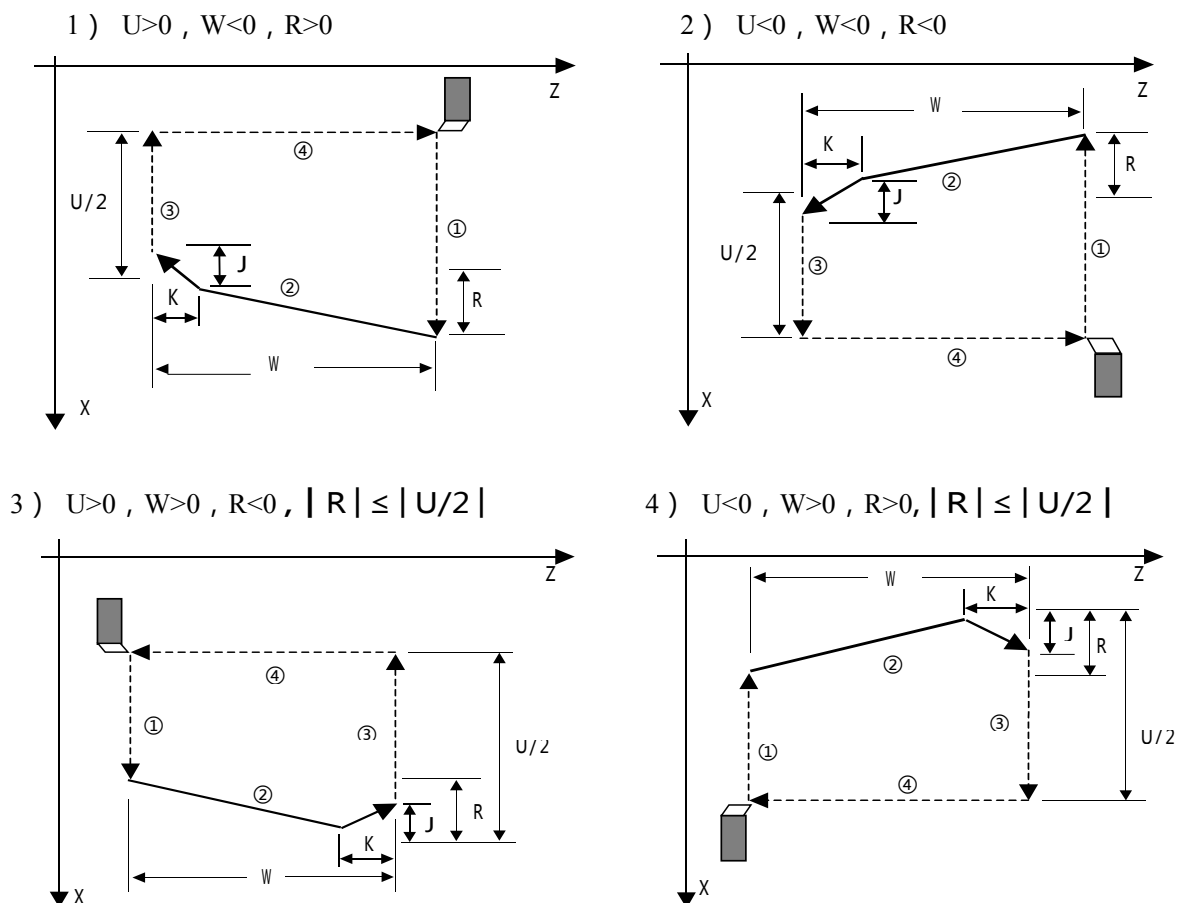


Fig.3-46

Example : Fig.3-47

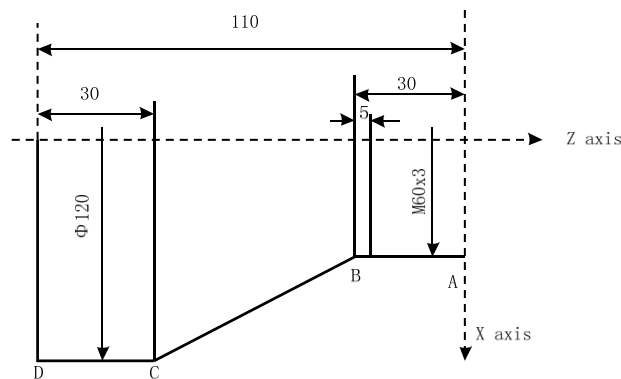


Fig.3-47

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(Δ dmin) R(d);

G76 X/U ___ Z/W ___ R(i) P(k) Q(Δ d) ___ F(l) ___;

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 $\sqrt{n} \times \Delta d$, n is the current roughing cycle times, Δd is the thread cutting depth of first roughing;

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

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.

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

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: $0.1 \times L$, 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 is 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 $(\sqrt{n} - \sqrt{n-1}) \times \Delta d < \Delta d_{min}$, Δd_{min} is regarded as the cutting travel of current roughing, i.e. depth of current thread cutting is $(\sqrt{n-1} \times \Delta d + \Delta d_{min})$. Setting Δd_{min} 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 Δd_{min} is valid and the system data parameter NO. 059 value is rewritten to Δd_{min} (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 B_e of thread finishing and B_f of thread roughing. After R(d) is executed, the specified value d is reserved and the system parameter №060 value is rewritten to $d \times 1000$ (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;

P(k): Depth of thread tooth, the total cutting depth of thread, range: 1 ~ 999999999× least input increment (radius value without sign symbols). The system alarms when P(k) 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



The diagram illustrates a multi-layered structure. On the left, a series of parallel lines represent layers, with points labeled Be, Bf, Bn, B2, B1, and B. To the right, a vertical stack of layers is shown with dimensions d , Δd , and k . A horizontal distance a is indicated between the points B1 and B.

96

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 B₁, and the thread cutting depth is Δd . 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 $a \neq 0$;
- ② The tool cuts threads paralleling with C→D to the intersection of D→E ($r \neq 0$: 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 $(\sqrt{n} \times \Delta d)$, $(\sqrt{n-1} \times \Delta d + \Delta d_{min})$, 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 ($r \neq 0$: 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(Δd_{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.

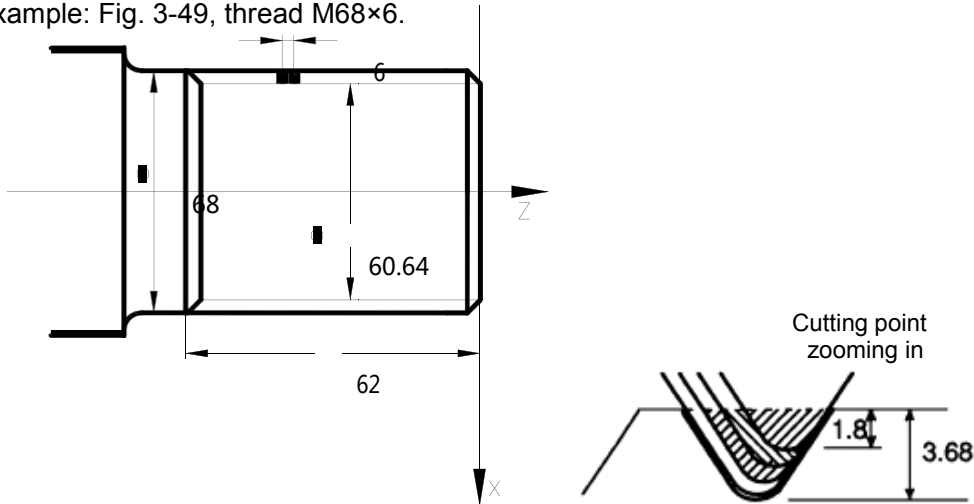


Fig.3-49

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 N076 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

KY980Tc 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 > +“ - # l”. 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		00099 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	#1500	#1600	#1700	#1800	#1900	#2000	#2100	#2200
...
32	#1531	#1631	#1731	#1831	#1931	#2031	#2131	#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 axis. For example: #5001 meanings to be X position information, #5002 meanings to be Y position information, #5003 meanings to be Z 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 P <u>n</u> Q# <u>j</u> R# <u>k</u> ;	Conditional jump 4	Jump to block n if # j < # k, otherwise the system executes in order
G65 H85 P <u>n</u> Q# <u>j</u> R# <u>k</u> ;	Conditional jump 5	Jump to block n if # j ≥ # k, otherwise the system executes in order
G65 H86 P <u>n</u> Q# <u>j</u> R# <u>k</u> ;	Conditional jump 6	Jump to block n if # j ≤ # k, otherwise the system executes in order
G65 H99 P <u>n</u> ;	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: $\# I = \# J \times \# K$

G65 H04 P#I Q#J R#K

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

- 5) Decimal division operation: $\# I = \# J \div \# K$

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 = $\sqrt{\text{\#J}}$

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: $\# I = \# J - \text{TRUNC}(\# J / \# K) \times \# K$, 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 = $\sqrt{\#101^2 + \#102^2}$)

- 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.

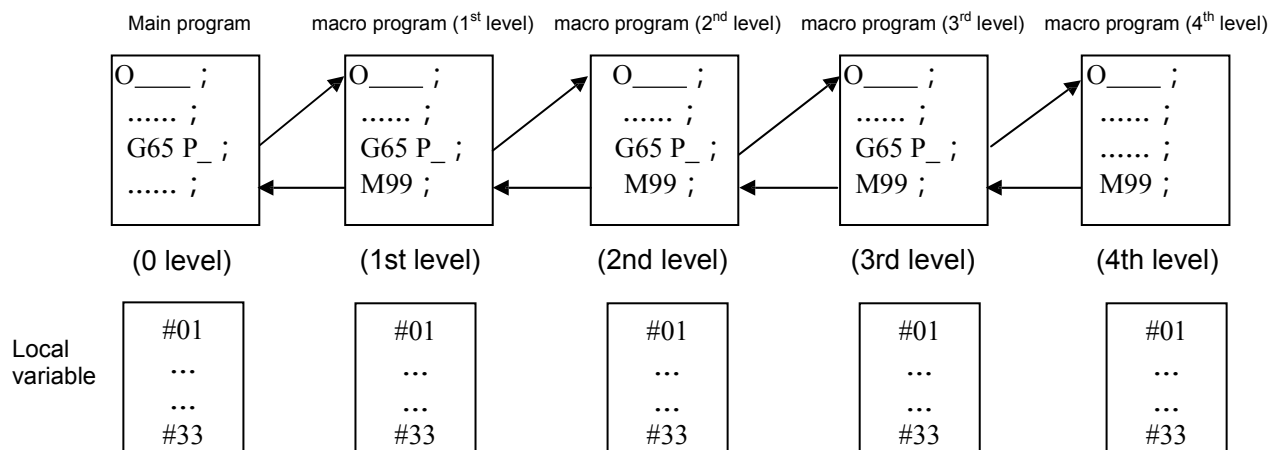
Command explanation:

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 KY-980Tc 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.

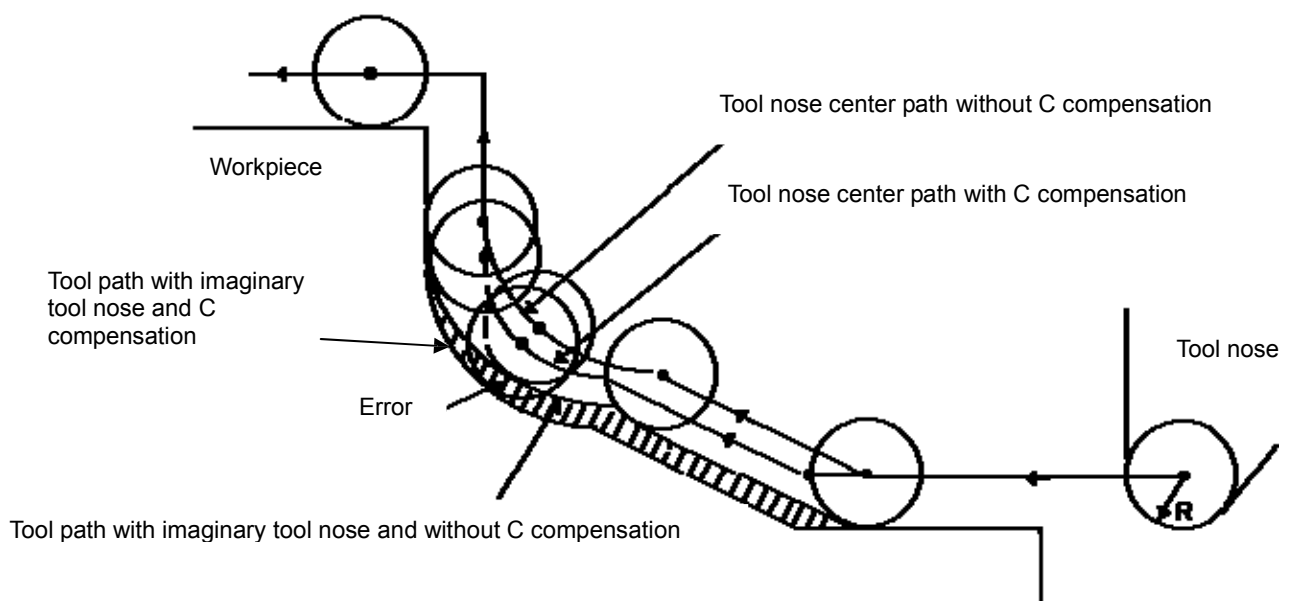


Fig. 4-2 Tool nose center path

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.

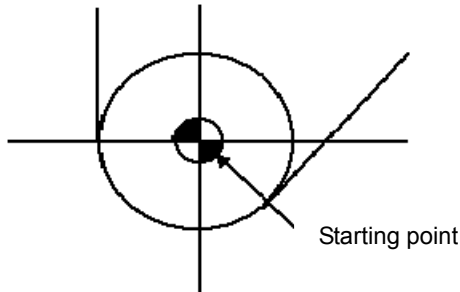


Fig. 4-3 Programming with tool nose

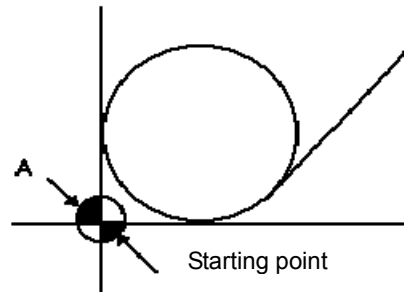
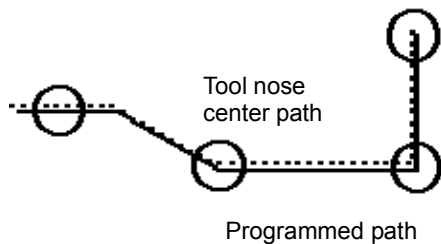
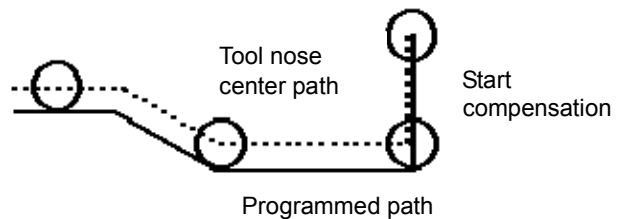


Fig. 4-4 Programming with imaginary tool nose

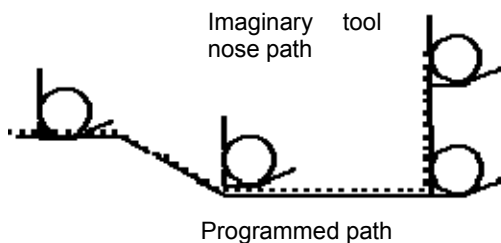
Tool nose path is the same as programming path without using tool nose radius compensation



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

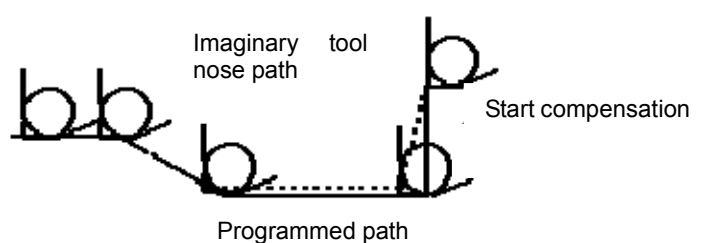


Fig. 4-5 Tool path in tool nose center programming

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.

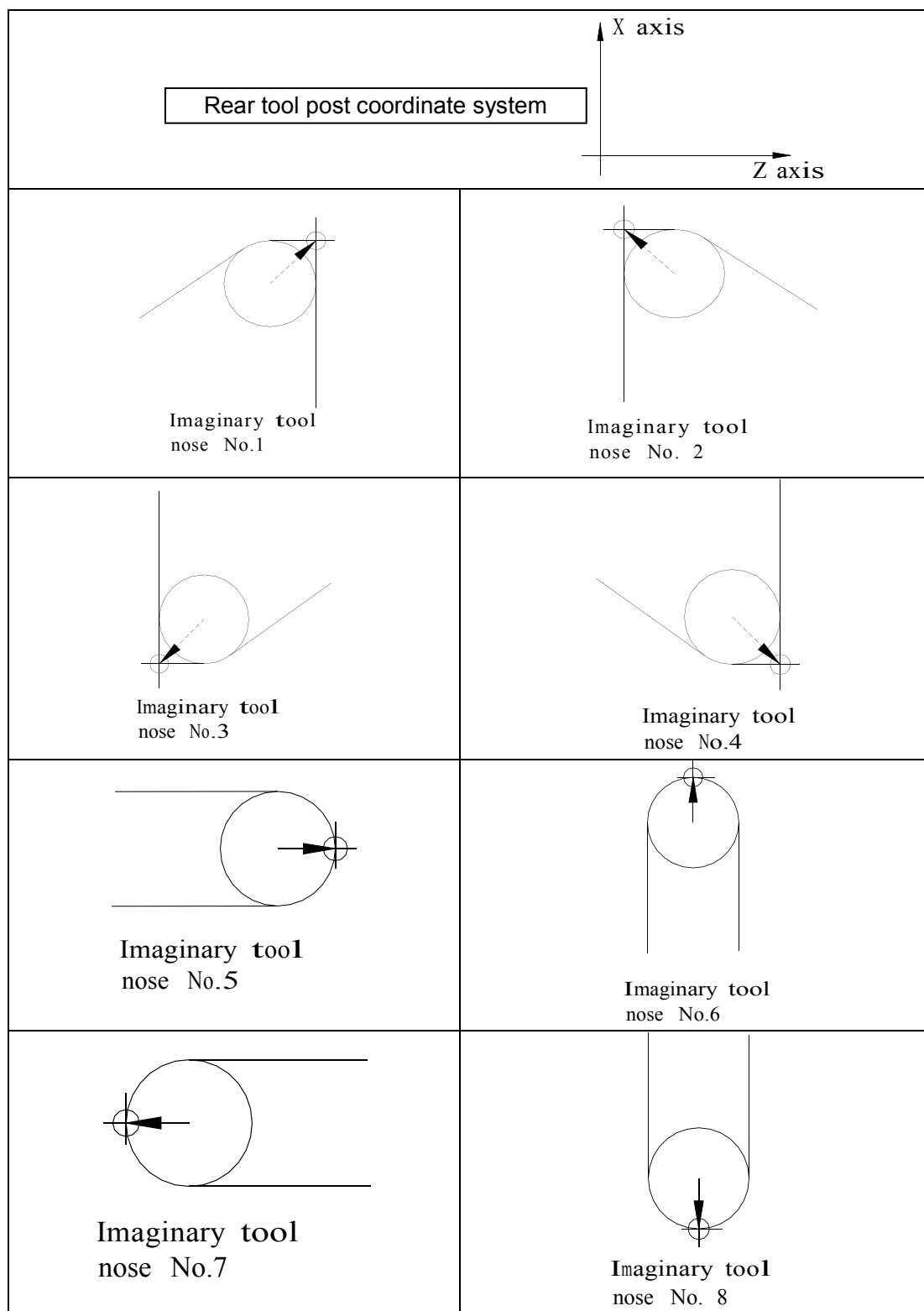


Fig. 4-7 Imaginary tool nose number in rear tool post coordinate system

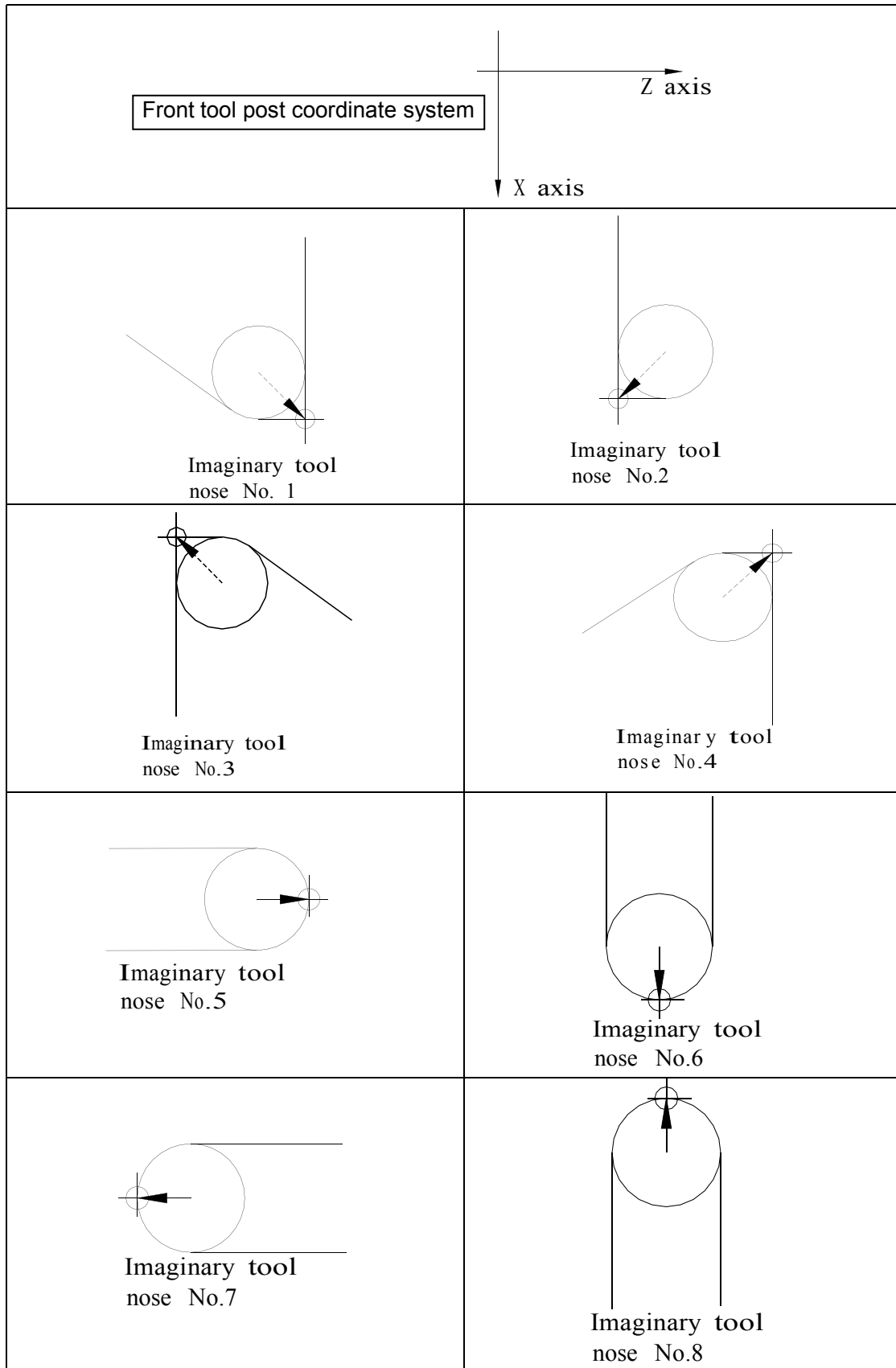


Fig. 4-8 Imaginary tool nose number in front tool post coordinate system

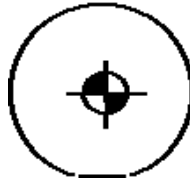


Fig. 4-9 Tool nose center on starting point

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 T_n (n=0~9) when taking T_n(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).

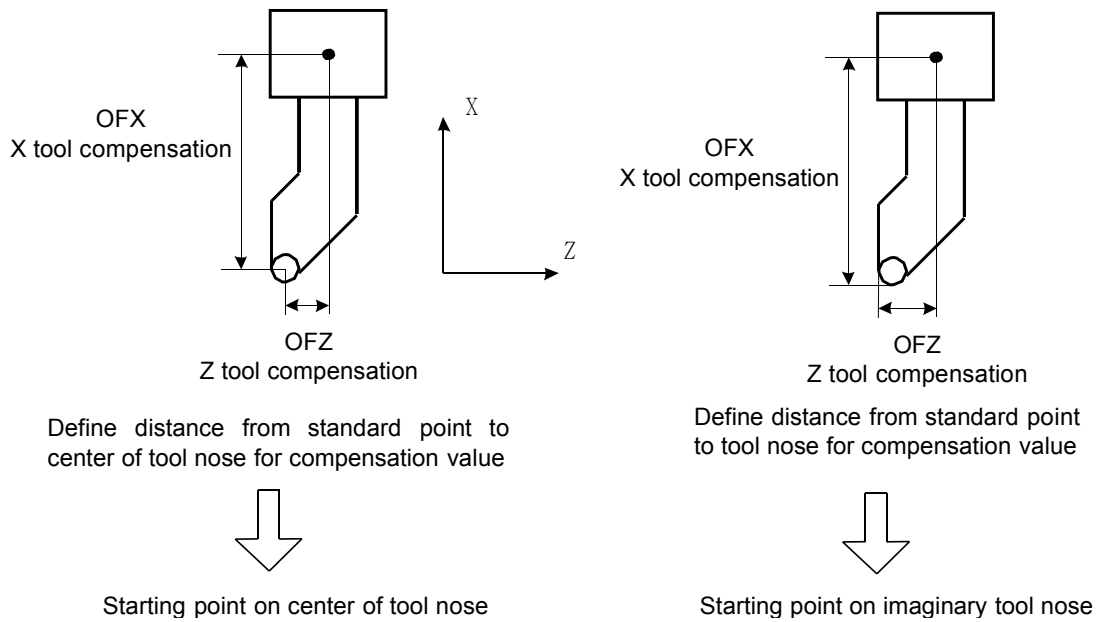


Fig. 4-10 Tool offset value of tool post center as benchmark

4.1.4 Command format

$$\left\{ \begin{array}{l} G40 \\ G41 \\ G42 \end{array} \right\} \left\{ \begin{array}{l} G00 \\ G01 \end{array} \right\} X_ Z_ T_ ;$$

Commands	Function specifications	Remark
G40	Cancel the tool nose radius compensation	See Fig.4-11 and 4-12
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	
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.

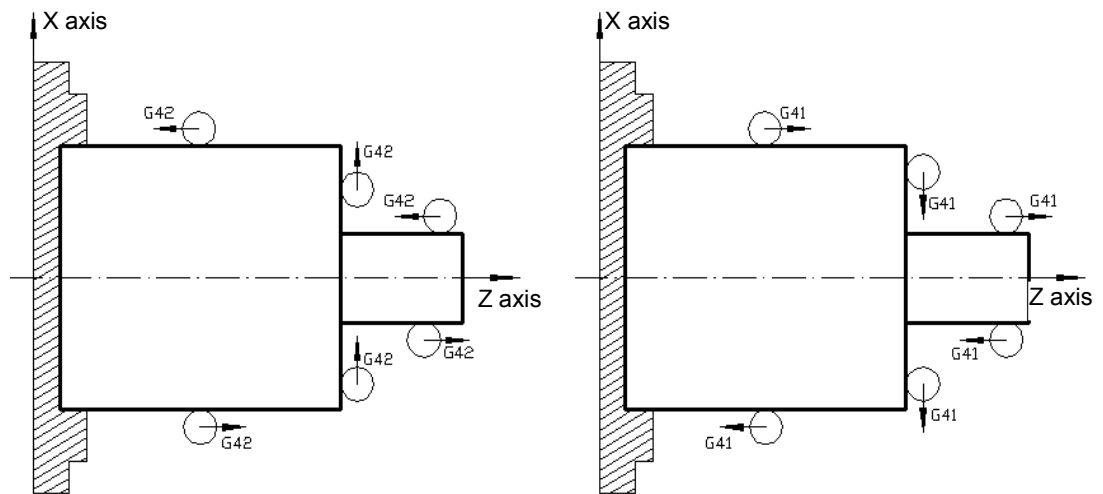
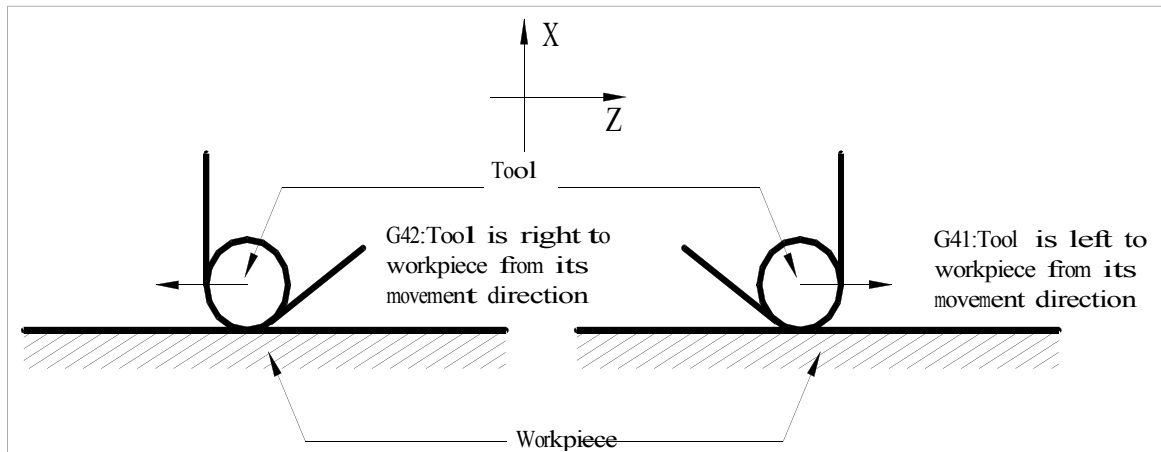


Fig. 4-11 Compensation direction of rear coordinate system

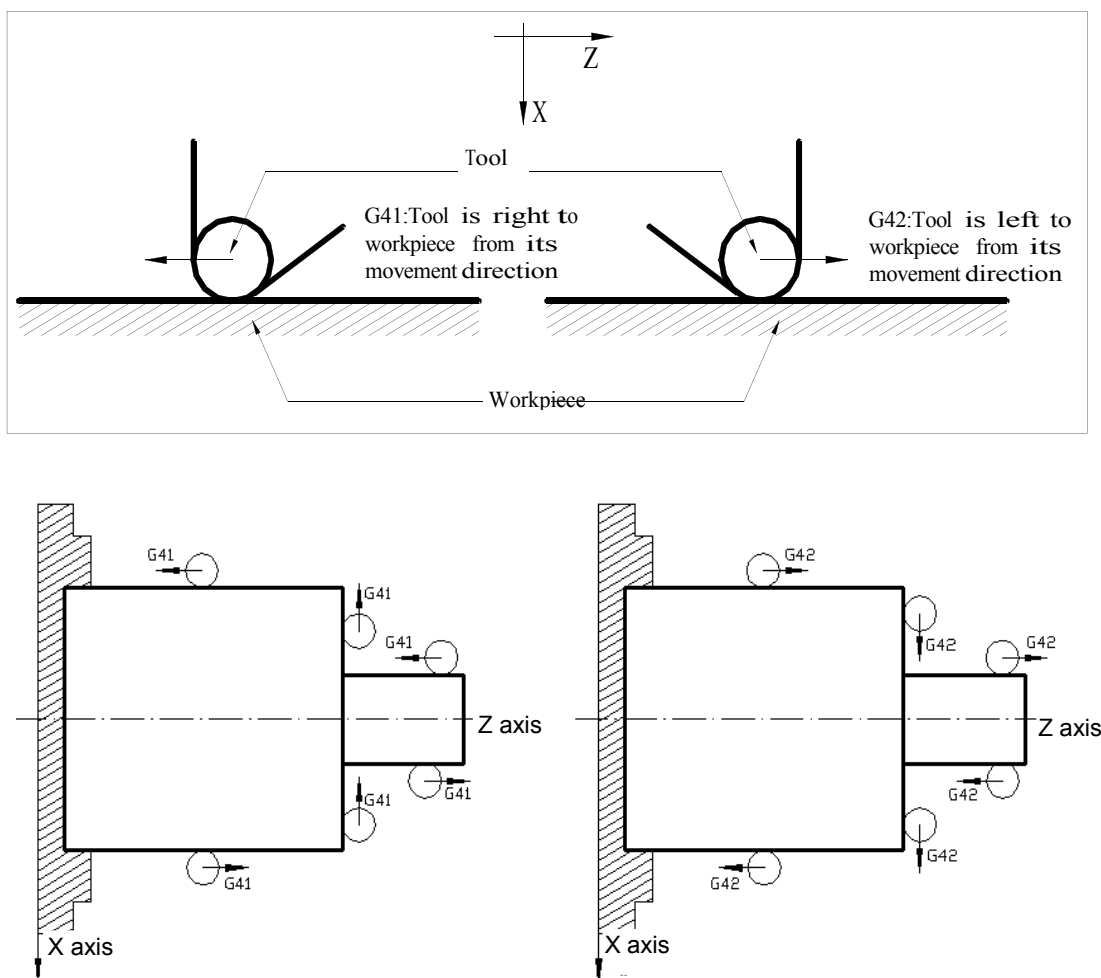



Fig. 4-12 Compensation direction of front coordinate system

4.1.6 Notes

- z 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.
- z 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.
- z The system cannot create and cancel tool nose radius compensation.
- z Tool nose radius R is without negative value, otherwise there is a mistake running path.
- z Tool nose radius compensation is created and cancelled in G00 or G01 instead of G02 or G03, otherwise, the system alarms.
- z The system cancels the tool nose radius compensation mode when pressing  key.
- z G40 must be specified to cancel offset mode before the program is ended, otherwise the tool path offsets one tool nose radius.
- z 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.

- z The system does not execute the tool nose radius compensation in G71, G72, G73, G74, G75, G76 and cancel it temporarily.
- z 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.

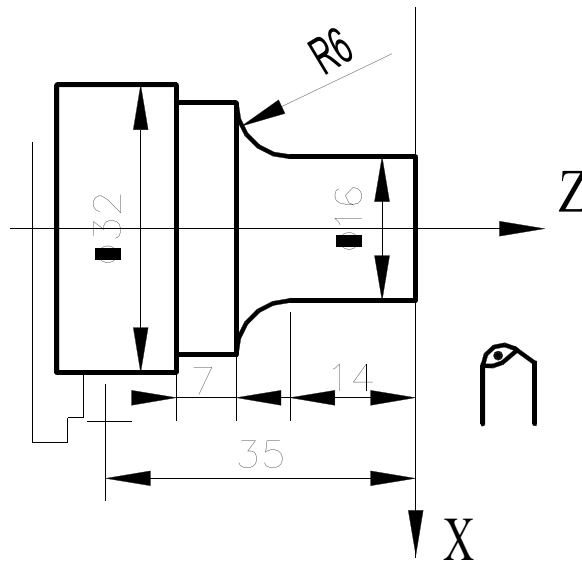


Fig. 4-13

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;
 G00 X100 Z50 T0100;
 M30;

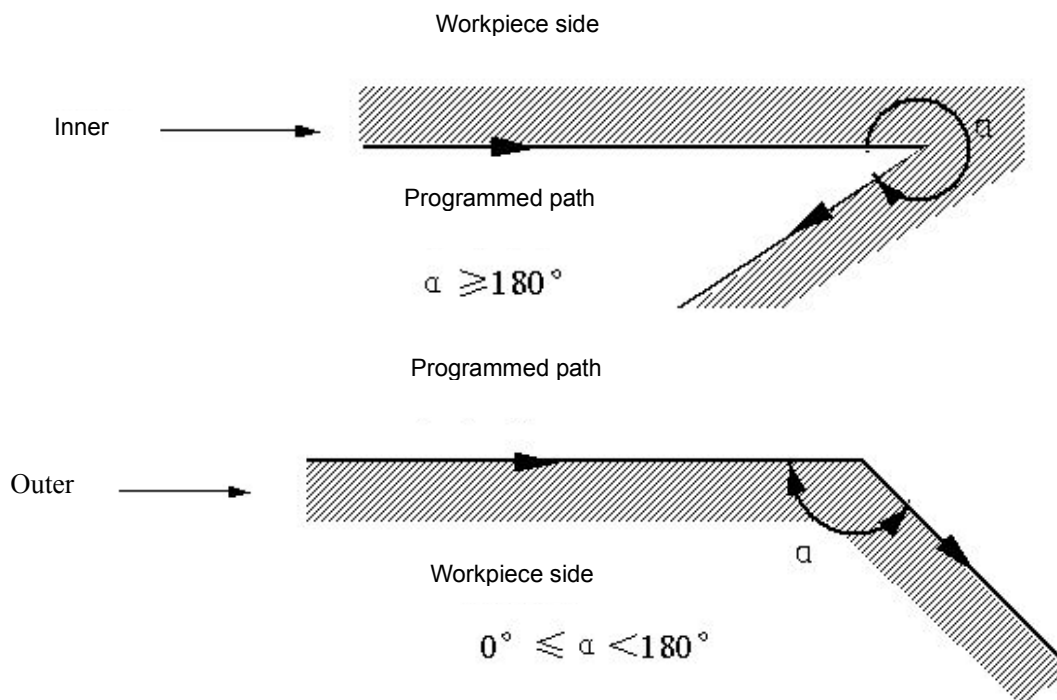
(Cancel tool nose radius compensation)

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 $0^\circ \sim 180^\circ$.



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.

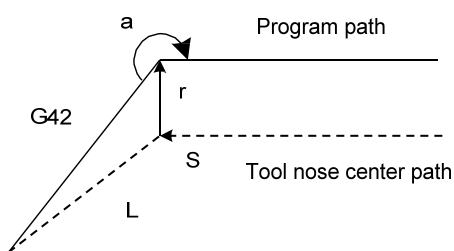
Tool traverse is called tool compensation creation (starting tool) from offset canceling to G41 or G42 execution.

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($\alpha \geq 180^\circ$)

1) linear —linear



2) Linear —circular

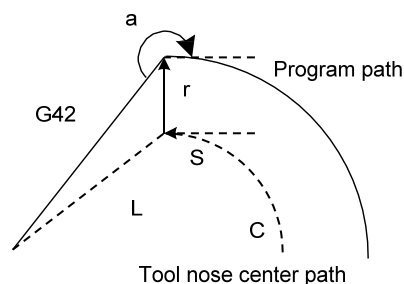
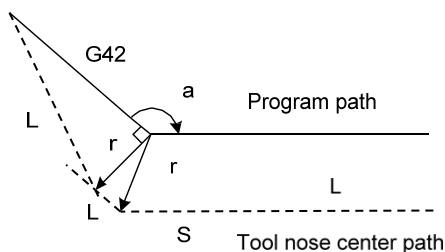


Fig.4-14a Linear —linear(starting tool inside)

Fig. 4-14b Linear —circular (starting tool inside)

(b) Tool traversing inside along corner($180^\circ > \alpha \geq 90^\circ$)

1) linear —linear



2) linear —circular

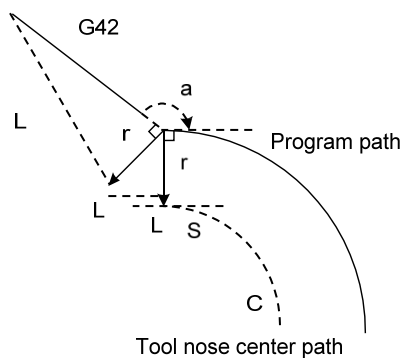
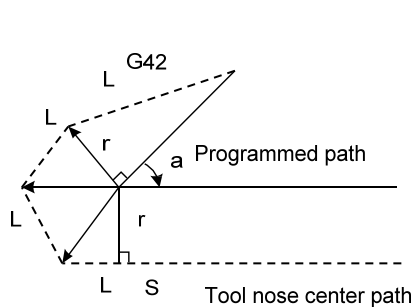


Fig.4-15a Linear —linear(starting tool outside)

Fig.4-15b Linear—circular(starting tool outside)

(c) Tool traversing inside along corner ($\alpha < 90^\circ$)

1) linear —linear



2) linear —circular

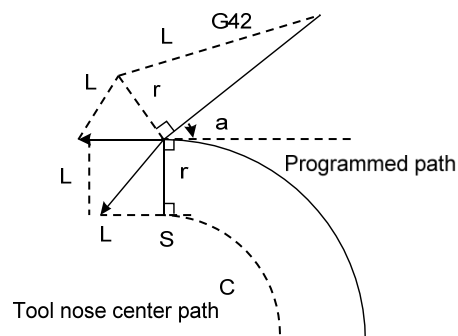
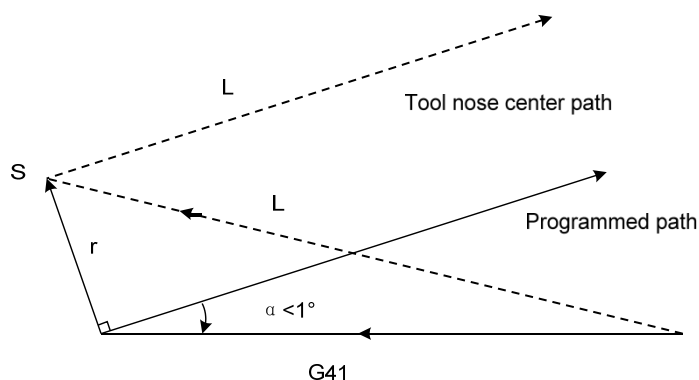


Fig.4-16a Linear —linear(starting tool outside)

Fig. 4-16b Linear—circular (starting tool outside)

(d) Tool traversing inside along corner ($\alpha \leq 1^\circ$), linear \rightarrow linearFig. 4-17 Linear—linear ($\alpha < 1^\circ$, 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.

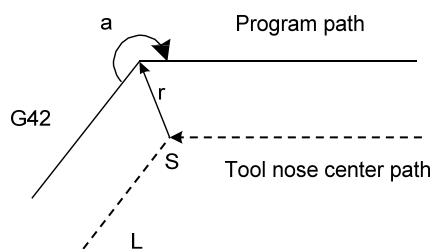
Z Offset path without changing compensation direction in compensation mode**(a) Tool traversing inside along corner ($\alpha \geq 180^\circ$)****1) linear —linear**

Fig. 4-18a Linear —linear (moving inside)

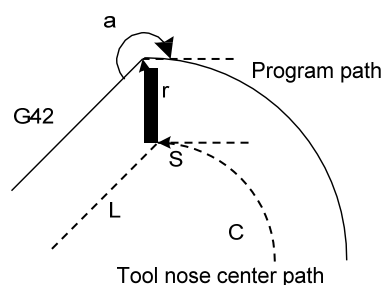
2) linear —circular

Fig. 4-18b Linear—circular (moving inside)

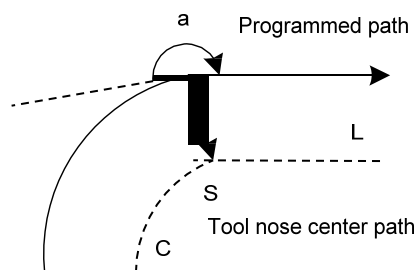
3) circular—linear

Fig. 4-18c Circular—linear (moving inside)

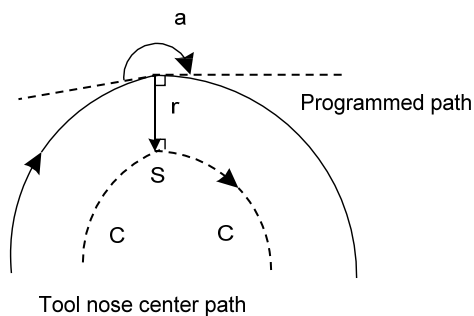
4) circular —circular

Fig. 4-18d Circular—circular (moving inside)

(b) Tool traversing outside along corner ($180^\circ > \alpha \geq 90^\circ$)

1) linear —linear

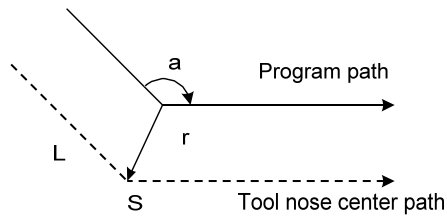


Fig. 4-19a Linear —linear
($180^\circ > \alpha \geq 90^\circ$, obtuse angle, moving outside)

2) linear —circular

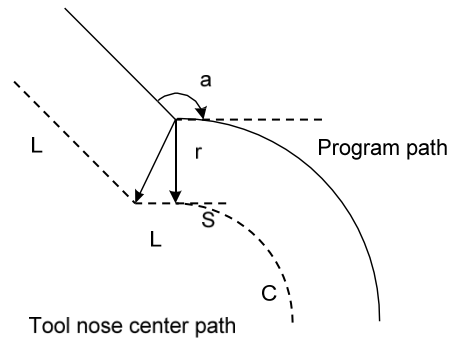


Fig. 4-19b Linear—circular
($180^\circ > \alpha \geq 90^\circ$, obtuse angle, moving outside)

3) circular—linear

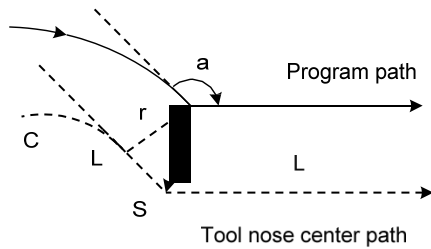


Fig. 4-19c circular —linear
($180^\circ > \alpha \geq 90^\circ$, obtuse angle, moving outside)

4) circular —circular

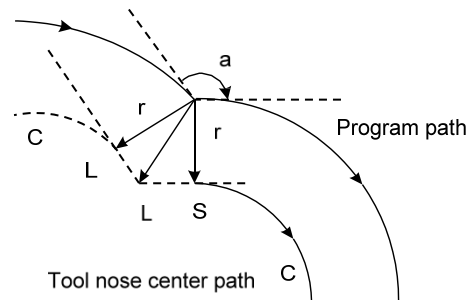
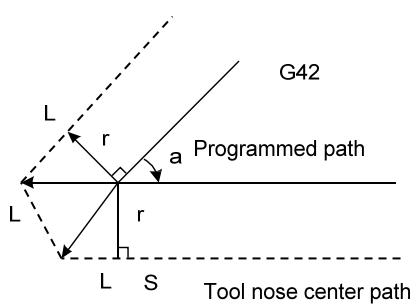


Fig. 4-19d circular —circular
($180^\circ > \alpha \geq 90^\circ$, obtuse angle, moving outside)

(c) Tool traversing outside along corner ($\alpha < 90^\circ$)

1) linear—linear



2) linear—circular

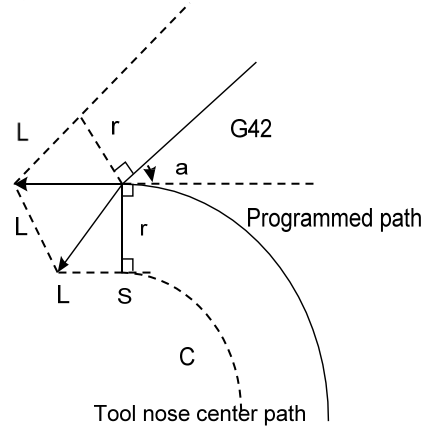
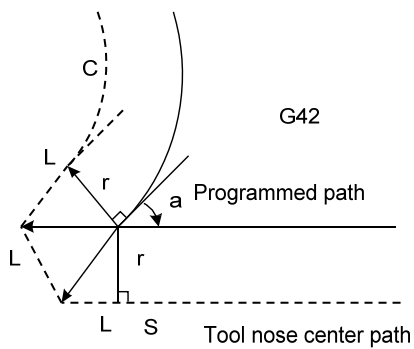


Fig. 4-20a Linear—Linea (moving outside)

Fig. 4-20b Linear—circular (moving outside)

3) circular—linear



4) circular—circular

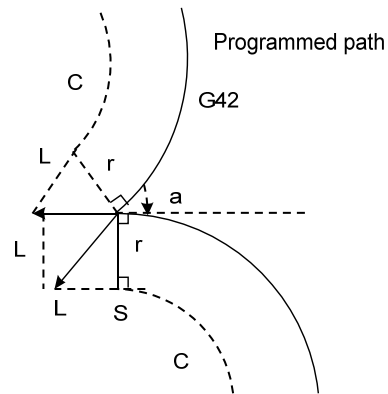


Fig.4-20c Circular—linear(moving outside)

Fig.4-20d Circular—circular(moving outside)

5) Machining inside ($\alpha < 1^\circ$) and zoom in the compensation vector

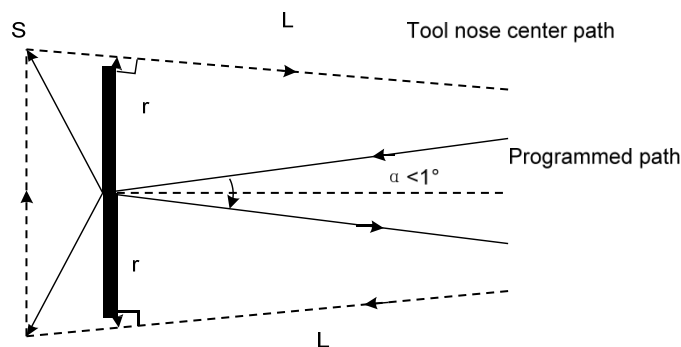


Fig. 4-20e Linear —linear ($\alpha < 1^\circ$, moving inside)

(d) Special cutting

1) Without intersection

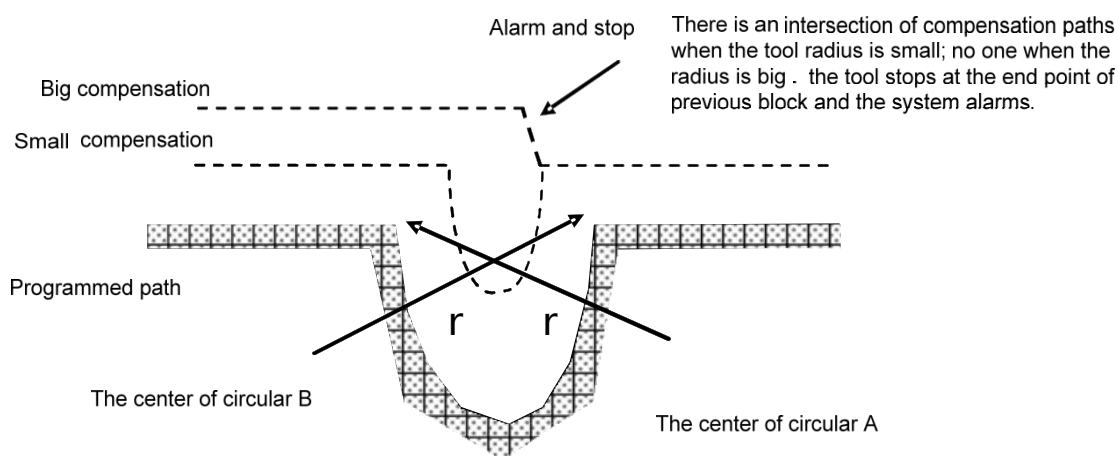


Fig. 4-21 Paths without intersection after offset

2) Center point and starting point of circular being the same one

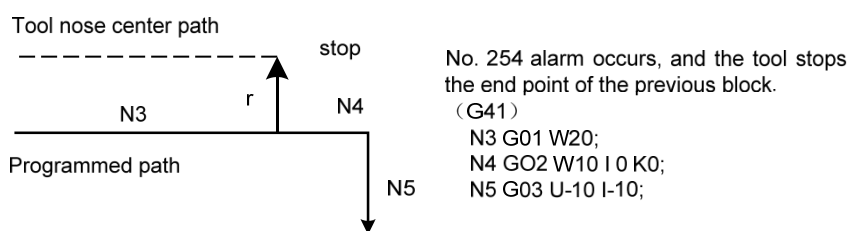


Fig. 4-22 Center point and starting point of circular being the same one

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	
	+	-
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

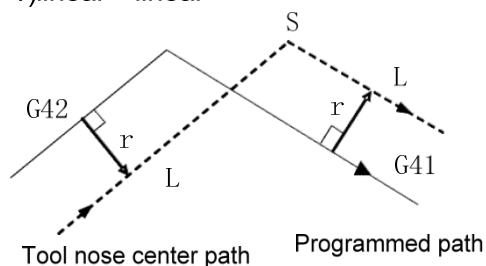


Fig. 4-23 Linear—linear
(changing compensation direction)

2) linear—circular

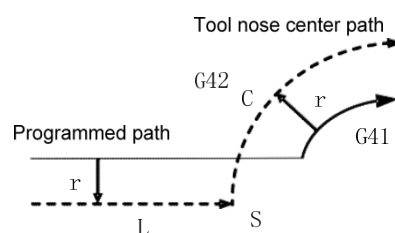


Fig. 4-24 Linear—circular
(changing compensation direction)

3) circular—linear

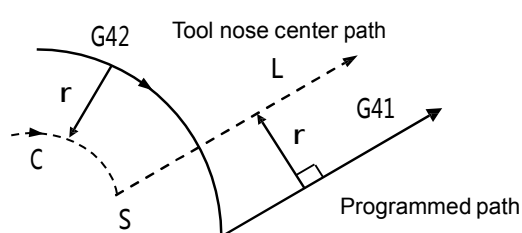


Fig. 4-25 circular—linear
(changing compensation direction)

4) circular—circular

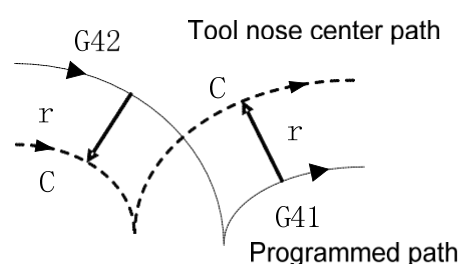


Fig. 4-26 circular—circular
(changing compensation direction)

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

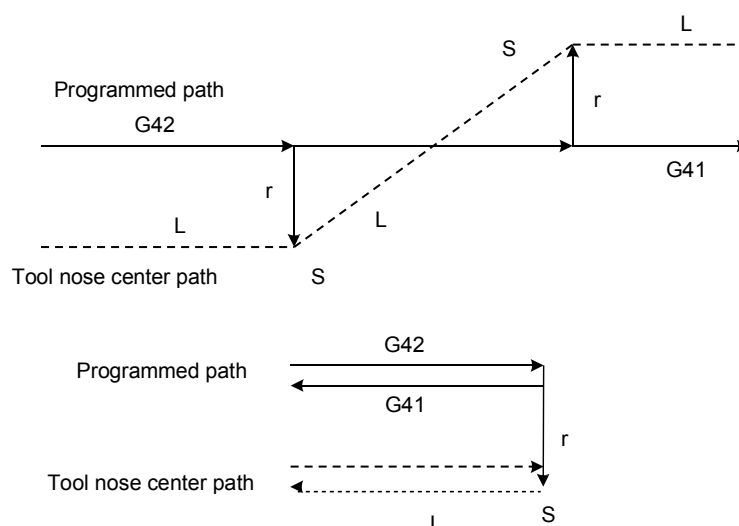


Fig. 4-27a Linear—linear, no intersection (changing compensation direction)

ii) Linear ---circular

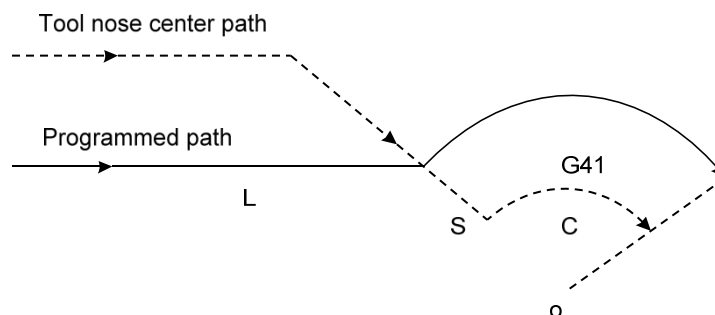


Fig. 4-27b Linear—circular without intersection (changing compensation direction)

iii) Circular-----circular

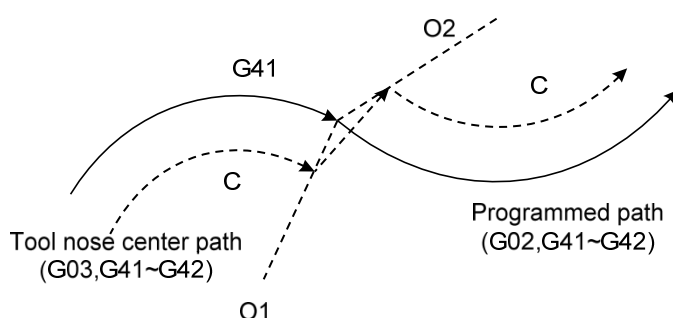


Fig. 4-27c Circular—circular without intersection (changing compensation direction)

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 block.

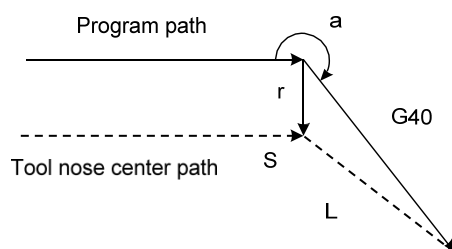
1. Execute G40 in a program;
2. Execute M30.

The system cannot execute G02 and G03 when canceling C tool compensation (tool nose radius compensation), otherwise the system alarms and stops run.

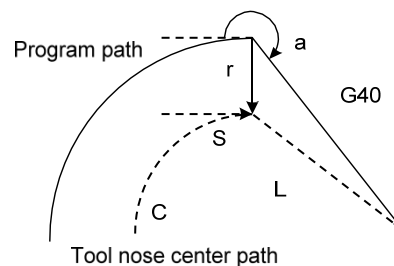
In compensation canceling mode, the system executes the block and ones in the register for tool nose radius compensation. At the moment, the run stops after one block is executed when single block is ON. The system executes the next one but does not read its following one when pressing **CYCLE START** button again.

(a) Tool traversing inside along corner($\alpha \geq 180^\circ$)

1) linear —linear

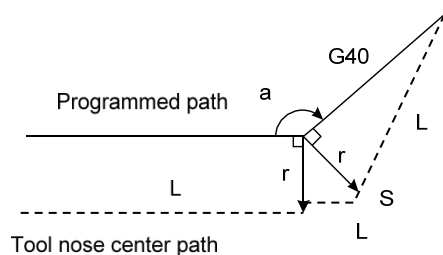
Fig. 4-28a linear-linear
(moving inner and canceling offset)

2) circular—linear

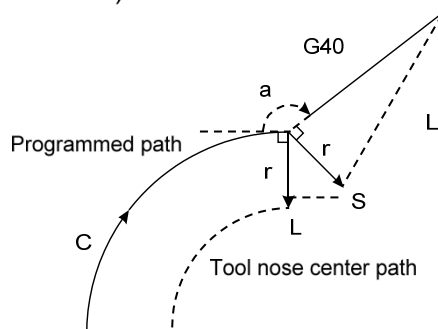
Fig. 4-28b Circular-linear
(moving inner and canceling offset)

(b) Tool traversing outside along corner($180^\circ > \alpha \geq 90^\circ$)

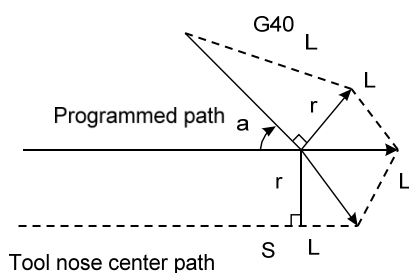
1) linear—linear

Fig. 4-29a linear—linear ($\alpha \geq 90^\circ$ moving outside and canceling offset)

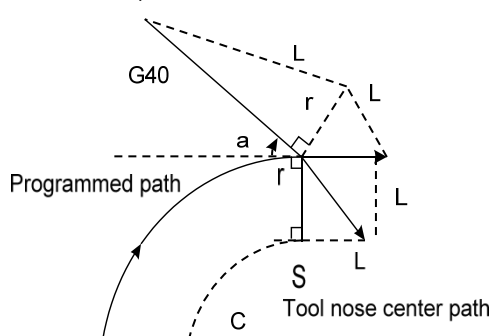
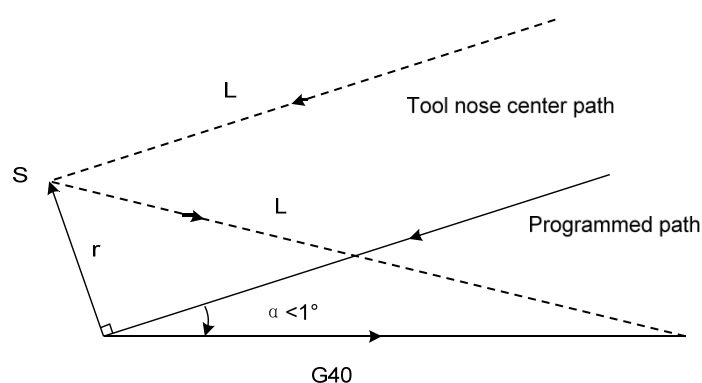
2) circular—linear

Fig. 4-29b Circular—linear ($\alpha \geq 90^\circ$ moving outside and canceling offset)**(c) Tool traversing outside along corner($\alpha < 90^\circ$)**

1) linear—linear

Fig. 4-30a Linear—linear ($\alpha < 90^\circ$ cutting outside and canceling offset)

2) circular—linear

Fig. 4-30b Circular—linear ($\alpha < 90^\circ$ cutting outside and canceling offset)**(d) Tool traversing outside along corner($\alpha < 1^\circ$); linear→linear**Fig. 4-31 Linear—linear ($\alpha < 1^\circ$ 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 $90^\circ \sim 270^\circ$).
- 2) There is a big difference ($\alpha > 180^\circ$) 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

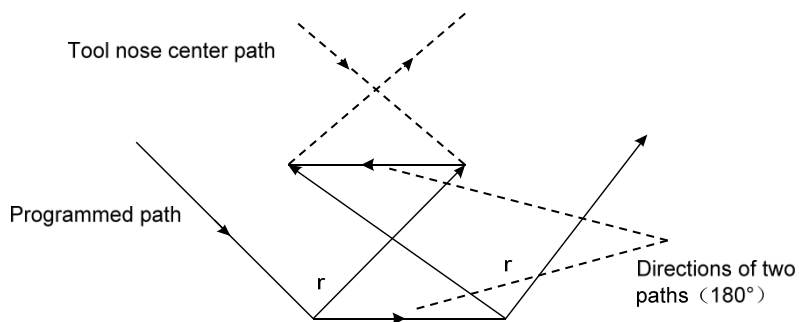


Fig. 4-32a Machining interference (1)

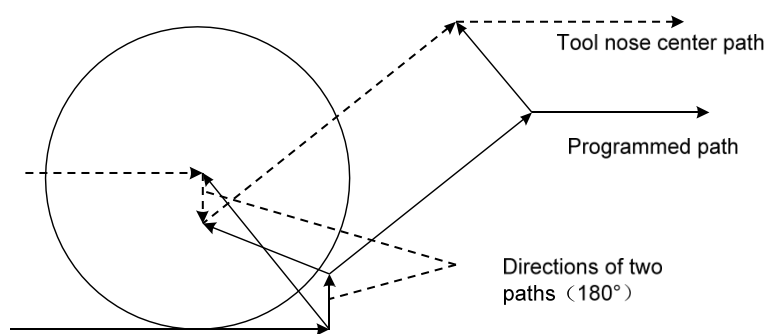


Fig. 4-32b Machining interference (2)

(2) Executing it without actual interference

- 1) Concave groove less than compensation value

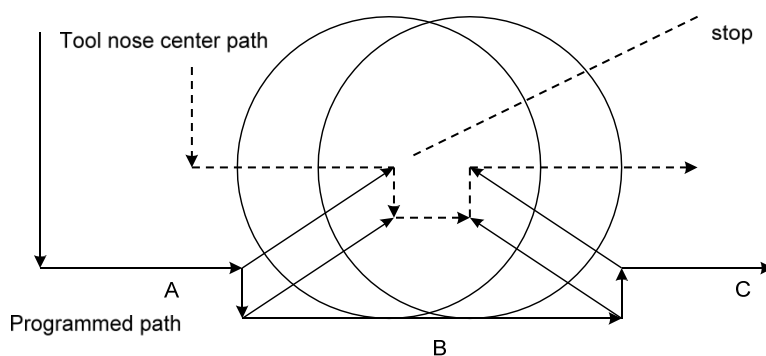


Fig. 4-33 Executing interference (1)

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

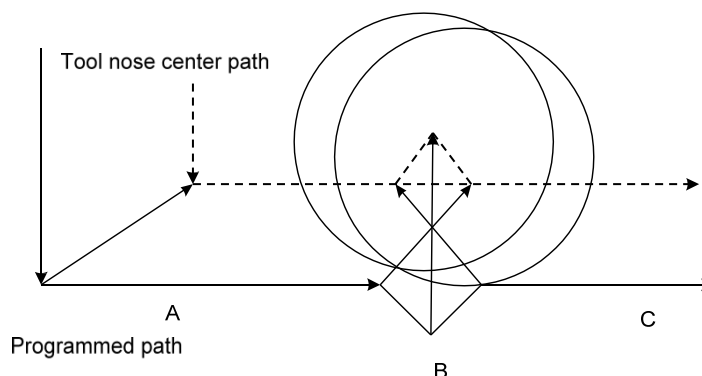


Fig. 4-34 Executing interference (2)

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.

Z Setting coordinate system in G50

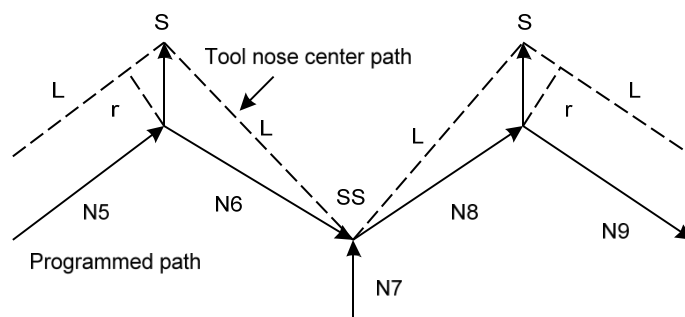


Fig. 4-35 Temporary compensation vector in G50

Note: SS indicates a point at which the tool stops twice in Single mode.

Z 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.

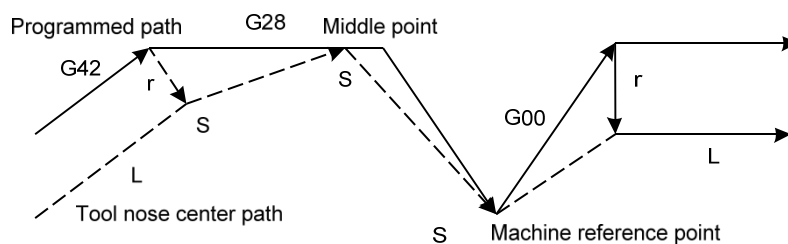


Fig. 4-36 Cancel compensation vector temporarily in G28

┌ **G71 ~ G75 compound cycle; G76, G92 thread cutting**

When executing G71 ~ G76, G92 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".

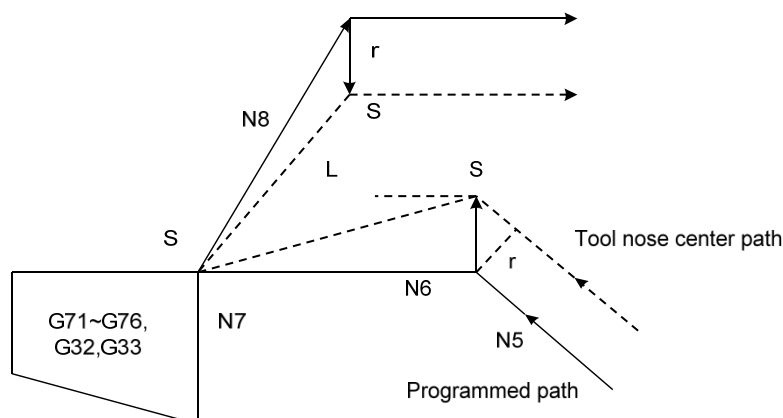


Fig. 4-37 Cancel compensation vector temporarily in G71 ~ G76

┌ **G90, G94**

Compensation method of tool nose radius compensation in G90 or G94:

- Cancel the previous tool nose radius compensation;
- Create the previous C compensation before cutting, and the path ① in the following figure creates the previous radius compensation mode;
- The paths ②, ③ in the following figure are the radius compensation cutting;
- 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.

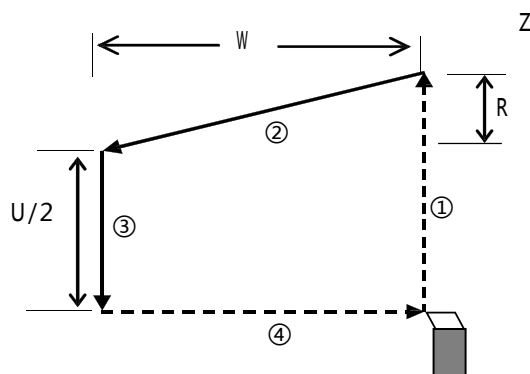


Fig. 4-38 Offset direction of tool nose radius compensation in G90

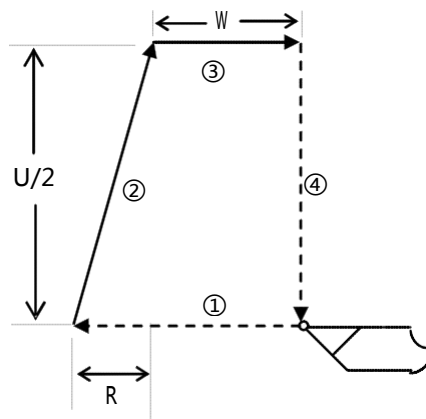


Fig. 4-39 Offset direction of tool nose radius compensation in G94

4.2.7 Particulars

z 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.

z 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.

z 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).

z 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.

z 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.

z 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

KY-980Tc CNC system uses an integrated panel, which is divided as follows:



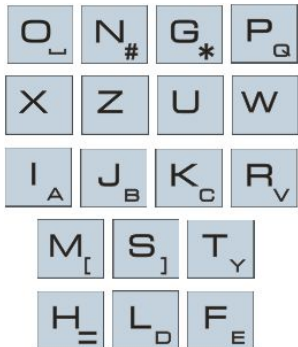
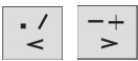
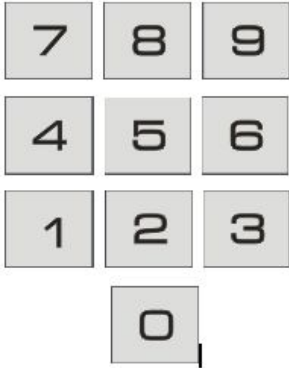









KY 980Tc panel division







1.1.1 State indication

	Axis zero return completion indicator
	Three color indicator









1.1.2 Edit keypad

Press key	Name	Function
	RESET	CNC reset, feed, output stop etc.
	Address key	Address input
		Double address key, switching them by pressing it repetitively
	symbol	Three address key, switching them by pressing it repetitively
	Number key	Number input
	Decimal point	Decimal point input
	Input key	Parameter, compensation value and other data input
	Output key	Communication output
	Change key	Switching message, display

	Edit key	Inserting, altering, deleting programs, fields in EDIT working mode( compound key, switching them by pressing it repetitively)
	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


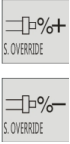

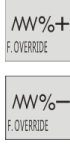





1.1.3 Menu display





















Menu key	Remark
	To enter POS interface. There are RELATIVE POS, ABSOLUTE POS, INTEGRATED POS, POS&PRG windows in this interface.
	To enter PRG interface. There are PRG CONTENT, PRG LIST, PRG STATE windows in this interface.
	To enter TOOL OFFSET, MACRO interface (switching between interfaces by pressing it repeatedly). OFFSET interface displays offset values; MACRO for CNC macro variables.
	To enter ALARM interface. There are ALARM, WARN LOG windows in this interface.
	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
	To enter BIT PARAMETER, DATA PARAMETER, SCREW-PITCH COMP interfaces (switching between each interface by pressing it repeatedly).
	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.
	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 KY-980Tc machine panel are defined by PLC program (ladder), the detailed function meanings are referred to machine manufacturer manual.

The functions of this KY-980Tc 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
  	Spindle control keys	For spindle CCW For spindle stop For spindle CW	Machine zero return, Manual, Step, MPG, Program zero return
	Rapid traverse key	For rapid traverse /feedrate switching	Auto, MDI, Manual
	X feed key	Positive/negative movement of each axis in Manual, Step mode	Machine zero return, Step, Manual, Program zero return mode
			
	Z feed key		
			
	Y feed key		
			
	Cs feed key		
	MPG axis selection key	Each axis selection in MPG mode	MPG mode
   	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 Block switch	For switching of block/blocks execution, Single block indicator lights up if Single mode is active	Auto, MDI
	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 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
	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 KY-980Tc, 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

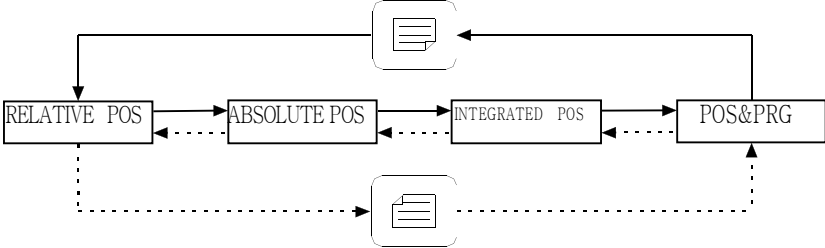
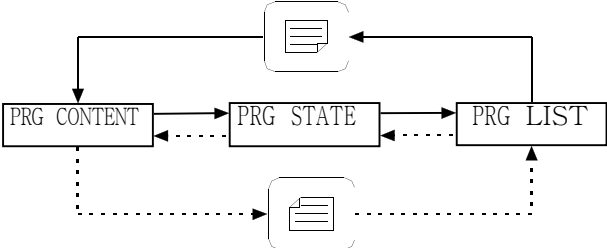
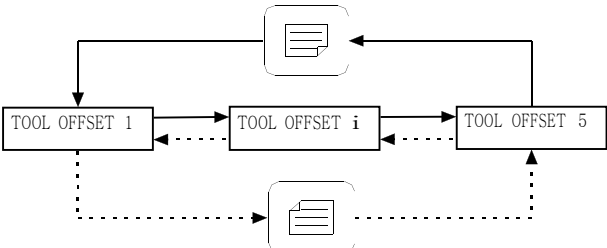
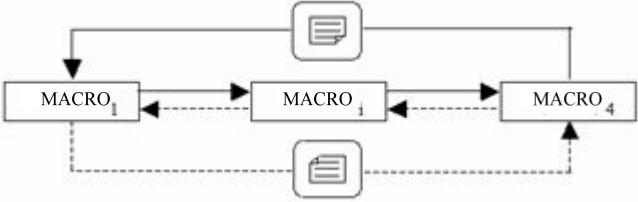
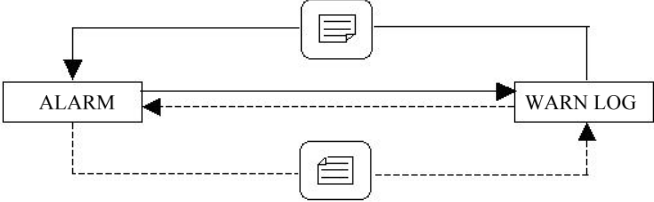
tool change can be performed.

● Program zero return mode

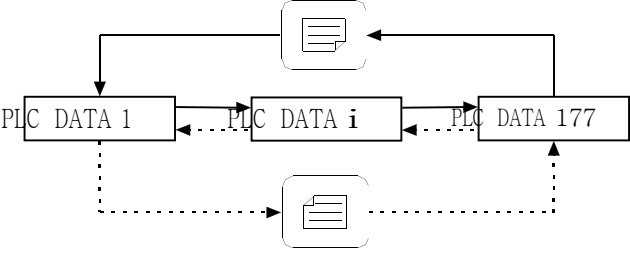



In this mode, the operation of X, Z program zero return can be performed separately.

1.3 Display interface



KY-980Tc 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	
PARAMETER	BIT PARAMETER	
	DATA PARAMETER	
	SCREW-PITCH COMP	
DIAGNOSIS	CNC DIAGNOSIS	
	PLC SIGNAL	

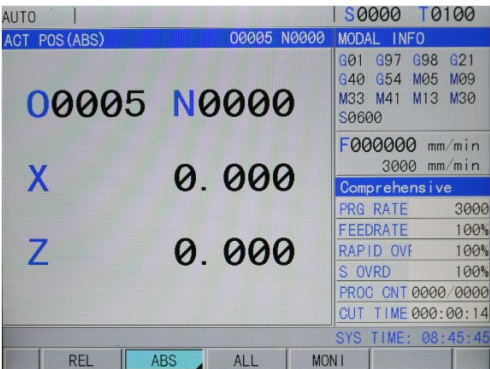
Menu key	Display interface	Display window
	PLC VALUE	
	TOOL PANEL	
	VERSION MESSAGE	
GRAPH	GRAPH	

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  key then press  key.



CUT TIME clearing: Press  key then press  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  key till the “U” in the window blinks, press  key to clear U coordinate;

In RELATIVE POS window, press and hold  key till the “W” in the window blinks, press  key to clear W coordinate.

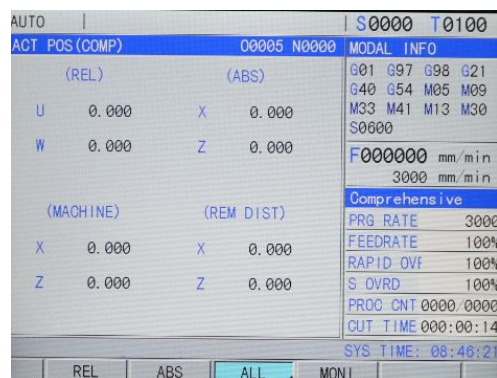
3) INTEGRATED POS display interface

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  in POSITION window and switch the cutting time, and the system time at the bottom right corner below:

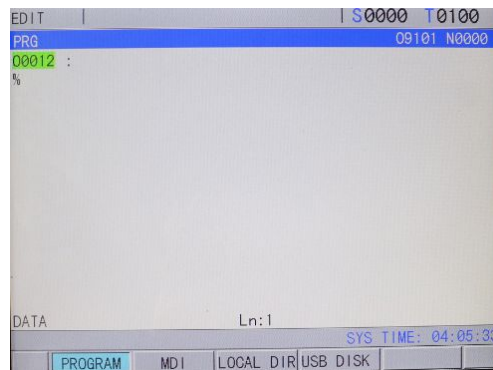


1.3.2 PRG interface


Press  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.

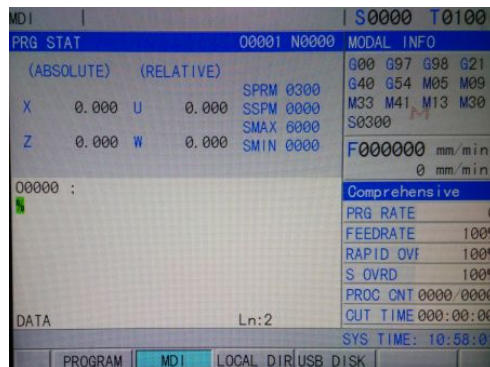
1) PRG CONTENT window

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.

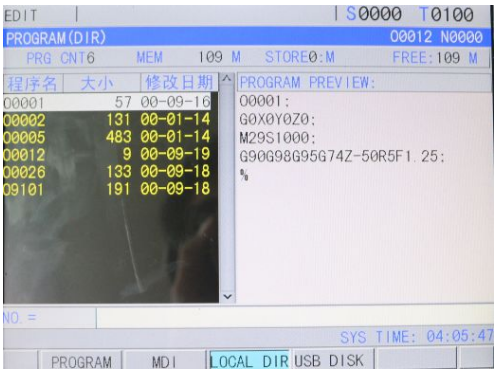


3) PRG LIST window

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

 is a compound key, press  key once in other window, it enters the TOOL OFFSET window, press  key again, it enters the MACRO interface.



If Bit0 of bit parameter No.002 is 1,  key is pressed again, it enters the TOOL-LIFE MANAGEMENT interface.

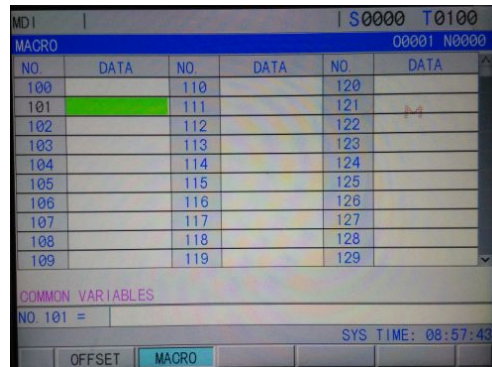
1. TOOL OFFSET&WEAR interface

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

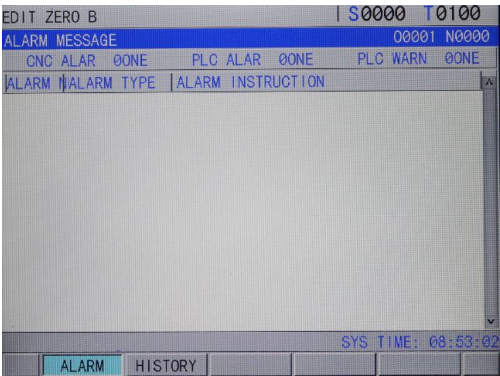
There are 20 windows in this interface, which can be shown by pressing  or  key. In Macro window there are 600 (No.100 ~ No.199 and No.500 ~ No.999) macro variables which can be specified by macro command or set by keypad.



1.3.4 ALARM interface




1) Alarm:

Press  key to enter alarm interface, which can be viewed by  or  key, the window is as follows:

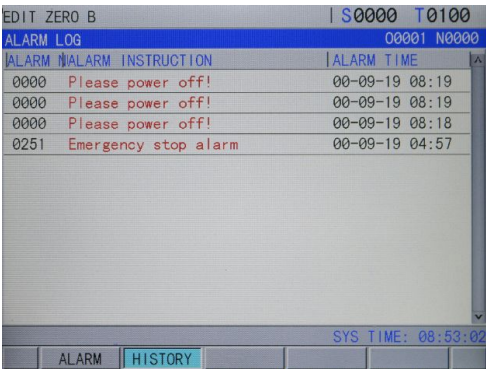


Note: Alarm clearing: It may clear alarms by pressing  (it should press  and  keys together to clear No.100 alarm). If the current alarm window is as follows:

2) Alarm log:




Press  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  +  in 2-level password to clear all log message.

1.3.5 Setting interface

 is a compound key, press  key in other window, it enters the SETTING interface, press it again, it enters the GRAPHIC interface. Press  key repeatedly, it switches between

SETTING and GRAPHIC interfaces.

1. SETTING interface

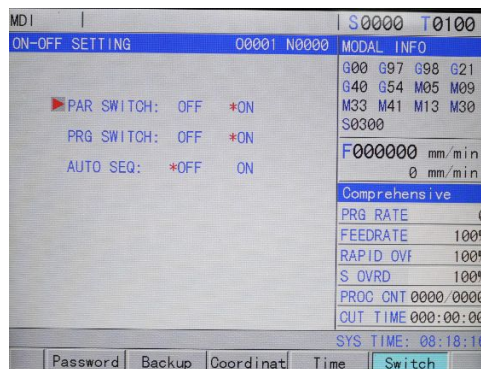
There are 5 windows in this interface, which can be viewed by  or  key.

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.

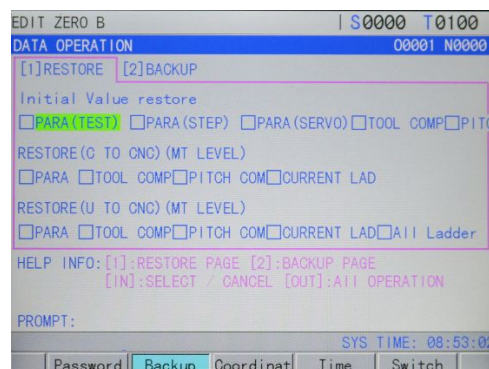
Restore original value: Restore parameters, tool compensation, screw-pitch to system default value.

C disk data restore to CNC: Restore data files which are backup to system disk to system

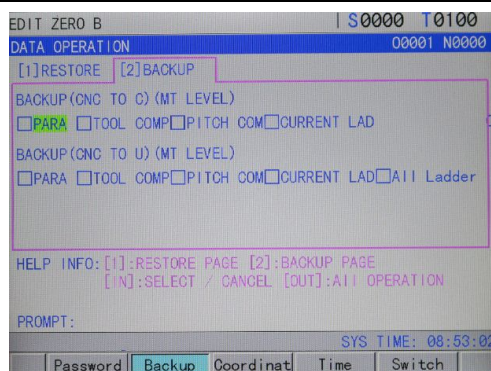
U disk data restore to CNC: Restore data files which are backup to U disk to system

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

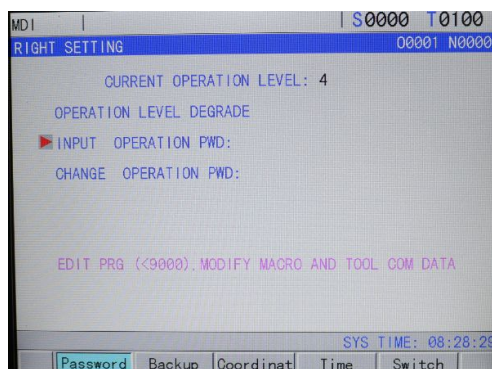
By descending sequence the password of KY-980Tc is classified for 4 levels, which are machine builder (2) level, equipment management (3) level, technician (4) level, machining operation (5) level.

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.

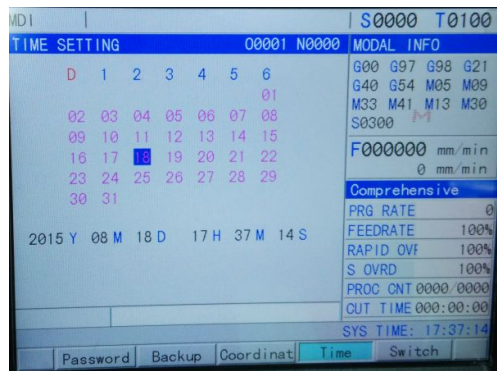
Operation level: No password. Only the machine panel operation is allowed, the operations of part program edit and selection, the alteration operations of CNC bit parameter, data parameter, screw-pitch parameter, tool offset data are forbidden.



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.

STATUS PARAMETER		00001 N0000	
NO.	DATA	NO.	DATA
001	00011000	011	00000000
002	01000010	012	00100001
003	00110111	013	11000000
004	01000000	014	00000000
005	00110001	164	00000000
006	00100000	168	10001101
007	10000001	172	00010000
008	00011111	173	00000000
009	10011111	174	10001000
010	00001000	175	00000000

**** SPTV SOHW **** INI
BIT0: (0: In mm 1: In inches) input
NO.001 =
SYS TIME: 17:53:31

BITPAR NUMPAR PITCH COM 用户参数

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.

NUM PARAMETER		00001 N0000	
NO.	DATA	NO.	DATA
015	1	025	100
016	1	026	100
017	1	027	8000
018	1	028	50
019	5	029	60
020	0	030	0
021	0.0000	031	1260
022	3800	032	400
023	5000	033	40
024	100	034	0.0000

X rapid movement speed(radius)
NO.022 =
SYS TIME: 08:22:54

BITPAR NUMPAR PITCH COM 用户参数

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.

PITCH COMP			00001 N0000		
NO.	X	Z	NO.	X	Z
000	12	30	011	0	0
001	0	0	012	0	0
002	0	0	013	0	0
003	0	0	014	0	0
004	0	0	015	0	0
005	0	0	016	0	0
006	0	0	017	0	0
007	0	0	018	0	0
008	0	0	019	0	0
009	0	0	020	0	0
010	0	0	021	0	0

NO.000 =
SYS TIME: 08:23:43

BITPAR NUMPAR PITCH COM 用户参数

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.

MDI ZERO B				S0000 T0100	
CNC DGN				00001 N0000	
NO.	DATA	NO.	DATA	NO.	DATA
000	00000000	010	00000000	020	00000000
001	00000000	011	00000000	021	00000000
002	00000000	012	00000000	022	00000000
003	00000000	013	00000000	023	00000000
004	00011111	014	00000000	024	00000000
005	00000000	015	00000000	025	00000000
006	00000001	016	00000000	026	00000000
007	00000000	017	00000000	027	00000000
008	00000000	018	00000000	028	00000000
009	00011111	019	00000000	029	00000000
BIT0:					
NO. 000					
SYS TIME: 08:53:02					
CNC DGN		PLC SIGNAL IO DIAGNO		VERSION 1 CNC HELP	

2) PLC SIGNAL 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

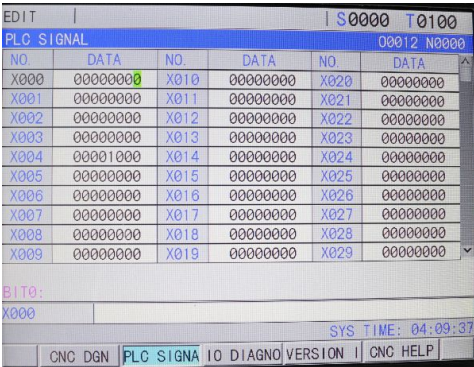


key.

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

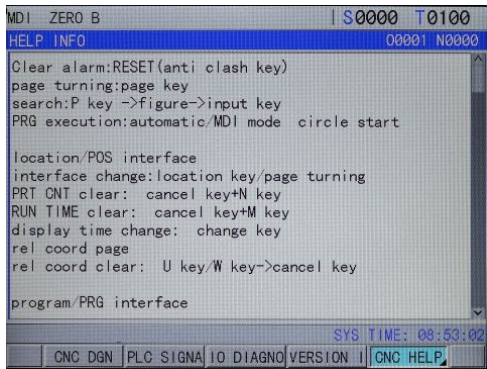


or  key. The 1st row shows the abbreviation of the bit address number where the cursor locates.




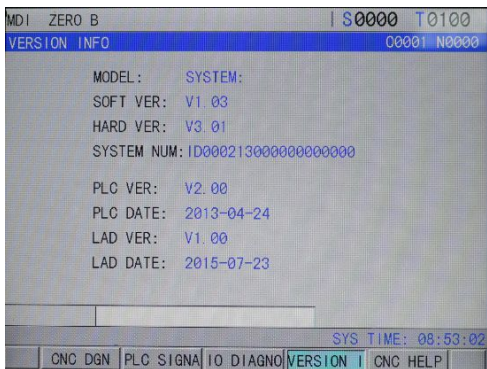
3)HELP MESSAGE interface

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:



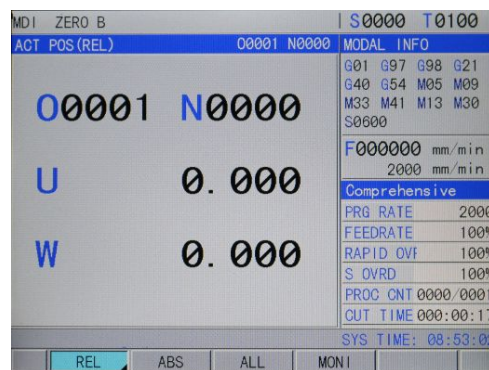
CHAPTER 2 POWER ON/OFF AND PROTECTION

2.1 System power on

Before KY-980Tc 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 KY980Tc automatic detection and initiation are finished.



2.2 System power off

Before power is off, ensure that:

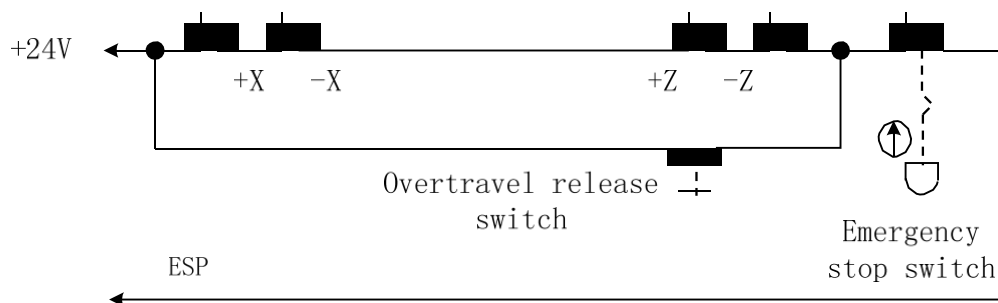
1. The feed axes of the CNC is at stop;
2. Miscellaneous functions (spindle, cooling etc.) are OFF;
3. Cut off CNC power prior to machine power cutting off.

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 KY-980Tc stop, and the emergency alarm is issued.



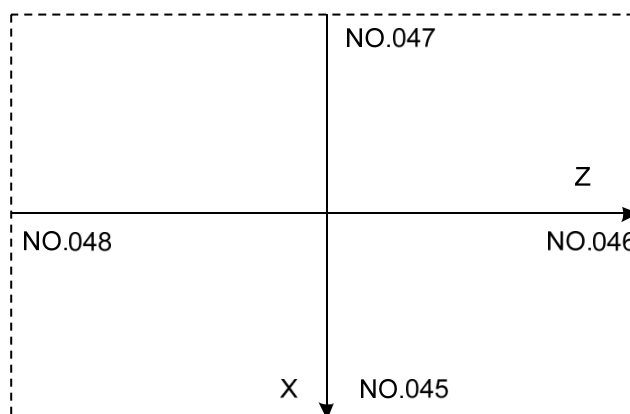
When the hardware overtravel occurs, there will be an “emergency stop” alarm in KY-980Tc. The steps to eliminate this alarm are: press the OVERTRAVEL button to switch to the ALARM window, view the alarm message, and reset the alarm and move the table reversely to detach the stroke switch (for positive 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.


The software stroke is set by data parameter No.194, No.197, which is referred to the machine coordinates.

2.4 Emergency operation

During the machining, some unexpected incidents may occur because of the user programming, operation and product fault etc. So this KY-980Tc 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 KY-980Tc 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 KY-980Tc 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 the KY-980Tc standard PLC programs!

















Press  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


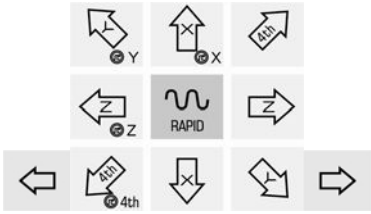






Press and hold X axis feed and direction key  or  in the selection area



  
  
    , 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;



In Manual mode, press  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  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  or , and the axis moving stops if the key is released; press and hold Z axis feed and direction key  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  or , Y can rapidly move negatively or positively, and Y moving stops if the key is released;

In Manual rapid mode, press  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 KY-980Tc system.

Note 2: In Edit/MPG mode,  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

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.



〰️%
R. OVERRIDE



In the manual rapid traverse, it can press or 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 KY-980Tc 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 overridden.



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.

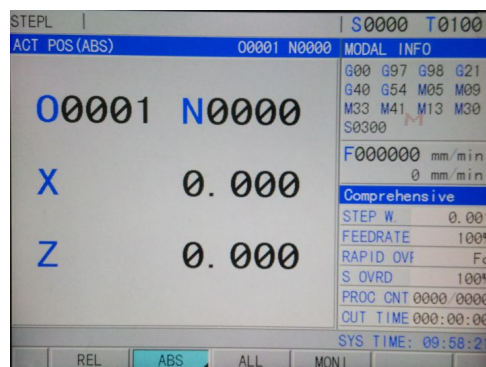
Note!

The key functions of this KY-980Tc 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 KY-980Tc standard PLC program!

4.1 Step feed






Set the system parameter No.001 Bit3 to 0, and press  key to enter the STEP working mode, it displays as follows:



4.1.1 Increment selection







Press     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   is inactive; when the BIT7


is 0,     are all active. For example, to press  key, the window is shown as follows:

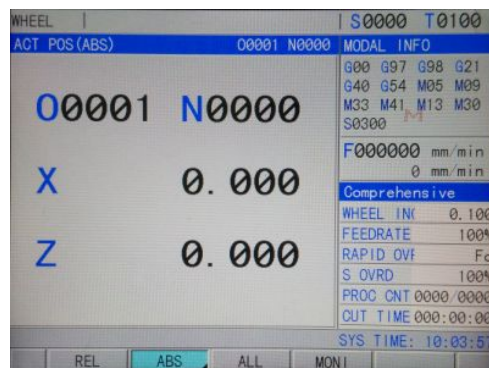


4.1.2 Moving direction selection

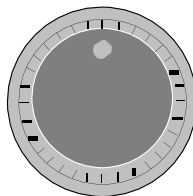
Pressing  or  key once can move X negatively or positively by a step increment;
 pressing  or  key once can move Z negatively or positively by a step increment;
 pressing  or  key once can move Y negatively or positively by a step increment.

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:

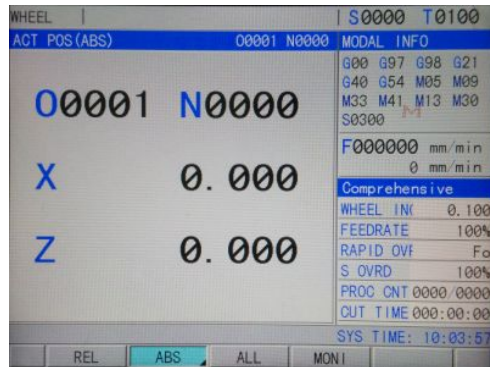


The handwheel (MPG) figure

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 , , ,  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



: In Manual/Step mode, the spindle rotates counterclockwise if pressing this key;



: In Manual/Step mode, the spindle stops if pressing this key;



: 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 overridden.



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:

MPG increment	Moving amount of each MPG scale			
	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 KY-980Tc 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 KY-980Tc 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  key to enter MDI mode;

2) Press  key ( or  or press  many times if needed) to enter PRG STATE window:



3) press address key , number key ,  ;


4) press address key , number key , ,  ;

5) press address key , number key ,  ;

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  and the display is as follows:



After the code words are input and  is pressed to display the block,  key is pressed



to execute the input block. During the execution, ,  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,  is pressed to cancel the display or  is pressed to clear all the input, then re-input the correct ones.

5.5 Other operations

1. 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

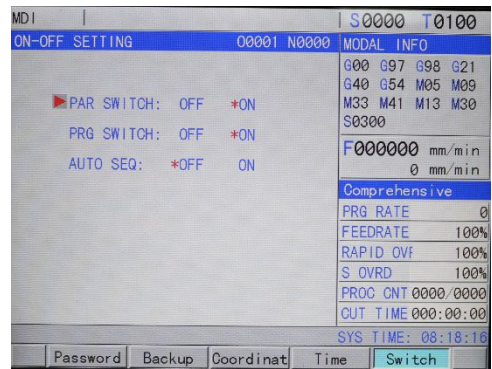
In Edit mode, a program can be created, selected, altered, copied and deleted, and the bidirectional communication of CNC to CNC, or CNC to PC can also be done. To prevent the program to be altered or deleted accidentally, a program switch is set up for this KY-980Tc system. And it must be turned on before program editing. Please see details in Section 10.1.1 of this part. Also 3 level user passwords are set in this KY-980Tc 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.

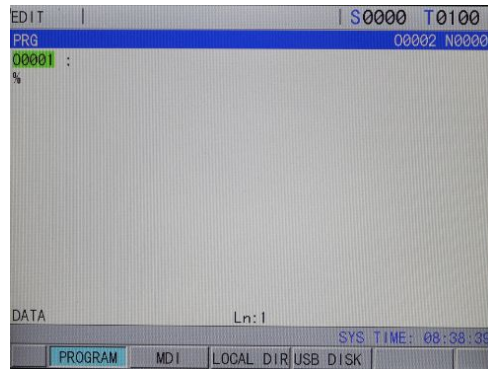
When the “AUTO SEG” switch in SWITCH SETTING window is ON, CNC generates the block number automatically, it automatically generates the next block number by pressing  key in editing. The block number increment is set by the CNC data parameter No.042. (See details in **Section 10.1.1** of this part.)









6.1.2 Inputting a program

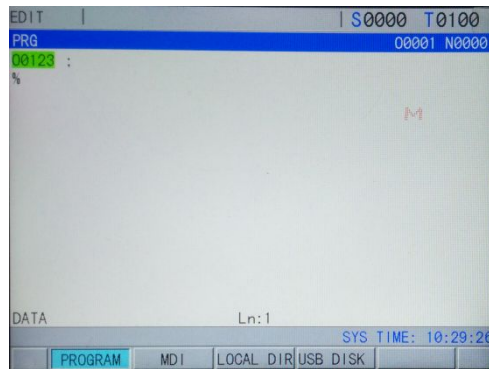
1. Press  key to enter the Edit mode;


Press  key to enter the Program interface, select the PRG CONTENT window by pressing  or  key;



2. Press address key , number key , ,  and  key by sequence (e.g. program O0001 creation);

3. Press  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  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  key to enter the Edit mode, then press  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

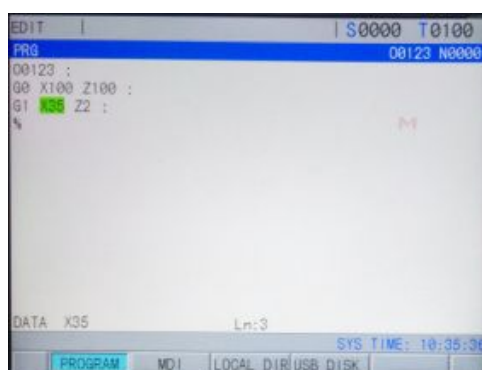
The steps of finding is as follows:



1) Press  key to enter Edit mode;

2) Press  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  key to enter Edit mode;

2) Press  key to enter the PRG CONTENT window;



3) Input the line No. to be searched

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   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  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   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  key,

And then press  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  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  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  key first, then press , 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  key, the blocks among cursor and inputted character will be copied


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  key, the segment among cursor and inputted character will be copied

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  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 , number key , , ,  by sequence (by program O0001);
- 3) Press  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 , symbol key , number key , , ,  ;
- 3) Press  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  key to enter the PRG CONTENT window;
- 3) Press address key  and key in the program No.;
- 4) Press  or  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.

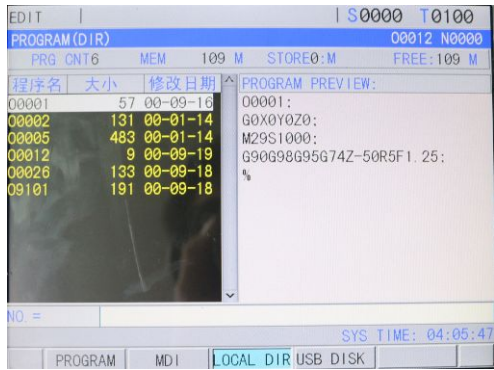
Note: In step 4, if the program does not exist in Edit mode, a new program will be created by CNC system after  key is pressed.





6.3.2 Scanning

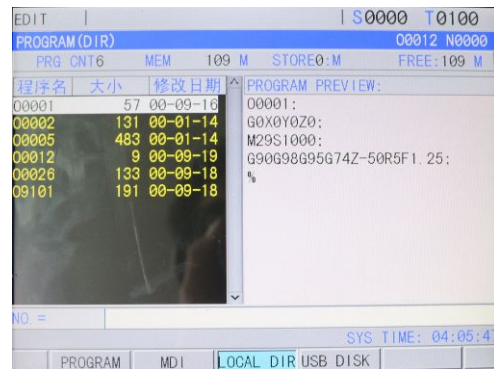
- 1) Select Edit or Auto mode;
- 2) Press  key to enter the Program window;
- 3) Press address key ;
- 4) Press  or  key to display the next or previous program;
- 5) Repeat step 3 and 4 to display the saved programs one by one.

6.3.3 Cursor

- 1) Select Auto mode (**must be in non-running state**);
- 2) Press  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  key or .

6.4 Renaming a program

- 1) Select the PRG CONTENT window in Edit mode;
- 2) Press address key , and key in the new program name;
- 3) Press   key.


6.5 Copy a program

To save the current program to a location:

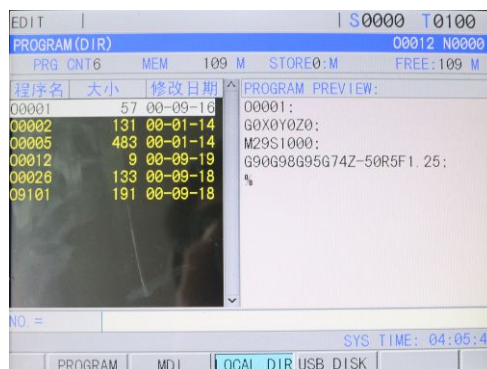
- 1) Select the PRG CONTENT window in Edit mode;
- 2) Press address key , and key in the new program No.;
- 3) Press  key.

6.6 Program management



6.6.1 Program list

In non-Edit mode, press  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 :  + No. + 

In Edit mode, it will creat program if program No. is not exit.

2) Delete program

1. In Edit mode, press  to delete program specified by cursor

2. In Edit mode, press  + No. +  or No. + 

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 KY-980Tc system. The tool offset data are obtained from this tool setting operation.

7.1 Tool positioning setting

Steps:

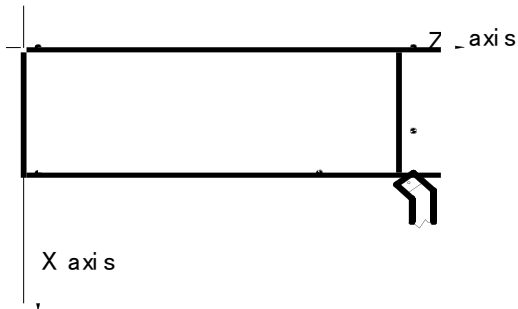


Fig. 7-1

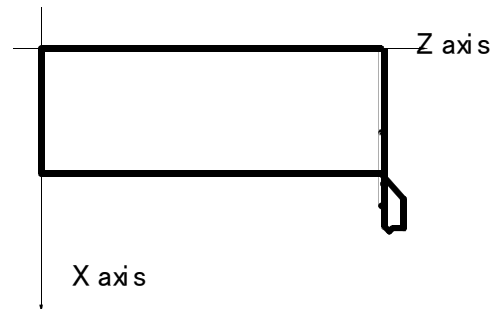








Fig. 7-2

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 **OFFSET** key and move the cursor by ,  key to select the corresponding offset number of that tool;
9. Press address key , then press **DATA INPUT** key to input the tool offset value of X axis into the corresponding offset number;
10. Press address key , then press **DATA INPUT** 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.

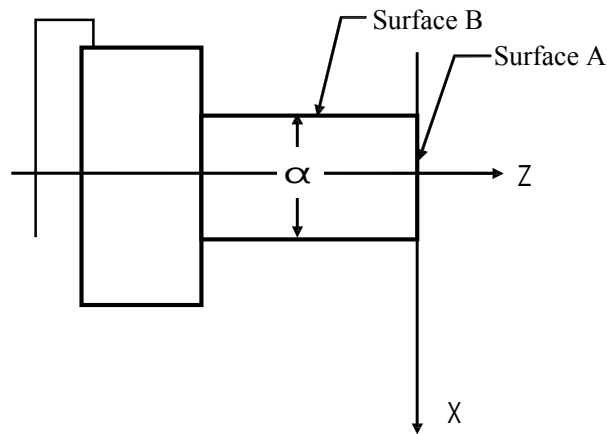
Note: The original system tool offset should be cleared in positioning tool setting, multiple but one input of

the new offset value by ,  keys are disabled, about the clearing ways, refer to Volume II Operation, Chapter 7.4.4.

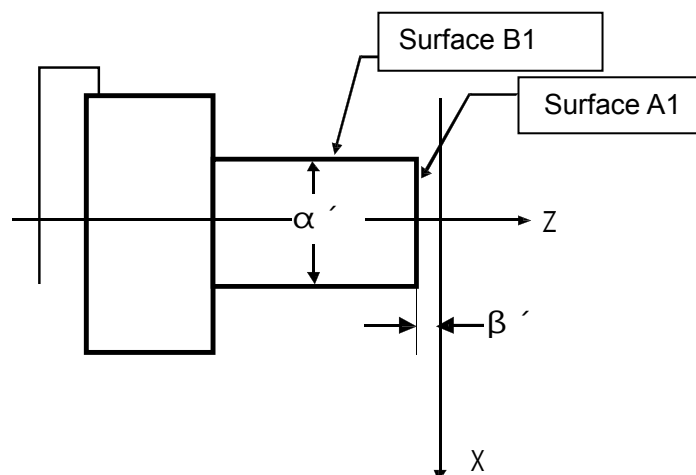
7.2 Trial tool setting

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 $\alpha=15$);
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 $\beta' = 1$);

14. Press  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 , sign key , number key , and 

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 $\alpha' = 10$);

19. Press  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 , number key ,  and  key;

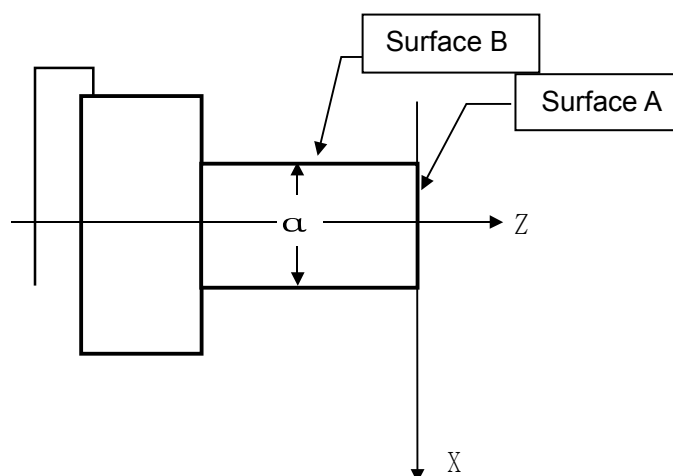
21. Repeat the execution from Step 10 to Step 20 to perform the tool setting operation for other tools.


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

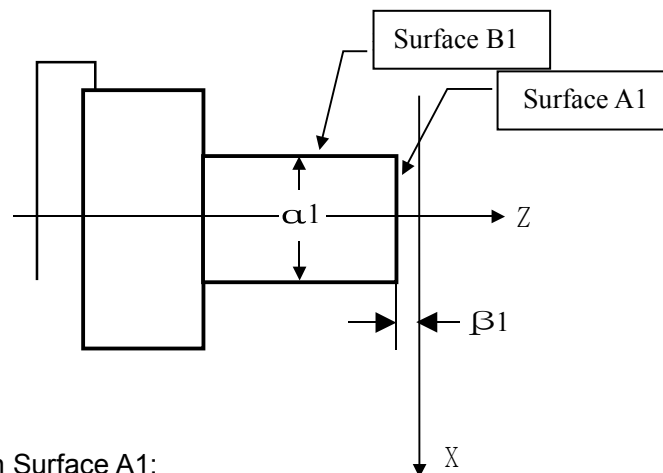
There is no reference tool in this tool setting methods, when the tool is worn or to be adjusted, it only needs to be set again, and a machine zero return should be done before the tool setting. The machining could be continued by performing a machine zero return at power on after power-off, which is very convenient for the operation.

Steps (workpiece coordinate system by part end surface):



1. Press  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 $\alpha=15$);
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 " $\beta 1$ " between the Surface A1 and the workpiece coordinate system origin (supposing $\beta 1=1$);
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 " $\alpha 1$ " (supposing $\alpha 1=10$);
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;

22. Key in by sequence the address key **X**, number key **1**, **□** 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;

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  + offset number + 

3. After pressing the address key  or , the numerical number may be keyed in (decimal point allowed)


4. By pressing the  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 ; as for that of Z axis, key in ;

3. Then key in the incremental value;

4. Press the  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  key, then press  key, this offset will be cleared;

If the offset value of Z axis is to be cleared, press  key, then press  key, this offset will be cleared;

3. Method (2)

If the current offset in X axis is α , input U- α , then press  key, this offset in X axis will be cleared;

If the current offset in Z axis is β , input W- β , then press  key, this offset in Z axis will be cleared.

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 KY-980Tc 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.

NO	类型	X	Z	R	T
00	偏置	0.000	0.000	0.000	0
01	偏置	0.000	0.000	0.000	0
02	偏置	0.000	0.000	0.000	0
03	偏置	0.000	0.000	0.000	0
04	偏置	0.000	0.000	0.000	0
05	偏置	0.000	0.000	0.000	0

RELATIVE
U M 0.000
W 0.000

ABSOLUTE
X 0.000
Z 0.000

000FFSET=

SYS TIME: 11:10:57

Before inputting No.0 tool offset

NO	类型	X	Z	R	T
00	偏置	100.000	100.000	0.000	0
01	偏置	0.000	0.000	0.000	0
02	偏置	0.000	0.000	0.000	0
03	偏置	0.000	0.000	0.000	0
04	偏置	0.000	0.000	0.000	0
05	偏置	0.000	0.000	0.000	0

RELATIVE
U M 0.000
W 0.000

ABSOLUTE
X 100.000
Z 100.000

000FFSET=

SYS TIME: 11:12:45

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 KY-980Tc 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 KY-980Tc 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  key to enter the PRG CONTENT window;
- 3) Press the address key , and key in the program No.;
- 4) Press  or  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  key to enter the PRG CONTENT window;
- 3) Press the address key ;
- 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  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  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);



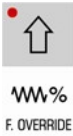
2. If the mode (G, M, T, F code) of the current block where the cursor locates is defaulted and inconsistent with the running mode of this block, the corresponding modal function should be executed to continue next step.


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

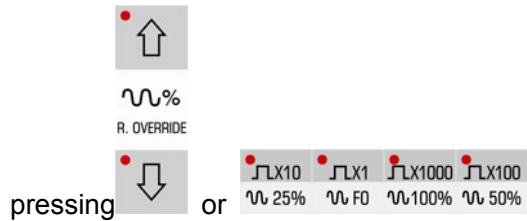


Press , 16-level real time feedrate can be obtained.

- 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





Note 1: The rapid traverse speeds of every axis are set by the CNC parameter No.090~No.094 respectively;

Actual rapid traverse rate = value set by parameter × rapid override

Note 2: When the rapid override is F0, the min. rapid traverse rate is set by CNC parameter No.085.

Note 3: For the spinning machine, only after return zero finish can run program automatically ;

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 KY-980Tc 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 KY-980Tc 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 KY-980Tc standard PLC program!

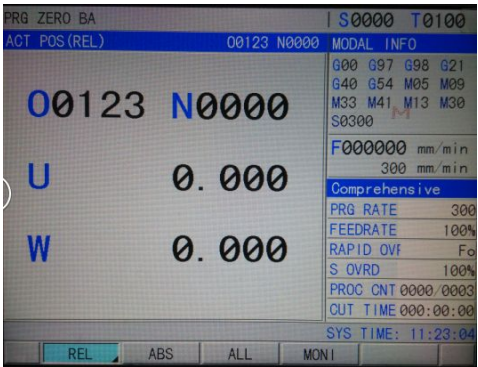
9.1 Program zero return

9.1.1 Program Zero

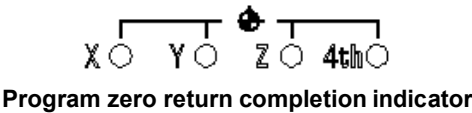
While the part is fixed on the machine, absolute coordinate of current tool position may be set by G50 code according to the relative position between the tool and the part, so a workpiece coordinate system is setup. The tool current position is called **program zero**, and this is the program zero return 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.



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)

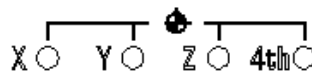
The **machine coordinate system** is a reference coordinate system for CNC coordinate operation. It is an inherent coordinate system of the machine. The origin of the machine coordinate system is called machine zero (or mechanical reference point). It is defined by the zero or zero return switch fixed 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

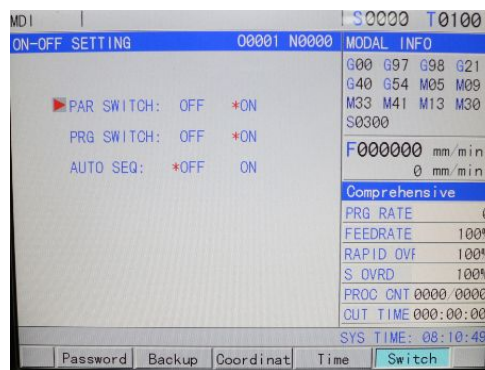
1. Press  key, the spindle rotates counterclockwise;
2. Press  key, the spindle stops;
3. Press  key, the spindle rotates clockwise;
4. Press  key, the cooling is switched ON or OFF;
5. Lubricating control(refer to OPERATION, Chapter 3);
6. Press  key, the tool change is executed;
7. Tune the spindle 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:

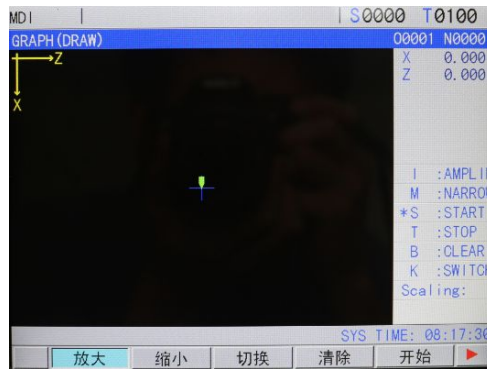


1. Press key to enter the Setting interface, then press or key to enter the SWITCH SETTING window;
2. Press or key to move the cursor to the item to be set;
3. Press and key to shift the ON-OFF state: press key, "*" moves to the left to set the switch for OFF, press key, "*" moves to the right to set 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 , 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 key to enter the path window






Graphic parameter meaning

1. Coordinate system setting: 8 types of graphic paths can be displayed in this KY-980Tc 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:

MDI		S0000 T0100	
STATUS		PARAMETER	
NO.		DATA	
001	00011000	011	00000000
002	01000010	012	01100001
003	00110111	013	11000000
004	01000000	014	00000000
005	00110001	164	00000000
006	00100000	168	10001101
007	10000001	172	00010000
008	00011111	173	00000000
009	10011111	174	10001000
010	00001000	175	00000000





**** SPTV SOHW **** INI
BIT0: (0: in mm 1: in inches) input
NO.001 =
SYS TIME: 11:39:04



BITPAR NUNPAR PITCH COM 用户参数


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 , key in parameter No., then press  key.

- 4) Key in the new parameter value;
- 5) Press  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:

MDI		S0000 T0100	
STATUS		PARAMETER	
NO.		DATA	
001	00011000	011	00000000
002	01000010	012	01100001
003	00110111	013	11000000
004	01000000	014	00000000
005	00110001	164	00000000
006	00100000	168	10001101
007	10000001	172	00010000
008	00011111	173	00000000
009	10011111	174	10001000
010	00001000	175	00000000

**** RDRN DECI ORC **** SOW
BIT0: (0: mm 1: inc) output
NO.004 = 01100000
SYS TIME: 11:43:28

BITPAR NUNPAR PITCH COM 用户参数

Press  key to finish the parameter alteration. The window is shown as follows:

STATUS		PARAMETER		00123 N0000	
NO	DATA	NO	DATA	NO	DATA
001	00011000	011	00000000	176	00000111
002	01000010	012	01100001	177	10000001
003	00110111	013	11000000	178	00000000
004	01100000	014	00000000	179	00000000
005	00110001	164	00000000	180	01000010
006	00100000	168	10001101	181	00000000
007	10000001	172	00010000	182	00000000
008	00011111	173	00000000	183	00000000
009	10011111	174	10001000	184	00000001
010	00001000	175	00000000	185	00000000

**** RDRN DECI ORC **** SOW
 BIT0: (0: mm 1: inc) output
 NO.004 =
 SYS TIME: 11:46:08
 BITPAR NUMPAR PITCH COM 用户参数

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:

MDI | S0000 T0100

STATUS PARAMETER 00123 N0000					
NO.	DATA	NO.	DATA	NO.	DATA
001	00011000	011	00000000	176	00000111
002	01000010	012	01100001	177	10000001
003	00110111	013	11000000	178	00000000
004	01000000	014	00000000	179	00000000
005	00110001	164	00000000	180	01000010
006	00100000	168	10001101	181	00000000
007	10000001	172	00010000	182	00000000
008	00011111	173	00000000	183	00000000
009	10011111	174	10001000	184	00000001
010	00001000	175	00000000	185	00000000

**** RDRN DECI ORC **** ***** SCW
 BIT7:Retention
 NO.004 =
 SYS TIME: 11:52:02
 BITPAR NUMPAR PITCH.COM 用户参数

Move the cursor to “BIT5” by pressing  or  key as follows:

MDI | S0000 T0100

STATUS PARAMETER 00123 N0000					
NO.	DATA	NO.	DATA	NO.	DATA
001	00011000	011	00000000	176	00000111
002	01000010	012	01100001	177	10000001
003	00110111	013	11000000	178	00000000
004	01000000	014	00000000	179	00000000
005	00110001	164	00000000	180	01000010
006	00100000	168	10001101	181	00000000
007	10000001	172	00010000	182	00000000
008	00011111	173	00000000	183	00000000
009	10011111	174	10001000	184	00000001
010	00001000	175	00000000	185	00000000

**** RDRN DECI ORC **** ***** SCW
 BIT5:signal(0:low level,1:high level)
 NO.004 =
 SYS TIME: 11:50:46
 BITPAR NUMPAR PITCH.COM 用户参数


Input “1” to finish the alteration.

MDI | S0000 T0100

STATUS PARAMETER 00123 N0000					
NO.	DATA	NO.	DATA	NO.	DATA
001	00011000	011	00000000	176	00000111
002	01000010	012	01100001	177	10000001
003	00110111	013	11000000	178	00000000
004	01000000	014	00000000	179	00000000
005	00110001	164	00000000	180	01000010
006	00100000	168	10001101	181	00000000
007	10000001	172	00010000	182	00000000
008	00011111	173	00000000	183	00000000
009	10011111	174	10001000	184	00000001
010	00001000	175	00000000	185	00000000

**** RDRN DECI ORC **** ***** SCW
 BIT5:signal(0:low level,1:high level)
 NO.004 =
 SYS TIME: 11:51:05
 BITPAR NUMPAR PITCH.COM 用户参数

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.

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:

MDI S0000 T0100					
NUM PARAMETER 00123 N0000					
NO.	DATA	NO.	DATA	NO.	DATA
015	1	025	100	035	10.0000
016	1	026	100	036	0
017	1	027	8000	037	6000
018	1	028	50	038	6000
019	5	029	60	039	6000
020	0	030	10	040	6000
021	0.0000	031	1260	041	40
022	3800	032	400	042	10
023	5000	033	40	043	100
024	100	034	0.0000	044	115200

X rapid movement speed(radius)
NO.022 = 3800
SYS TIME: 15:19:33
BITPAR NUMPAR PITCH COM 用户参数

Press  key to finish the alteration. The window is shown as follows:

MDI S0000 T0100					
NUM PARAMETER 00001 N0000					
NO.	DATA	NO.	DATA	NO.	DATA
015	1	025	100	035	0.0000
016	1	026	100	036	0
017	1	027	8000	037	6000
018	1	028	50	038	6000
019	5	029	60	039	6000
020	0	030	0	040	6000
021	0.0000	031	1260	041	40
022	3800	032	400	042	10
023	5000	033	40	043	100
024	100	034	0.0000	044	115200

X rapid movement speed(radius)
NO.022 =
SYS TIME: 08:22:54
BITPAR NUMPAR PITCH COM 用户参数


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  key to finish the alteration.

MDI S0000 T0100					
PITCH COMP 00001 N0000					
NO.	X	Z	NO.	X	Z
000	12	0	011	0	0
001	0	0	012	0	0
002	0	0	013	0	0
003	0	0	014	0	0
004	0	0	015	0	0
005	0	0	016	0	0
006	0	0	017	0	0
007	0	0	018	0	0
008	0	0	019	0	0
009	0	0	020	0	0
010	0	0	021	0	0

NO.000 =
SYS TIME: 08:23:24
BITPAR NUMPAR PITCH COM 用户参数

The same as above, key in “Z30” by sequence in the cue line, press  key to finish the alteration. The window is as follows:



NO.	X	Z	NO.	X	Z
000	12	30	011	0	0
001	0	0	012	0	0
002	0	0	013	0	0
003	0	0	014	0	0
004	0	0	015	0	0
005	0	0	016	0	0
006	0	0	017	0	0
007	0	0	018	0	0
008	0	0	019	0	0
009	0	0	020	0	0
010	0	0	021	0	0

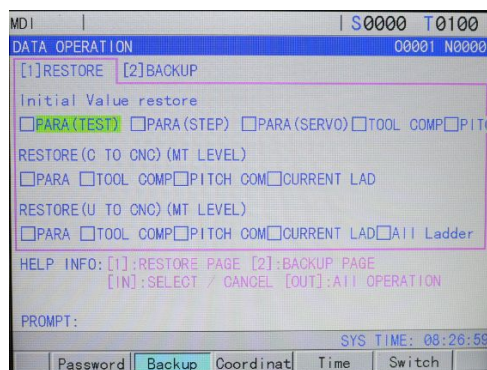
NO. 000 =

SYS TIME: 08:23:43

BITPAR NUMPAR PITCH COM 用户参数

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 KY-980Tc 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:



MDI | S0000 T0100

DATA OPERATION 00001 N0000

[1]RESTORE [2]BACKUP

Initial Value restore

☐ PARA(TEST) ☐ PARA(STEP) ☐ PARA(SERVO) ☐ TOOL COMP ☐ PITCH

RESTORE(C TO CNC) (MT LEVEL)

☐ PARA ☐ TOOL COMP ☐ PITCH COM ☐ CURRENT LAD

RESTORE(U TO CNC) (MT LEVEL)

☐ PARA ☐ TOOL COMP ☐ PITCH COM ☐ CURRENT LAD ☐ All Ladder








HELP INFO: [1]:RESTORE PAGE [2]:BACKUP PAGE

[IN]:SELECT / CANCEL [OUT]:ALL OPERATION

PROMPT:

SYS TIME: 08:26:59

Password Backup Coordinat Time Switch

1. Turn on the parameter switch;
2. Press  key to enter the MDI mode, then press  key ( or  key if necessary) to enter OPERATE DATA window;
3. Press  key to enter Backup DATA Window, press  to enter RECOVERY DATA Window
4. Move cursor to operated option, press  key to select / cancel 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

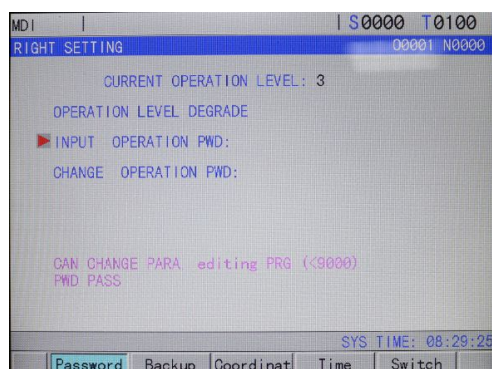
To protect the part programs, CNC parameters from malignant alteration, this KY-980Tc provides password setting function that is graded for 4 levels. By descending sequence, they are machine builder (2) level, equipment management (3) level, technician (4) level, machining operation (5) level. The current 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

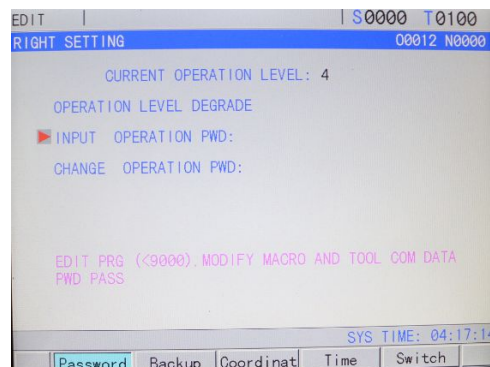
- 1 Move the cursor to the “INPUT PASSWORD:” row after the system enters the PASSWORD SETTING window;
- 2 Input the password (an “*” sign added each time inputting a character);
- 3 Press  key to finish the inputting, and the system enters the corresponding password level.


Note: The length of KY-980Tc password corresponds to the operation level, which can't be added or reduced by user at will. The detailed is as follows:

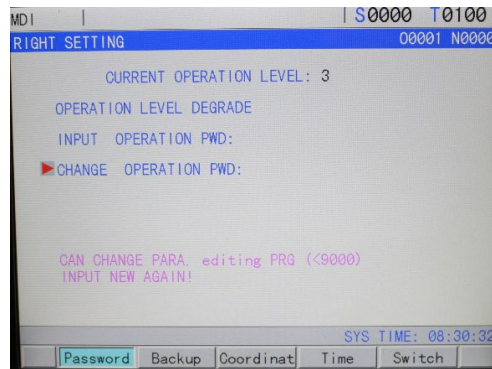
Operation level	Password length	Initial password
3	5 bytes	12345
4	4 bytes	1234
5	No	No

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:

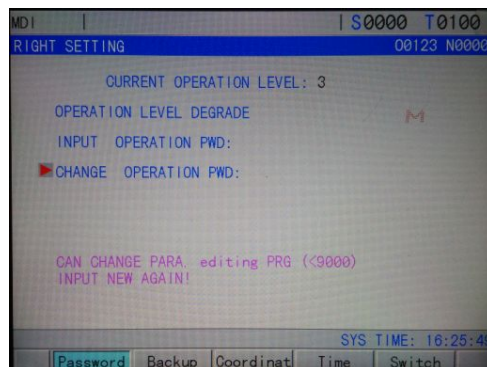


Note: When current operation level is lower than or equal to the 3 level (3, 4, 5 level), this level is not changed if the CNC system is turned on again. If previous level is the 2 level, it defaults the 3 level when the system is turned on again.

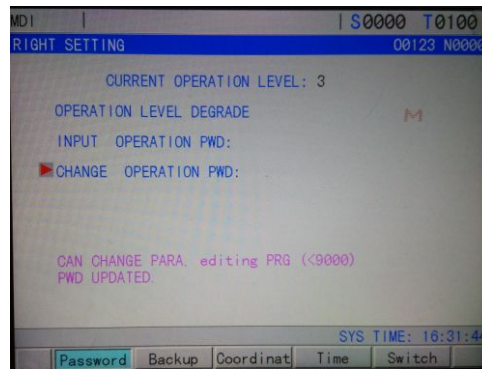
10.3.2 Altering the password

Steps for password alteration:

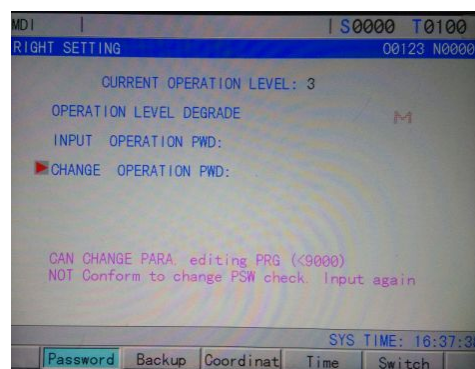
- 1 After entering the PASSWORD SETTING window, enter the password by the methods in Section 10.3.1;
- 2 Move the cursor to the “ALTER PASSWORD:” row;
- 3 Key in the new password, then press  key;
- 4 The CNC system prompts “PLEASE INPUT USER PASSWORD AGAIN!”, the window display is as follows:



- 5 After re-inputting the password, press  key, if the passwords input are identical, CNC prompts “PASSWORD UPDATED.”. So the password alteration is successful.



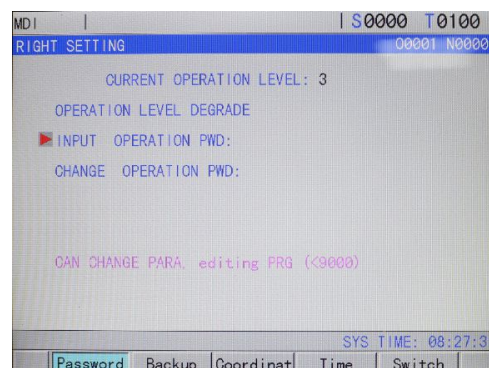
6 If the inputs of the passwords are not identical, CNC prompts “PASSWORD CHECKOUT ERROR.”, the window is as follows:




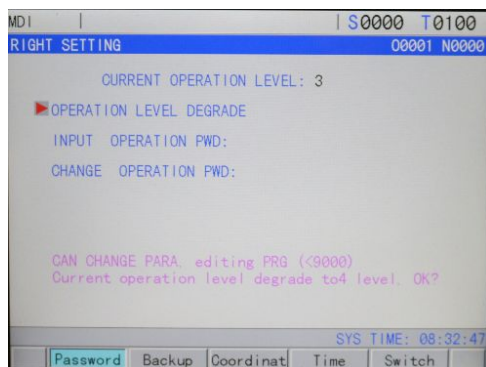
10.3.3 Setting the lower password level


The demotion of the operation level is used to enter a lower level from a higher level, the steps are as follows:

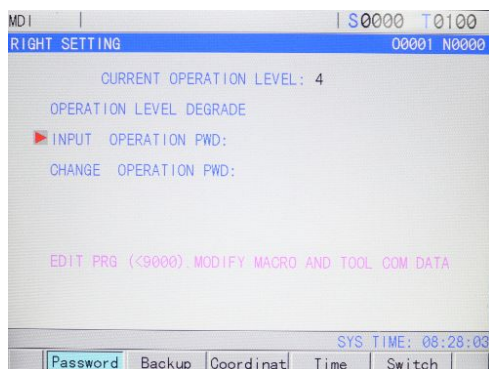
- 1 After entering the PASSWORD SETTING window, key in the password by the method in Section 10.3.1;
- 2 Move the cursor to the “SET LOWER LEVEL:” row, if the current CNC operation is the 3 level, the window is as follows:



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:

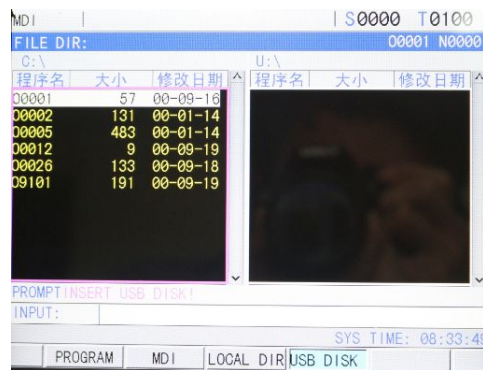


Note: If the current level is the 5 level, the demotion operation is forbidden.


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 $\Phi 136\text{mm} \times 180\text{ mm}$, as follows:

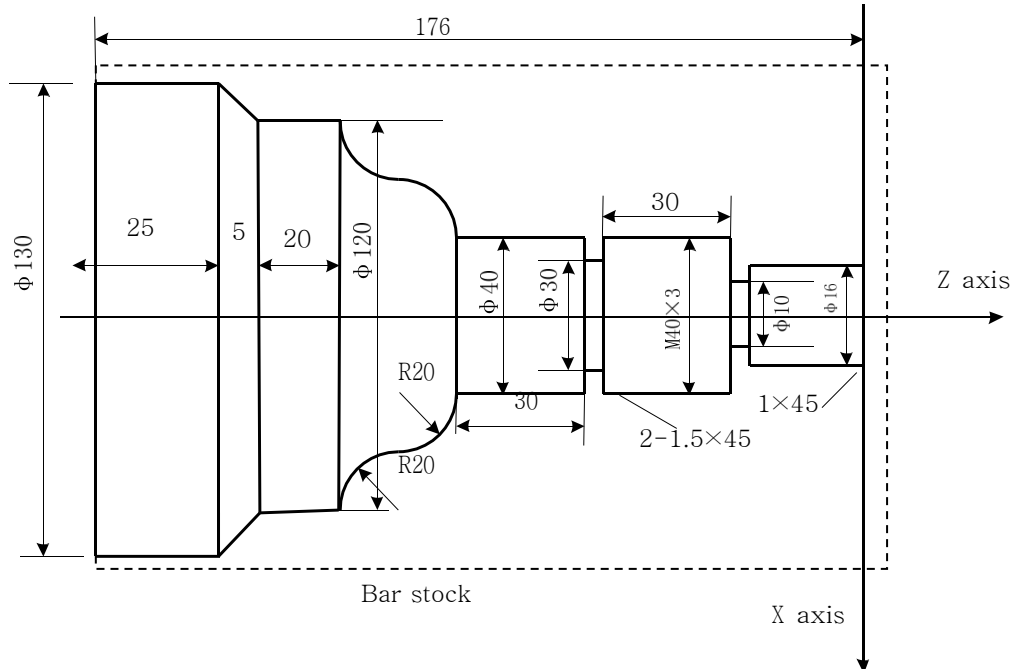


Fig. 14-1

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:

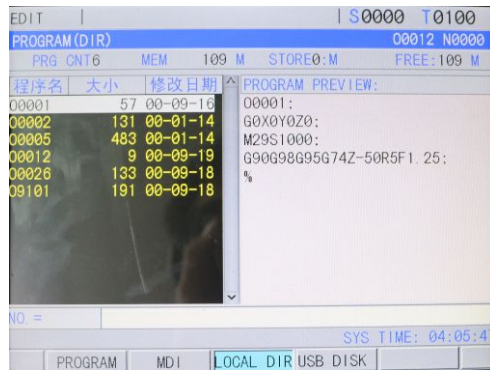
O 0 0 0 1 ;		Name of the part program
N 0 0 0 0	G0 X150 Z50 ;	Position to the safety height for tool change
N 0 0 0 5	M12 ;	Clamp the chuck
N 0 0 1 0	M3 S800 ;	Start the spindle with speed 800
N 0 0 2 0	M8 ;	Turn on the cooling N 0
0 3 0	T0101 ;	Change for the No. 1 tool
N 0 0 4 0	G0 X136 Z2 ;	Approach the part
N 0 0 5 0	G71 U0.5 R0.5 F200 ;	Cut depth 1mm and retract 1mm
N 0 0 5 5	G71 P0060 Q0150 U0.25 W0.5 ;	0.5mm pre-reserved in X axis, 0.5mm machining allowance in Z axis
N 0 0 6 0	G0 X16 ;	Approach to the end face of the part
N 0 0 7 0	G1 Z-23 ;	Cut the $\Phi 16$ outer circle N
0 0 8 0	X39.98 ;	Cut the end face N 0 0 9
0	W-33 ;	Cut the $\Phi 39.98$ outer circle
N 0 1 0 0	X40 ;	Cut the end face N 0 1 0
5	W-30 ;	Cut the $\Phi 40$ outer circle N
0 1 1 0	G3 X80 W-20 R20 ;	Cut the convex arc N 0 1
2 0	G2 X120 W-20 R20 ;	Cut the concave arc N 0 1
3 0	G1 W-20 ;	Cut the $\Phi 120$ outer circle N
0 1 4 0	G1 X130 W-5 ;	Cut the cone
N 0 1 5 0	G1 W-25 ;	Cut the $\Phi 130$ outer circle
N 0 1 6 0	G0 X150 Z185 ;	Rough cut end and back to the tool change point
N 0 1 7 0	T0202 ;	Change for the No.2 tool and execute its offset
N 0 1 8 0	G70 P0060 Q0150 ;	Fine cut cycle
N 0 1 9 0	G0 X150 Z185 ;	Rough cut end and back to the tool change point
N 0 2 0 0	T0303 ;	Change for the No.3 tool and execute its offset
N 0 2 1 0	G0 Z-56 X42 ;	Approach to the part

N 0 2 2 0	G1 X30 F100 ;	Cut the $\Phi 30$ groove
N 0 2 3 0	G1 X37 F300 ;	Return
N 0 2 4 0	G1 X40 W1.5 ;	Chamfering
N 0 2 5 0	G0 X42 W30 ;	Keep the width of the grooving
N 0 2 6 0	G1 X40 ;	
N 0 2 6 2	G1 X37 W1.5 ;	
N 0 2 6 4	G1 X10 ;	Chamfering
N 0 2 6 6	G0 X17 Z-1 ;	Cut the $\Phi 10$ groove
N 0 2 6 8	G1 X16 ;	
N 0 2 7 0	G1 X14 Z0 F200 ;	Chamfering
N 0 2 8 0	G0 X150 Z50 ;	Return to the tool change point
N 0 2 9 0	T0404 S100 ;	Changing for the No. 4 tool and set the spindle speed for 100
N 0 3 0 0	G0 X42 Z-20 ;	Approach the part N 0 3 1 0
	G92 X39 W-34 F3 ;	Thread-cutting cycle N 0 3 2
0	X38 ;	Feed 1mm for the 2 nd cutting N
0 3 2 0	X37 ;	Feed 1mm for the 3rd cutting N
0 3 3 0	X36.4 ;	Feed 0.6mm for the 4th cutting
N 0 3 3 2	X36 ;	Feed 0.4mm for the 5th cutting
N 0 3 4 0	G0 X150 Z50 ;	Return to the tool change point
N 0 3 5 0	T0100 U0 W0 ;	Change for the No.1 tool and execute its offset
N 0 3 6 0	M5 ;	Turn off the spindle
N 0 3 7 0	M9 ;	Turn off cooling N
0 3 8 0	M13 ;	Unclamp the chuck
N 0 3 9 0	M30 ;	Program ends

12.2 Program input

12.2.1 View a saved program

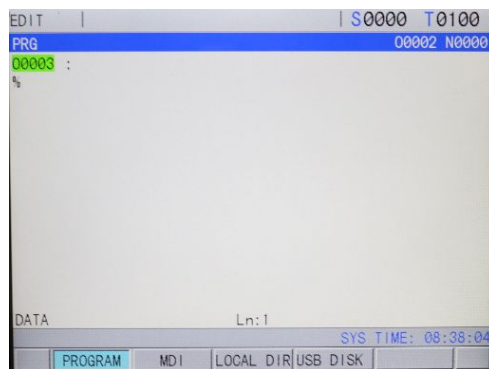
In a non-Edit mode, press  key to enter Program interface, select the PRG LIST window by pressing  or  key, the window is as follows:



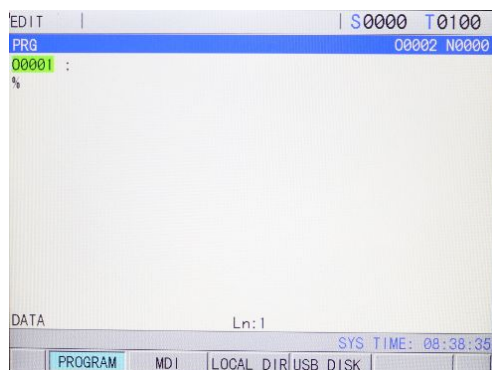
In above window the names of the programs saved can be viewed for renaming the new program.

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:

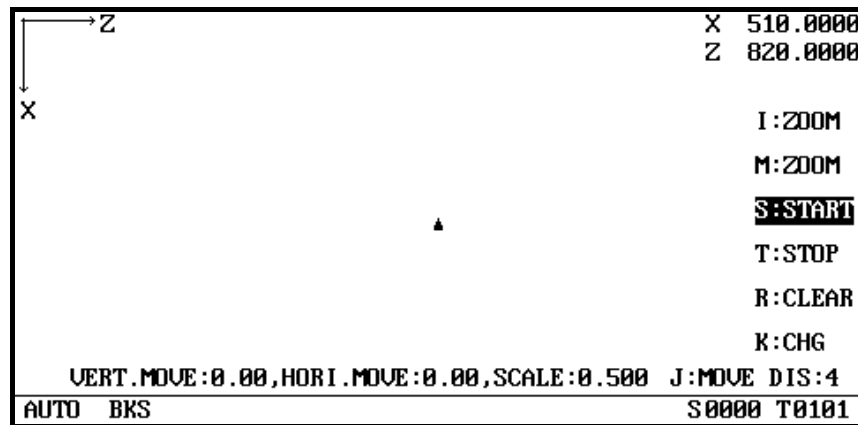


Complete the program editing by inputting the above program word by word.












12.3 Checkout a program




12.3.1 Graphic setting

1. Press  to enter the graphic window as follows:



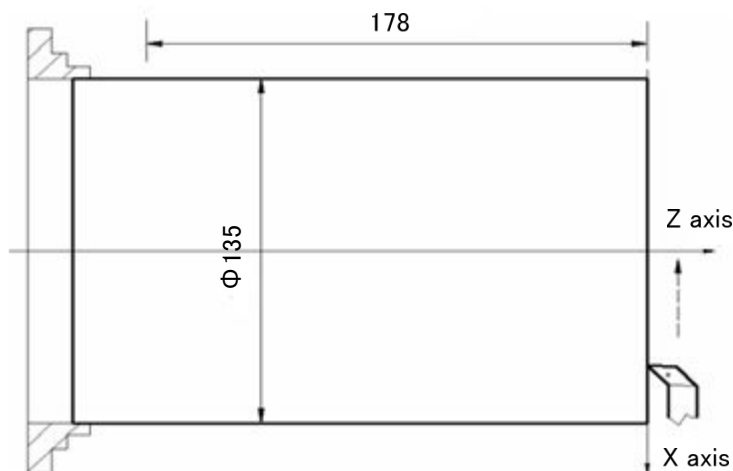
12.3.2 Program check

Press  or  to enter the graph display window, press  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  to start the drawing, press  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:

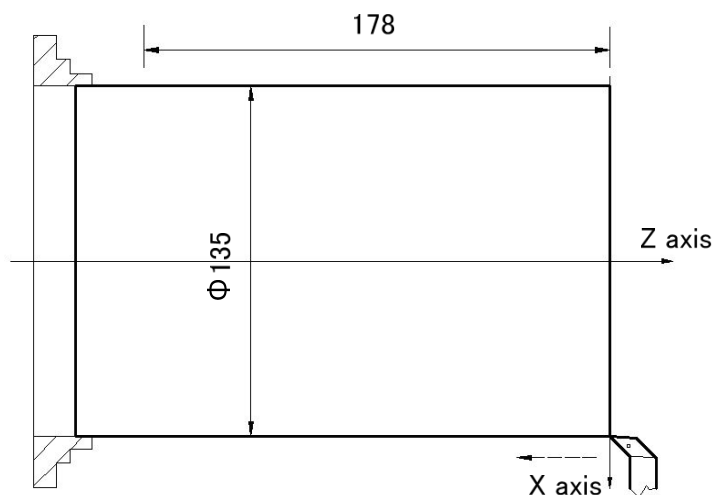
If there is error in the program path, make a diagnosis for the error in the program and modify the program. Then make another checkout for the program by the method above till the error is eliminated. In the Graphic interface, press  key on the panel to start drawing, or press  key to stop drawing, or press  key to clear the drawing.


12.4 Tool setting and running

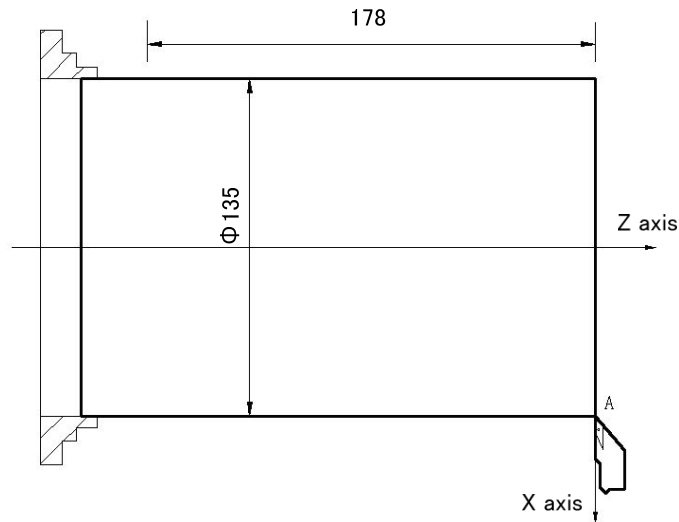
1. Move the tool to a safe position, run the T0100 U0 W0 command in the PRG STATE window of the MDI mode, and cancel the tool offset;
2. Move the tool to cut in the part end surface;




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;



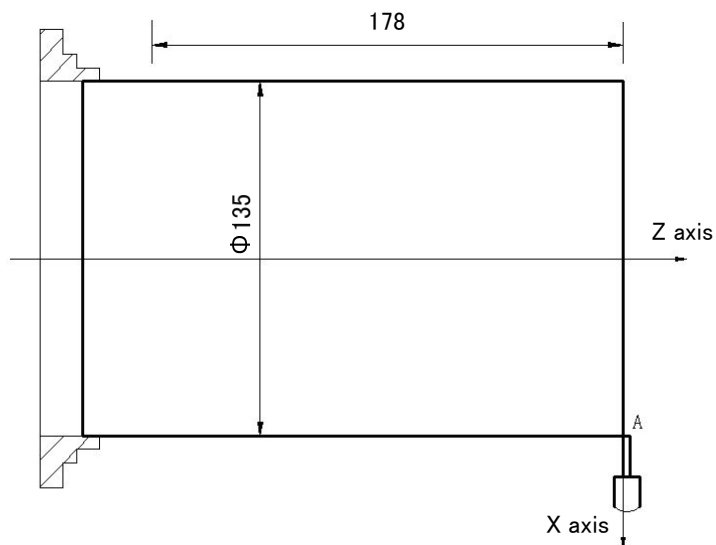
6. Release the tool along Z when X does not move, and stop the spindle, measure the dimensions of the outer circle of the part (e.g. The measuring value is 135mm);
7. Execute G50 X135 command in the PRG STATE window of the MDI mode to set the coordinate of X axis;
8. Switch to the TOOL OFFSET window, 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;

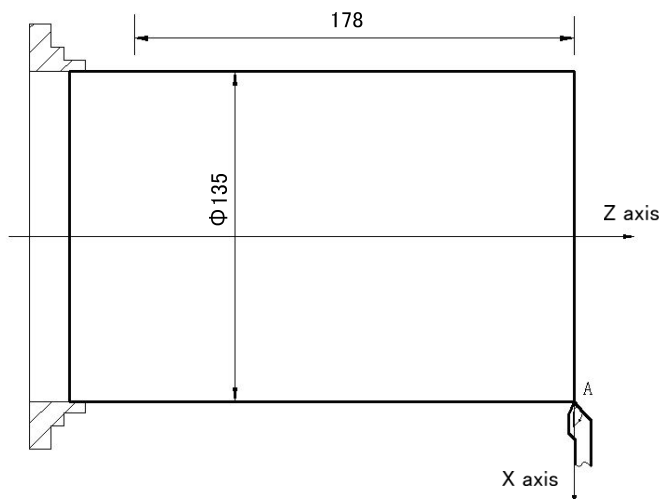
13. Start the spindle and move the tool to the tool setting point, as A point in the following figure;






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 KY980Tc system connection

1.1.1 KY980Tc back cover interface layout

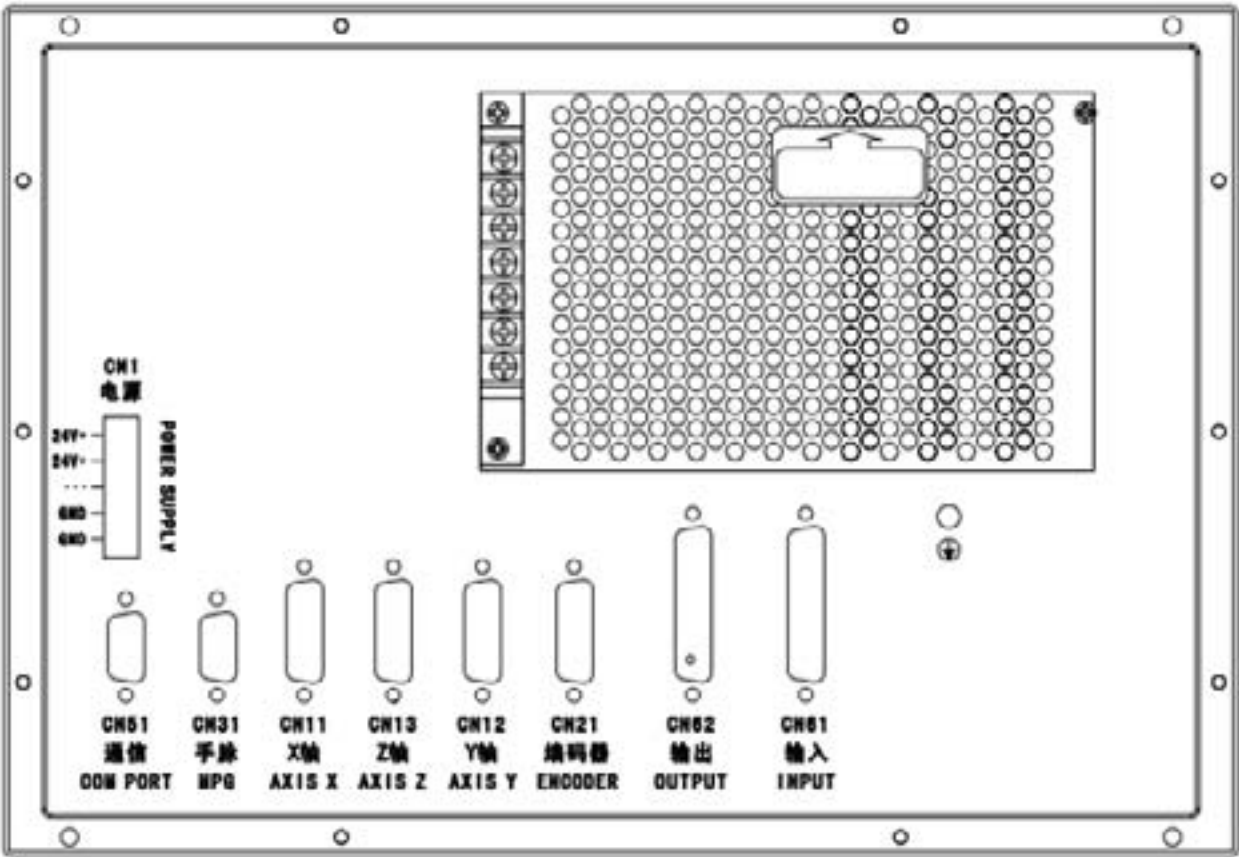


Fig. 1-1 KY980Tc back cover interface layout

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 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
- 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 KY980Tc installation

1.2.1 KY980Tc 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);

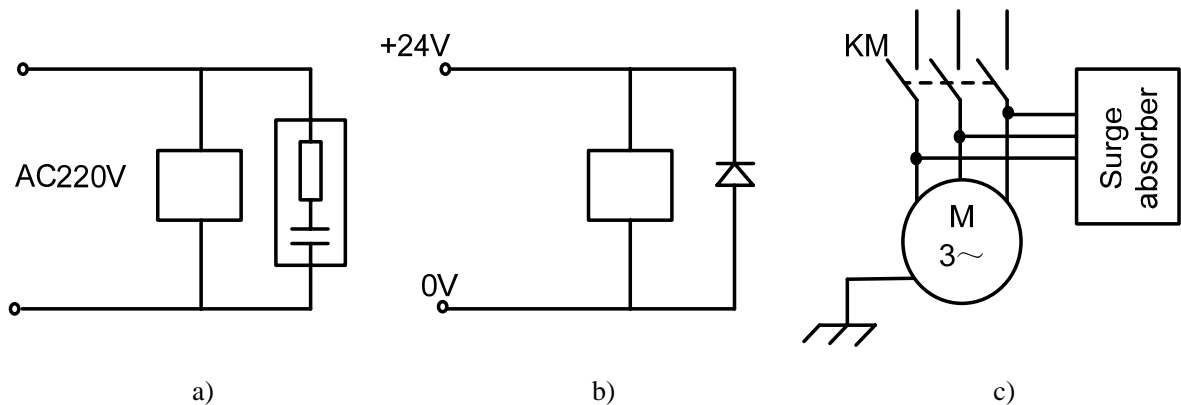


Fig. 1-3

- 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

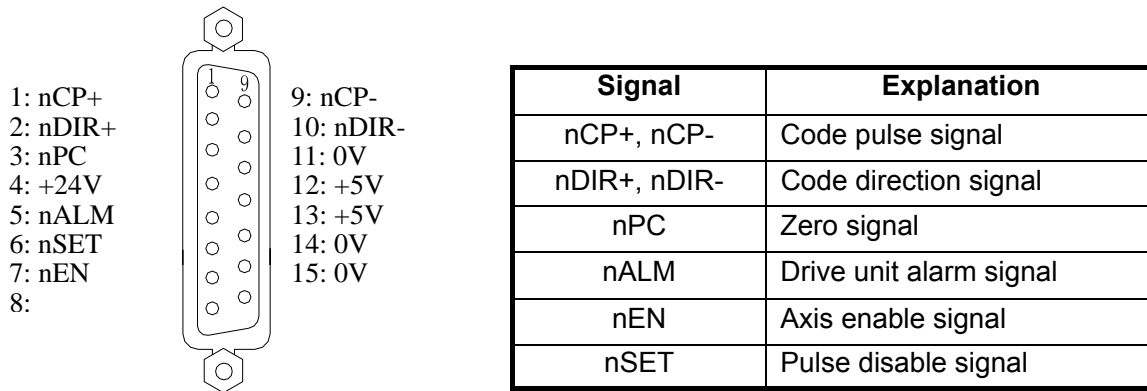


Fig.2-1 CN11, CN12, CN13, CN14 interface
(15-core D type female socket)

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:

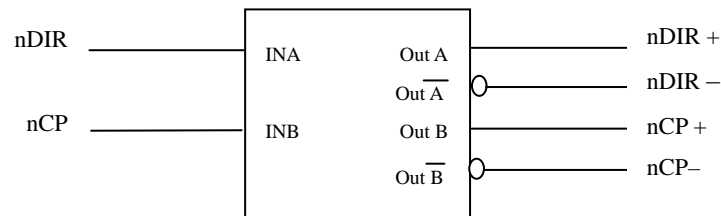


Fig. 2-2 Interior circuit of code pulse and direction signals

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:

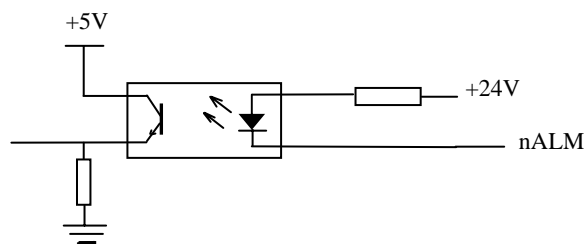


Fig. 2-3 Interior circuit of drive unit alarm signal

This type of input circuit requires that the drive unit transmits signal by the types in Fig. 2-4:

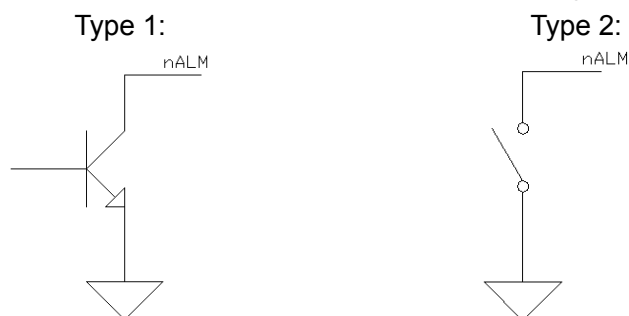


Fig. 2-4 Signal types by drive unit

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:

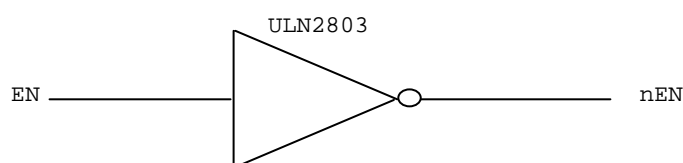


Fig. 2-5 Interior interface circuit for axis enable signal

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:

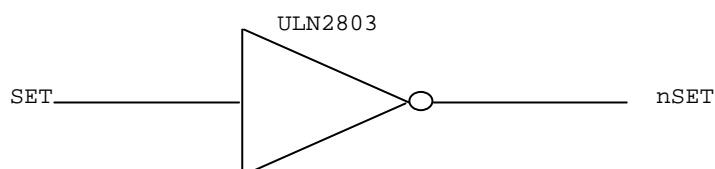


Fig. 2-6 Pulse disable signal circuit

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.

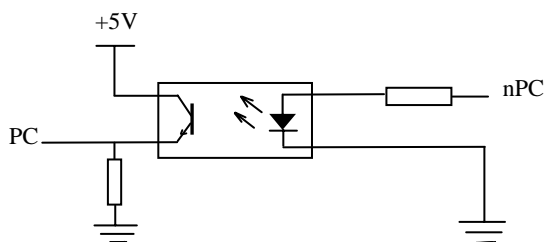


Fig. 2-7 Zero signal circuit

Note: nPC signal uses +24V level.

a) The wave of PC signal by user is shown in Fig. 2-8:

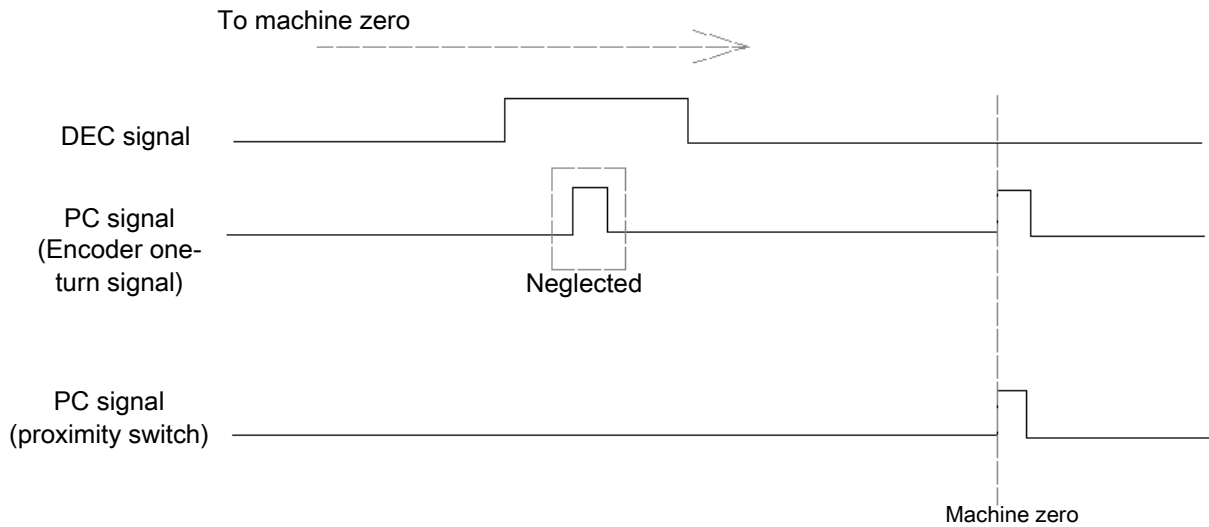


Fig. 2-8 PC signal wave

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:

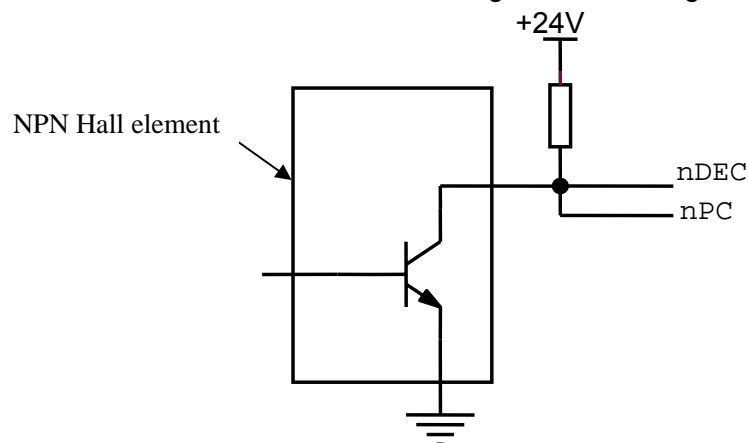


Fig. 2-9 Wiring by a NPN Hall element

c) The wiring of PNP Hall elements taken as both DEC signal and zero signal is shown in Fig. 2-10:

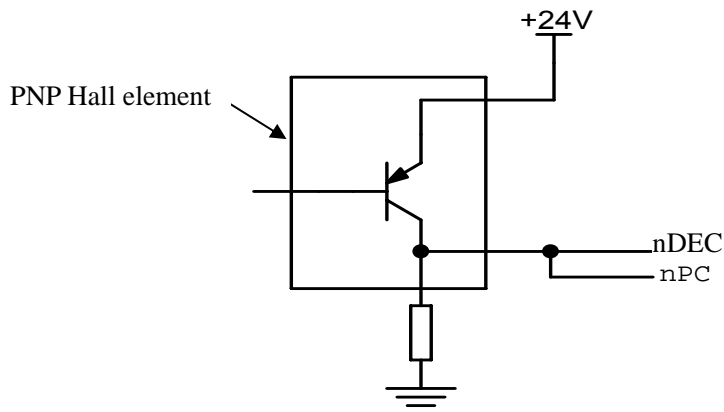
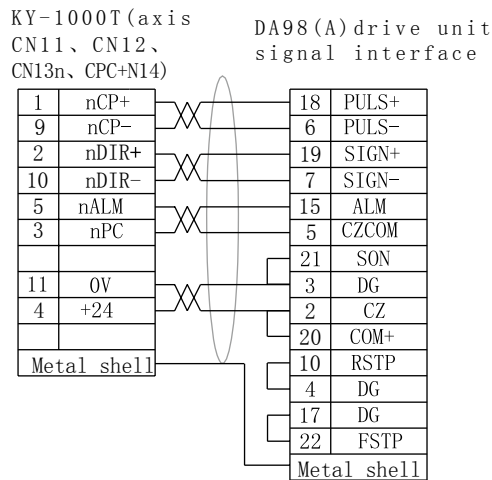


Fig. 2-10 Wiring by a PNP Hall element

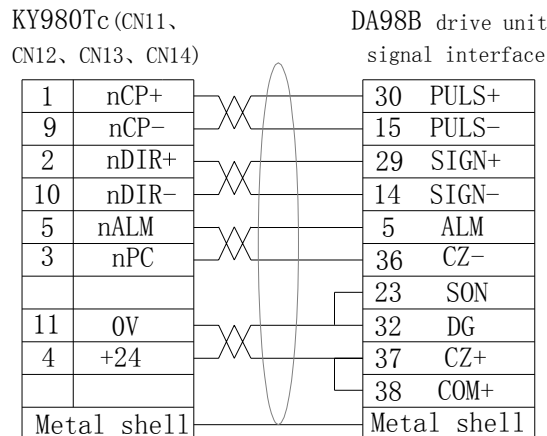
2.1.7 Connection to a drive unit

KY980Tc is connected with our drive unit shown in Fig. 2-11:

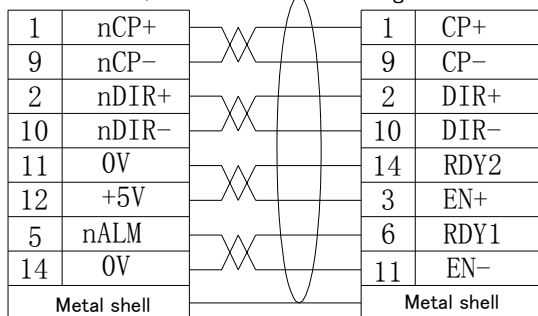
KY980Tc is connected with DA98(A)drive unit



KY980Tc is connected with DA98B drive unit



KY-1000T (axis
CN11, CN12,
CN13n, CPC+N14)



KY-1000T (axis
CN11, CN12,
CN13n, CPC+N14)

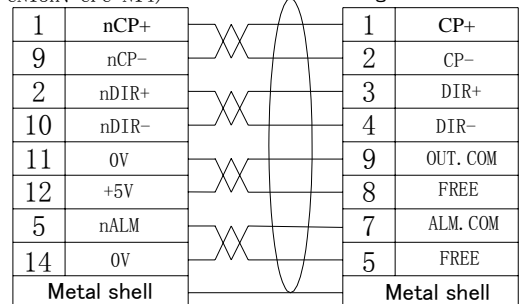


Fig. 2-11 KY980Tc CNC systems are connected with KY drive unit

2.2 Being connected with spindle encoder

2.2.1 Spindle encoder interface definition

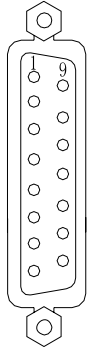


Fig.2-12 CN21 encoder interface
(15-core D type male socket)

NO.	Name	Explanation
1,10,11,14,15	GND	Spindle 0V
2	nALM_M	Spindle alarm signal
3	#PCS	Encoder C phase - pulse
4	PCS	Encoder C phase + pulse
5	#PBS	Encoder B phase - pulse
6	PBS	Encoder B phase + pulse
7	#PAS	Encoder A phase - pulse
8	PAS	Encoder A phase + pulse
9	SVC_OUT	Analog voltage
12	TH5IO5V	5V
13	TH5IO5	5V

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 KY980Tc 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)

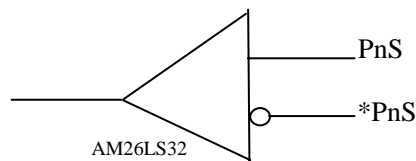


Fig. 2-13 Encoder signal circuit

2.2.3 Being connected with spindle encoder interface

KY980Tc 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):

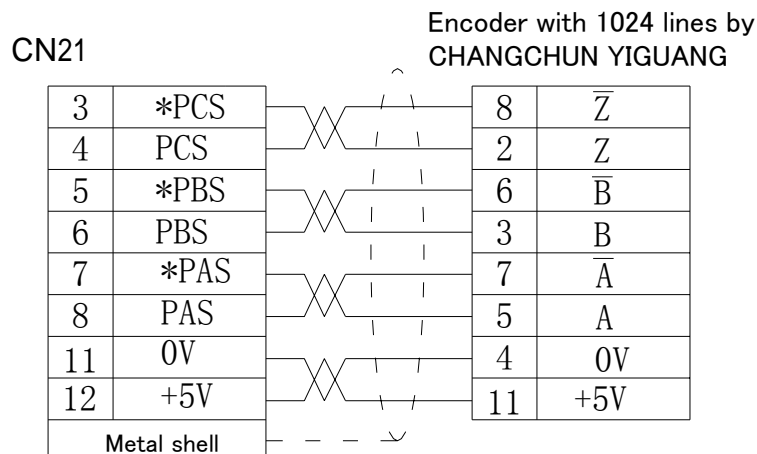


Fig. 2-14 KY980Tc is connected with the encoder

2.3 Being connected with MPG (Manual Pulse Generator)

2.3.1 MPG interface definition

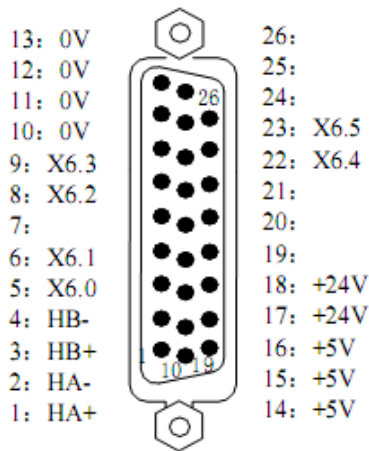


Fig. 2-15 CN31 MPG interface
(26-core DB type male socket)

Signal	Explanation
HA+, HA-	MPG A phase signal
HA+, HA-	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:

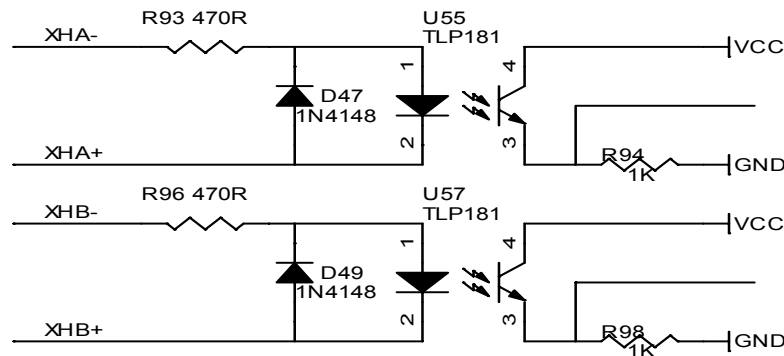
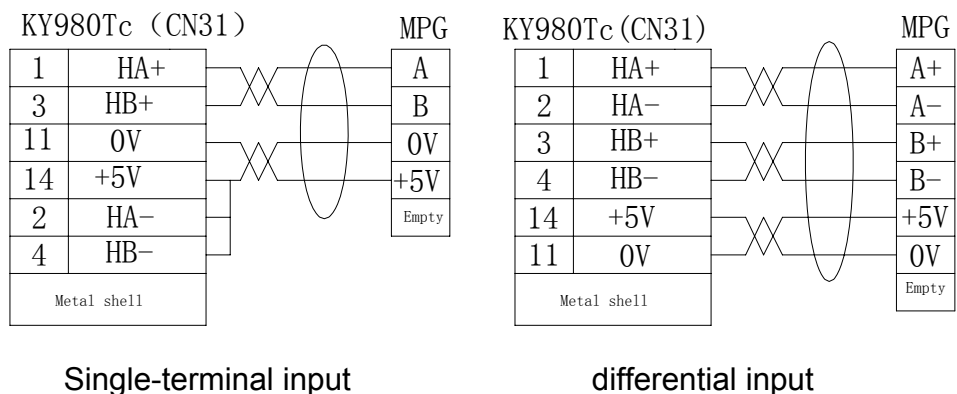


Fig. 2-16 MPG signal circuit

KY980Tc is connected with MPG shown in Fig. 2-17:



Single-terminal input

differential input

Fig. 2-17 KY980Tc is connected with MPG

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:

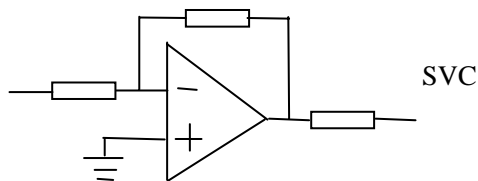


Fig. 2-20 SVC signal

circuit KY980Tc is connected with the inverter shown in Fig.

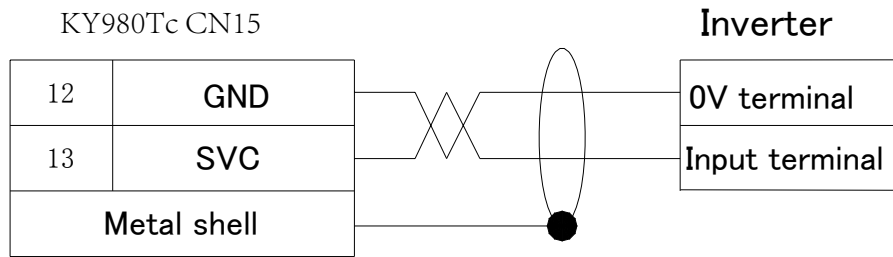


Fig. 2-21 KY980Tc is connected with the inverter

2.5 KY980Tc being connected with PC

2.5.1 Communication interface definition

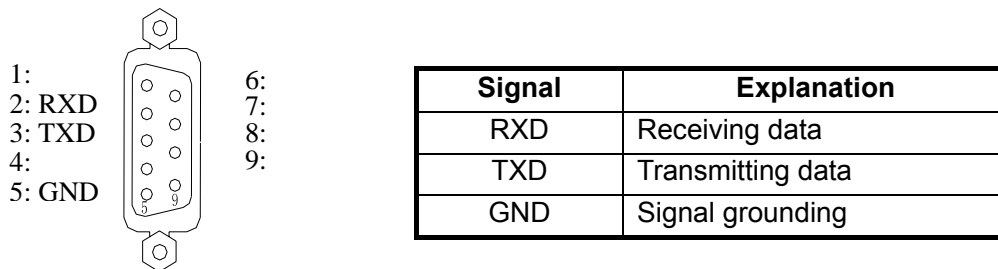


Fig. 2-22 CN51 communication interface (DB9-female)

2.5.2 Communication interface connection

KY980Tc can perform the communication by CN51 and PC(optional communication software). KY980Tc is connected with PC shown in Fig 2-23A:

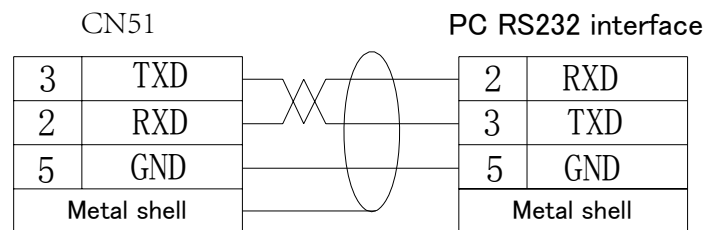


Fig. 2-23A KY980Tc is connected with PC

The communication between a KY980Tc system to another KY980Tc system can be done by CN51 shown in Fig. 2-23B:

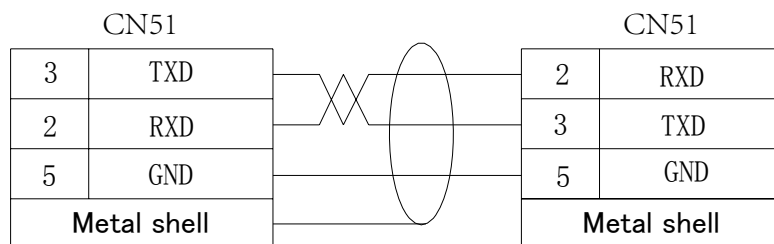


Fig. 2-23B Communication between a KY980Tc system and another KY980Tc system

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.

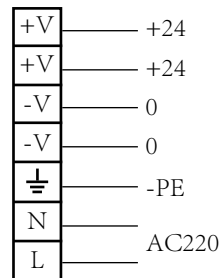


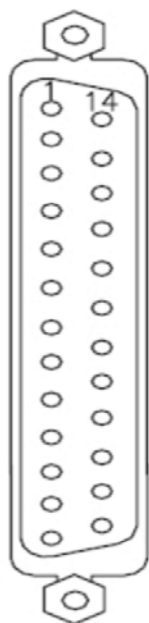
Fig. 2-24 system power interface CN1

2.7 I/O interface definition

Note!

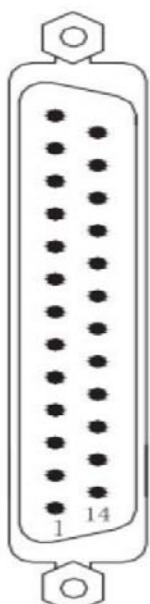
The I/O function significances of the unlabelled fixed addresses of this KY980Tc turning machine CNC system are defined by PLC programs (ladders), and they are defined by the machine builder when matching with a machine, please refer to the manual by the machine builder.

The fixed address I/O function not be marked are described for KY980Tc PLC. The described contents without other special explanation are also applied to KY980Tc-V.



interface(CN61)

Pin	Address	Function	Explanation
3,10,19	0V	Power supply interface	Power supply 0V side
11,23,16 7,20,4	+24V	Power supply interface	Power supply 24V side
1	X0.0	WQPJ	Inner/outer chuck releasing/ clamping in-position signal
14	X0.1	SP	External suspend/three-pos switch 1
2	X0.2	DIQP	Chuck control input signal
15	X0.3	DECX	X deceleration signal
17	X0.4	DITW	Tailstock control input/three-pos 2
5	X0.5	ESP	Emergency stop signal
18	X0.6	TCP	Tool rest locked signal
6	X0.7	T01	Tool position signal T01
8	X1.0	T02	Tool position signal T02
21	X1.1	T03	Tool position signal T03
9	X1.2	T04	Tool position signal T04
22	X1.3	DECZ	Z axis deceleration signal
24	X1.4	ST	External cycle start signal
12	X1.5	LMIX	X axis overtravel input signal
25	X1.6	LMIZ	Z axis overtrave input signall
13	X1.7	NQPJ	Inner/outer chuck clamping/ releasing in-position signal



Interface (CN62)

Pin	Address	Function	Explanation
3,6,19, 7,10,23	0V	Power supply interface	Power supply 0V terminal
4,20,11	+24V	Power supply interface	Power supply +24V terminal
1	Y0.0	COOL	Cooling output
2	Y0.1	M32	Lubricating output
3	Y0.2	SPZD	Spindle brake(R330.2)
4	Y0.3	M03	Spindle rotation (CCW)
5	Y0.4	M04	Spindle rotation (CW)
6	Y0.5	M05	Spindle stop
7	Y0.6	SCLP	Tailshaft forward/M70 M71(K26.4)
8	Y0.7	SPZD	Tailshaft backward/M72 M73(K26.4)
9	Y1.0	S1/M41	Spindle machine gear output 2/green light
10	Y1.1	S2/M42	Spindle machine gear output 1/yellow light
11	Y1.2	S3/M43	Spindle machine gear output 3/red light
12	Y1.3	S4/M44	Spindle machine gear output 4/M34/M35
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

- 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 KY980Tc-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:

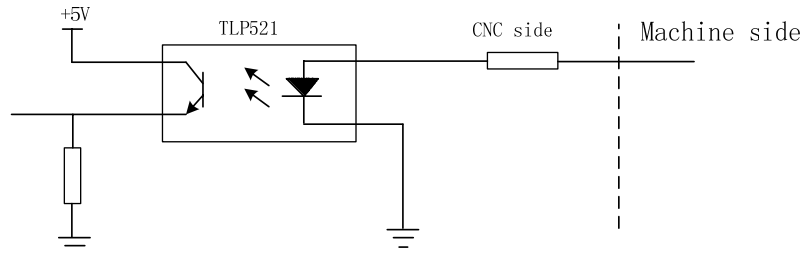


Fig. 2-27

The other type is input by switch with no contacts (transistor) as shown in Fig. 2-28A, 2-28B:

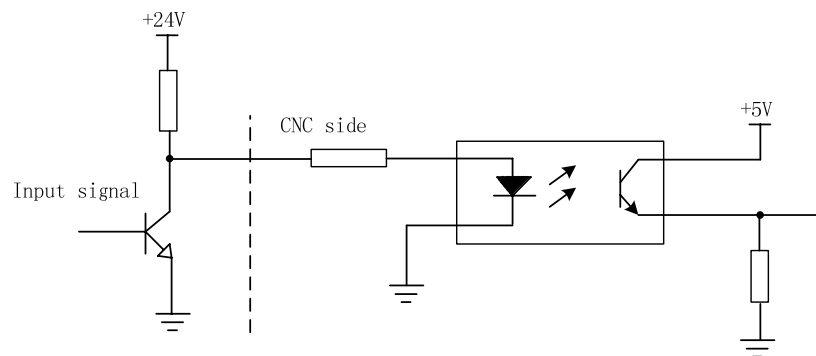


Fig. 2-28A NPN connection

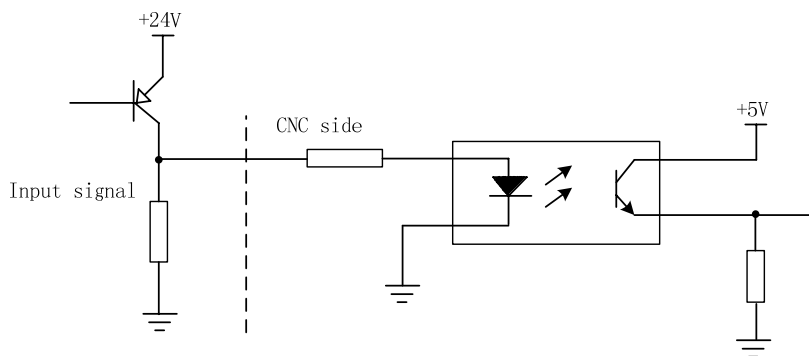


Fig. 2-28B PNP connection

The input interface signals defined by PLC of KY980Tc-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:

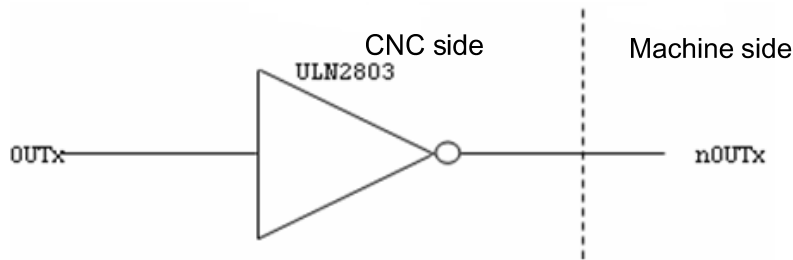


Fig. 2-29 Circuit for digital volume output module

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:

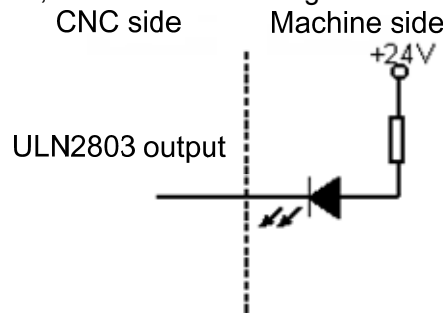


Fig. 2-30

- 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:

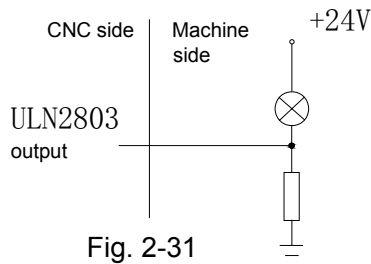


Fig. 2-31

- 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:

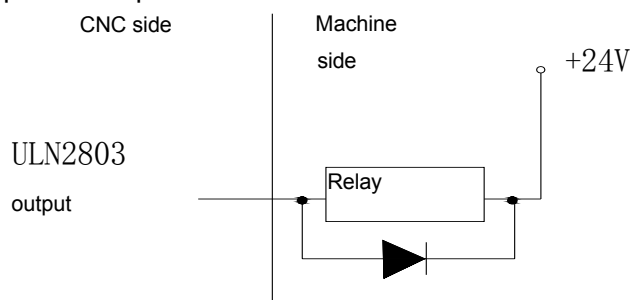


Fig. 2-32

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 KY980Tc 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 KY980Tc 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	X3.0	X3.1	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:

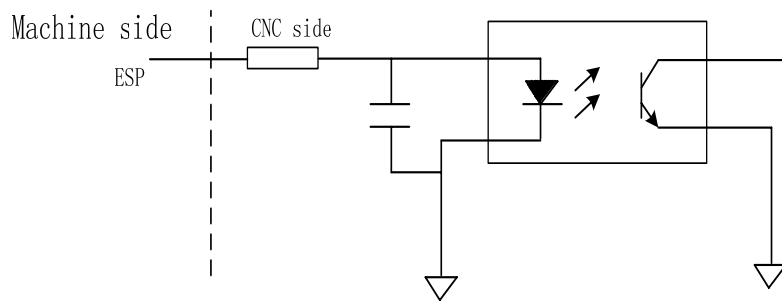


Fig. 2-33 The ESP signal circuit

● Machine external connection

(1) The series connection between the emergency stop and travel switch is shown in Fig. 2-34A:

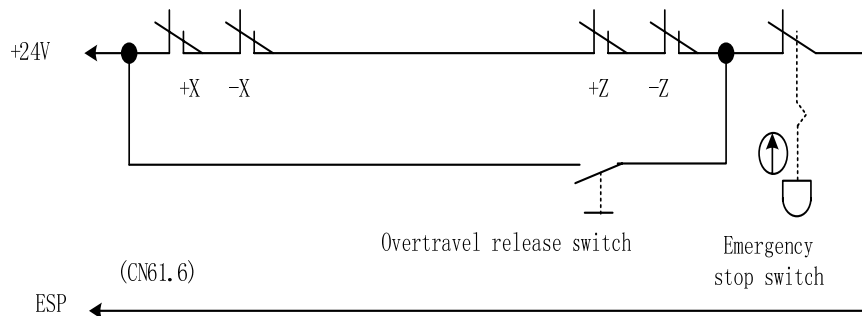


Fig.2-34A Series connection between emergency stop and travel switch

Bit paramete

k	1	1	CHET	TCPS	CTCP	TSGN	CHT	CHT
----------	----------	----------	------	------	------	------	-----	-----

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

- The T01~T08, TCP signals input are employed with photocoupler, its interior circuit is shown in Fig. 2-35:

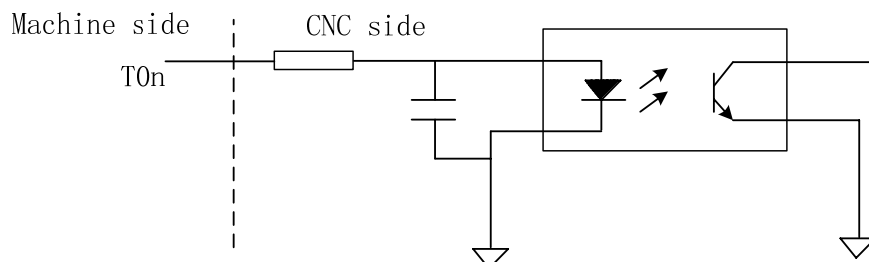


Fig. 2-35

- TL+, TL- are tool post CCW/CW output signal, its interior circuit is shown in Fig.2-36:

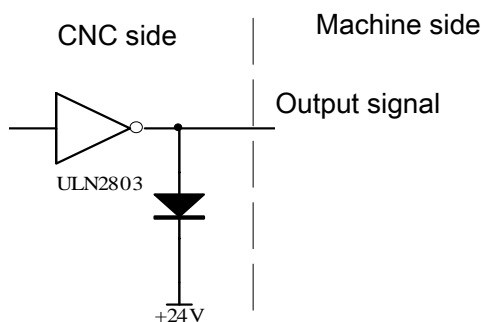


Fig. 2-36

- 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.

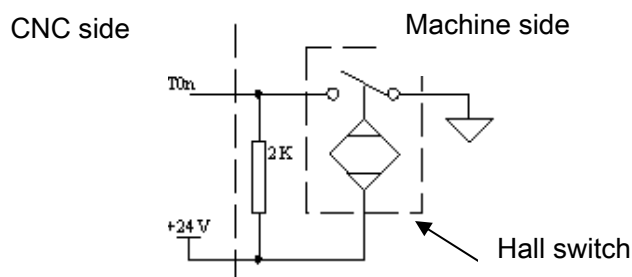


Fig. 2-37

● **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

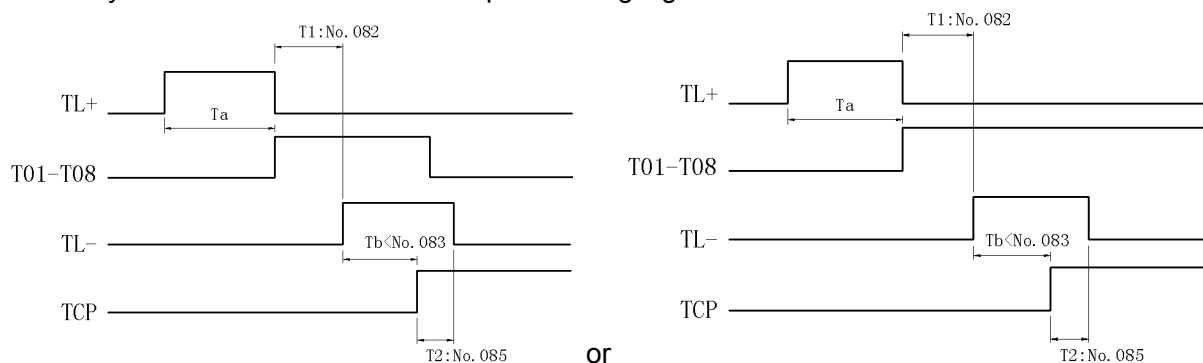
① During the tool change process, CNC outputs TL+ signal until the tool in-position signal is detected, then CNC turns off TL+ signal output and outputs TL- signal after a delay time specified by data parameter No.082. Then CNC detects TCP signal till it is detected, the CNC turns off TL- signal after a delay time specified by the data parameter No.085. So the tool change is over.

② When 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.



Sequence of tool change mode

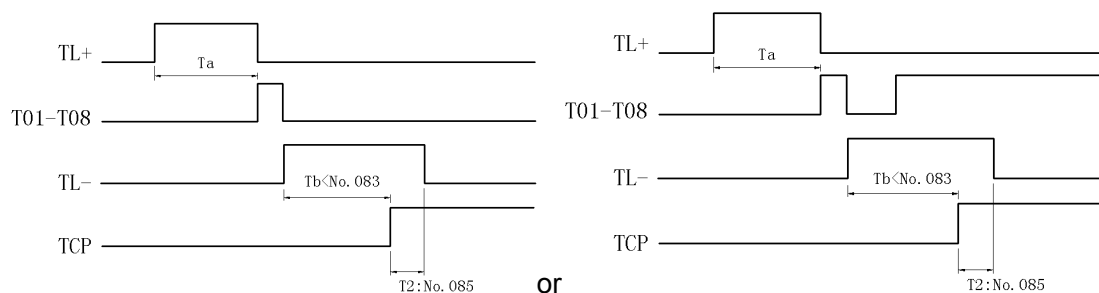
B 2.CHT =1, tool change mode A

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 tool 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;

PCX: X zero signal;

PCY: Y zero signal;

PCZ: Z zero signal;

- Diagnosis data

0	0	0						DECZ		DECX
Interface pin								CN61.12		CN61.4

- Control parameter

K	2	2		DECY	DECZ	DECX				
---	---	---	--	------	------	------	--	--	--	--

DECY=0: Y decelerates as DEC signal is LOW level
 =1: Y decelerates 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

=0: The direction of machine zero return is positive

● Data parameter

0	3	3	ZRNFL
---	---	---	-------

ZRNFL =Low rate of axes reference return

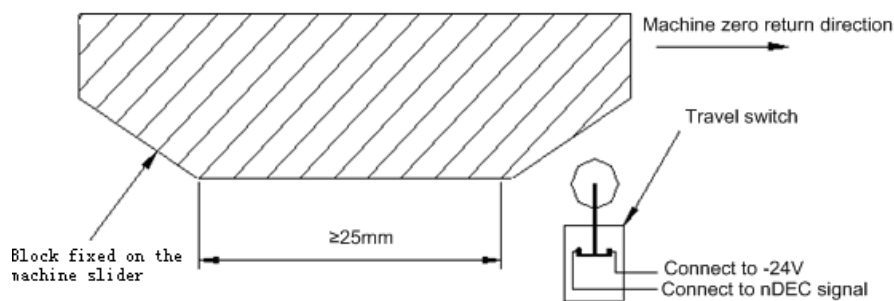
1	1	3	ZRNFH
---	---	---	-------

ZRNFH =High-speed of X,Z axes reference return

1	7	7	ZRNPHY
---	---	---	--------

ZRNPHY =High-speed of Y 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

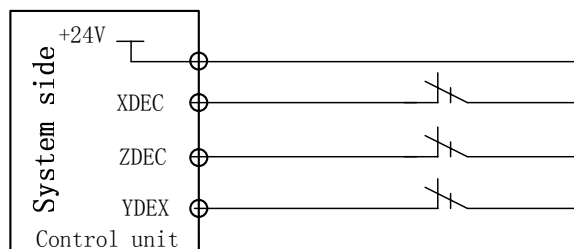
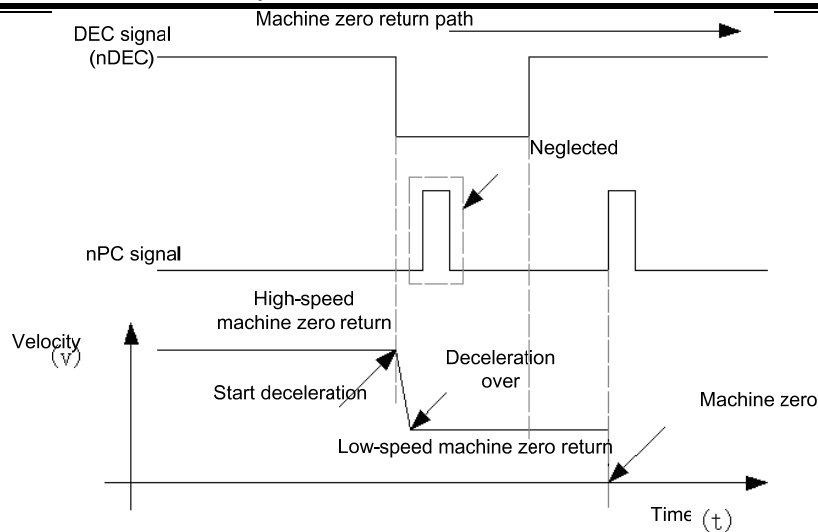


Fig. 2-40

③ Sequence of machine zero return

When the BIT0 (ZMX) of the bit parameter No.006 is set to 0, and the BIT5(DEC1) 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(DEC1) 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:

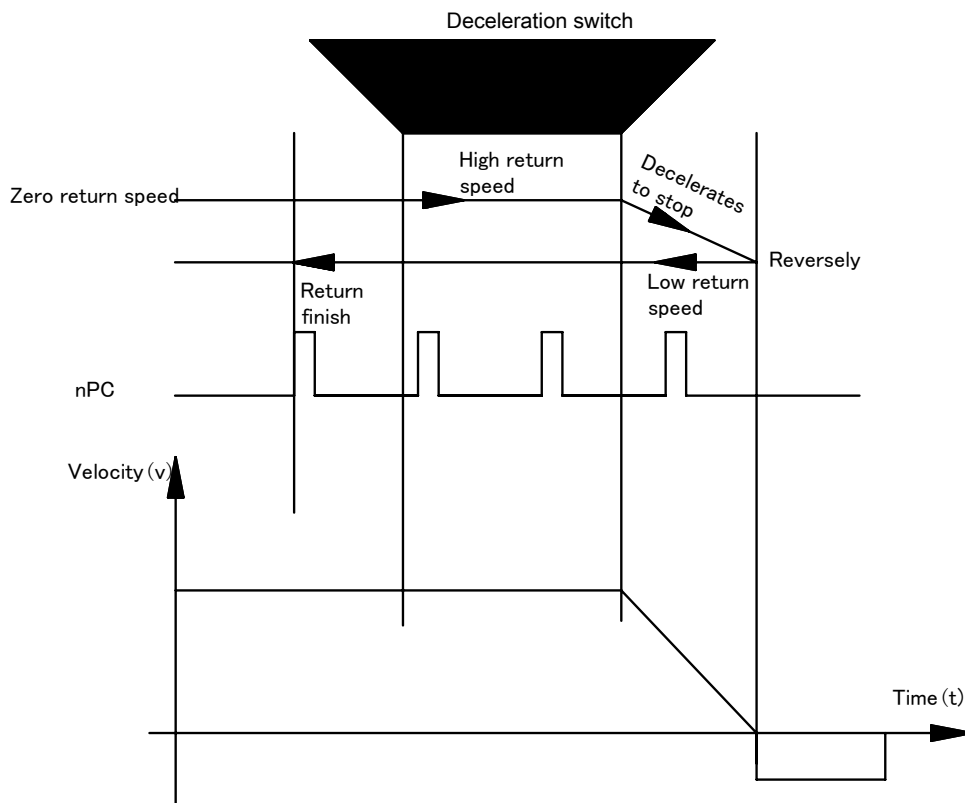


Fig. 2-41-b

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	SALM	CN21.2	X4.3	Spindle abnormality alarm input	
Output signal	M03	CN62.15	Y0.3	Spindle rotation(CCW)	
	M04	CN62.17	Y0.4	Spindle rotation(CW)	
	M05	CN62.5	Y0.5	Spindle stop	
	SPZD	CN62.2	Y0.2	Spindle brake	
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

K	1	0						RSJG	
----------	----------	----------	--	--	--	--	--	-------------	--

RSJG =1: CNC not turn off M03, M04, M08, M32 output signals when pressing  key;

=0: CNC turns off M03, M04, M08, M32 output signals when pressing  key.

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.

0	8	7		SPDDL T							
---	---	---	--	---------	--	--	--	--	--	--	--

Spindle brake delay output time(ms) after spindle stop outputting(M05)

0	8	9		SPZD TIME							
---	---	---	--	-----------	--	--	--	--	--	--	--

Spindle brake output time(ms)

D	9	9		MTIME							
---	---	---	--	-------	--	--	--	--	--	--	--

Spindle zero speed output range(r/min)

● 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.

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

0	0	1				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			Y1.3	Y1.2	Y1.1	Y1.0
Interface pin			CN62.12	CN62.11	CN62.10	CN62.09

● Signal connection

The circuit for M41~M44 is shown in Fig.2-47:

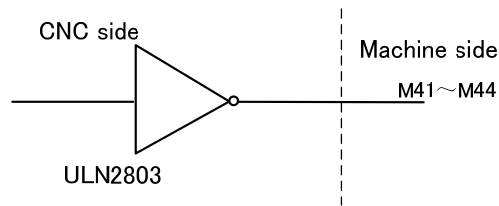


Fig. 2-47

● 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.

2.8.11 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	CN61.14	CN61.24

- **Signal connection**

The interior circuit of SP/ST signal is shown in Fig. 2-48:

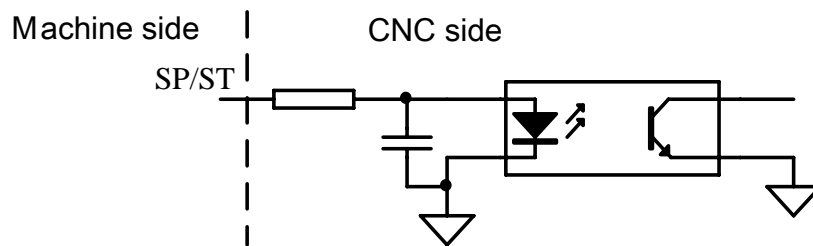


Fig. 2-48

- **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:

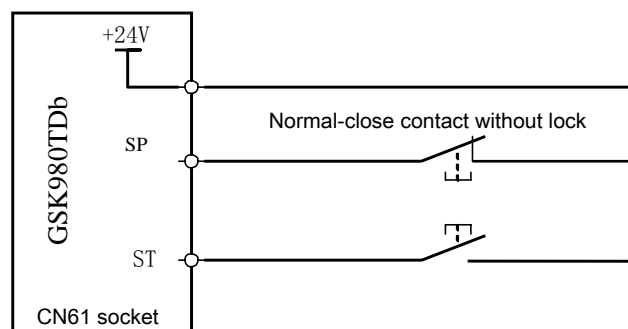


Fig. 2-49

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			Cooling ON	
	M09			Cooling OFF	

- **Signal connection**

Its internal circuit is shown in Fig. 2-50:

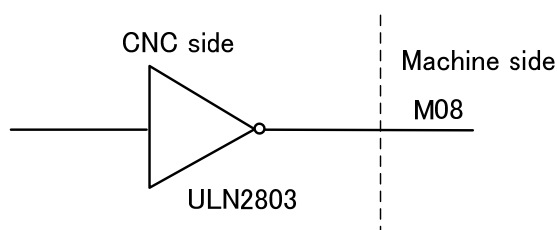


Fig. 2-50 M08 internal circuit

- **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 KY980Tc 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


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 KY980Tc 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

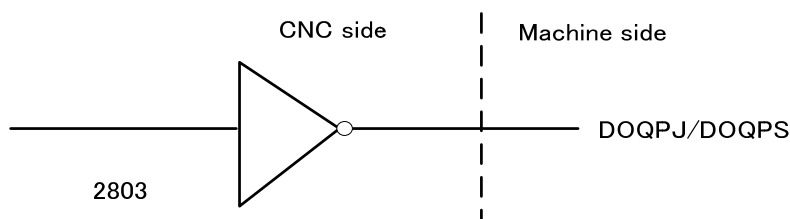


Fig. 2-52

● 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.

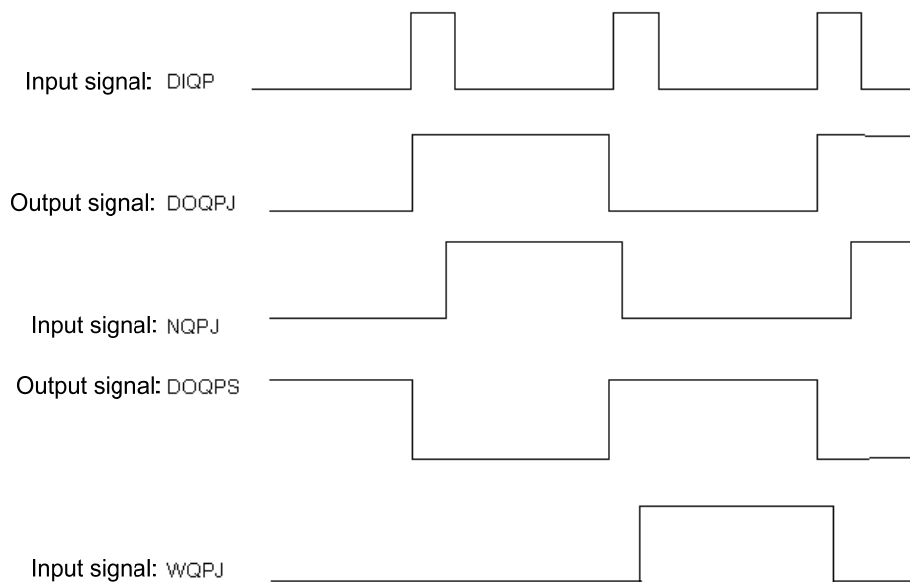


Fig. 2-53 (Chuck clamping, releasing signals are level output)

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.

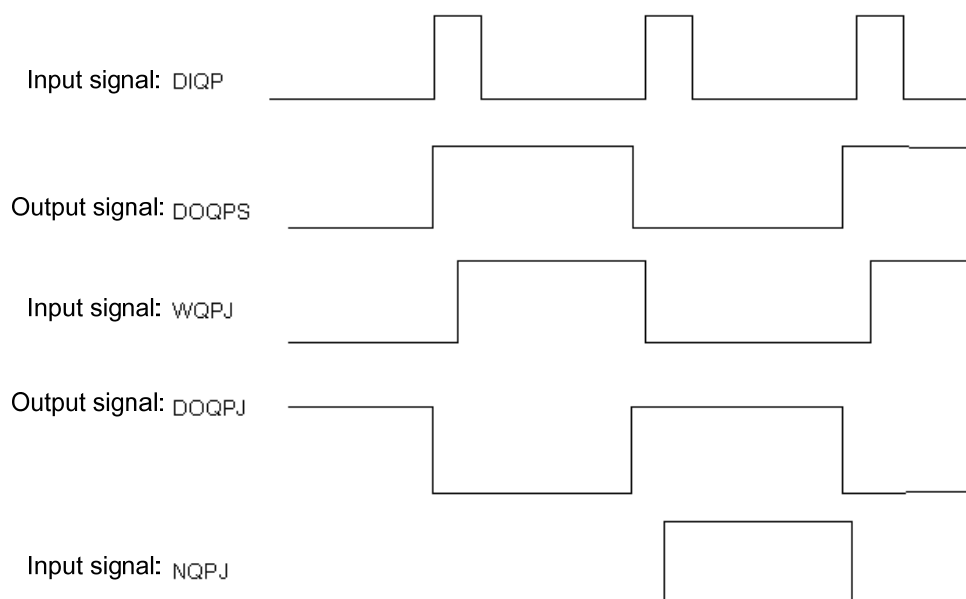


Fig. 2-54 Chuck clamping, releasing signals are level output

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.

③ The 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 GSK980TDb-V,  can replace DITW input signal, and the diagnosis address is X0026.5.

● Signal diagnosis

Signal	DITW	DOTWJ	DOTWS
Diagnosis address	X0.4	Y0.6	Y0.7
Interface pin	CN61.17	CN62.18	CN61.6

● 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:

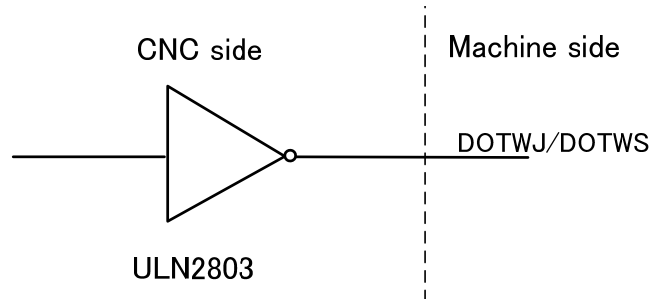


Fig. 2-55

● Sequence (defined by standard PLC program)

The sequence of tailstock is shown in Fig. 2-56:

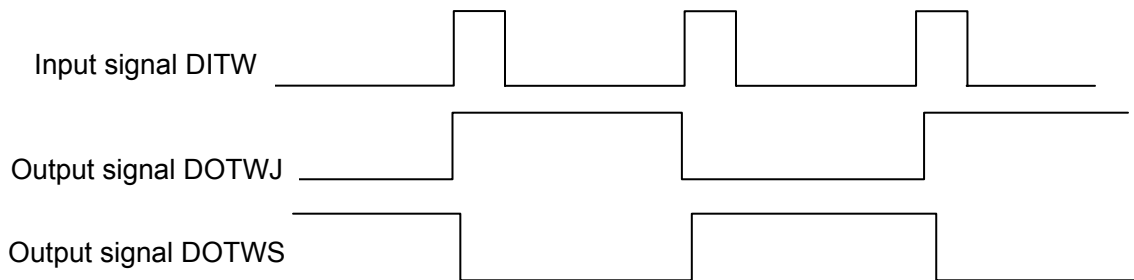


Fig. 2-56 Tailstock sequence

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:

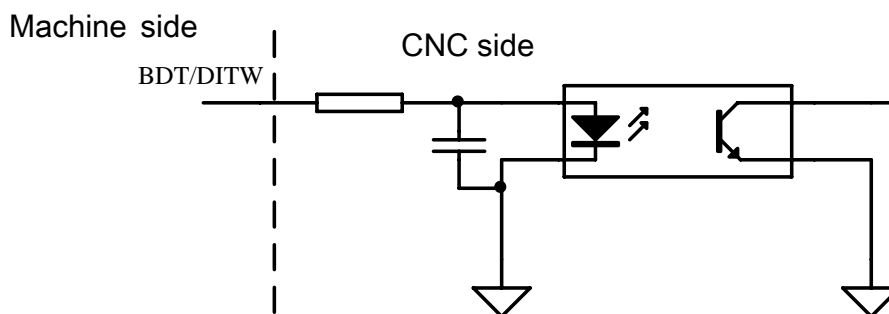


Fig. 2-57

- **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:

Y1.0 (CN62.8): Yellow light, normal (non-running, non-alarming)

Y1.1 (CN62.21): Green light, running

Y1.2 (CN62.9): Red light, alarming

2.8.16 External MPG

● Related signals

CN31(MPG)	PLC address	Address character	Function	Remark
5	X5.0	EHDY	X MPG	Applied to PSG-100-05E/L, ZSSY2080 MPG
6	X5.1	EHDY	Y MPG	
8	X5.2	EHDZ	Z MPG	
9	X5.3	EMP0	Increment ×1	
22	X5.4	EMP1	Increment ×10	
23	X5.5	EMP2	Increment ×100	
11, 12, 13	GND			
14,15	+5V			
17,18	+24V			

● Related parameters

State parameter

0	0	1					SHOW			
---	---	---	--	--	--	--	------	--	--	--

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.

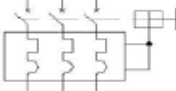


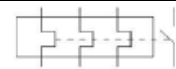





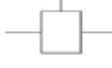






=1: MPG, STEP mode×1000-gear increment is invalid.

● Function description

- ① When SINC is set to 1, MPG/STEP mode ×1000-gear selection is disabled. When ×1000-gear is 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

KY980Tc 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		Contactors 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:

<table><tr><td></td><td></td><td></td></tr></table>				<table><tr><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td></tr></table>								
Parameter No.	BIT7	BIT6	BIT5	BIT4	BIT3	BIT2	BIT1	BIT0				
<table><tr><td>0</td><td>0</td><td>1</td></tr></table>	0	0	1	<table><tr><td>***</td><td>***</td><td>***</td><td>SPTY</td><td>SOHW</td><td>RDC</td><td>***</td><td>INI</td></tr></table>	***	***	***	SPTY	SOHW	RDC	***	INI
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			ZMY	ZMZ	ZMX
---	---	---	-----	-----	------	--	--	-----	-----	-----

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 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	***	***	***			YALM	ZALM	XALM
----------	----------	----------	------------	------------	------------	--	--	-------------	-------------	-------------

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	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	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

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 1 0


1	8	3	***	***	***			MZRY	MZRZ	MARX
---	---	---	-----	-----	-----	--	--	------	------	------


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

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

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 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 invalid 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

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

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

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 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 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

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

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

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

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: Ignore hard limit alarm

0: Not Ignore hard limit alarm

Bit2 1: Ignore emergency stop alarm

0: Not Ignore emergency stop alarm

Bit1 1: Ignore alarm of the spindle drive

0: Not Ignore alarm of the spindle drive

Bit0 1: Ignore alarm of the feed shaft drive

0: Not Ignore alarm of the feed shaft drive

Default: 0 0 0 0 0 0 0 0

3.1.2 Data parameter

0	1	5
0	1	6

[Data range]

Default value

CMRX(X axis) multiplier coefficient
CMRZ(Z axis) multiplier coefficient

1 ~ 65536

1

0	1	7
0	1	8

[Data range]

X axes frequency division coefficient
Z axes frequency division coefficient

1 ~ 65536

Electronic gear ratio formula:

$$\frac{CMR}{CMD} = \frac{P}{L \times 1000}$$

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
---	---	---

[Data unit]

Rapid traverse rate as axis rapid override is F0

Setting unit	Data unit
Metric machine	mm/min
Inch machine	0.1inch/min

[Data range] 6~4000

Default value 1260

0	3	3
---	---	---

[Data unit]

Low rate as axes return to machine zero

Setting unit	Data unit
Metric machine	mm/min
Inch machine	0.1inch/min

[Data range] 6~4000

Default value 40

0	3	4
---	---	---

[Data unit]

X backlash compensation.**Z backlash compensation**

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
---	---	---

0	3	8
---	---	---

0	3	9
---	---	---

0	4	0
---	---	---

Max. speeds of gear1**Max. speeds of gear2****Max. speeds of gear3****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
0	4	6

X positive max. travel**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
0	4	8

X negative max. travel
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**

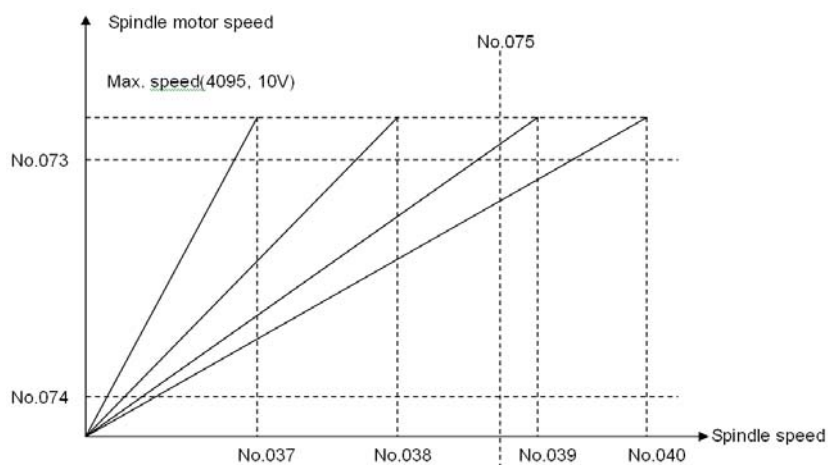
[Data unit] r/min
 [Data range] 0 ~4095

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

0

Default value

1	1	9
---	---	---

Z negative max. travel

[Data range]

-9999.9999~9999.9999

Default value

-9999

1	2	0
1	2	1
1	2	2
1	2	3
1	2	4
1	2	5
1	2	6
1	2	7

X 1st reference point machine coordinates**Z 1st reference point machine coordinates****X 2nd reference point machine coordinates****Z 2nd reference point machine coordinates****X 3rd reference point machine coordinates****Z 3rd reference point machine coordinates****X 4th reference point machine coordinates****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
1	2	9

G54_X Offset of the X axis coordinate system 1**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
1	3	1

G55_X Offset of the X axis coordinate system 2**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

[Data unit]

[Data range]

Default value

1	3	4
1	3	5

[Data unit]

[Data range]

Default value

1	3	6
1	3	7

[Data unit]

[Data range]

Default value

1	3	8
1	3	9

[Data unit]

[Data range]

Default value

G56_X Offset of the X axis coordinate system 3
G56_Z Offset of the X axis coordinate system 3

Setting unit	Data unit
Metric machine	mm
Inch machine	inch

-999.999~999.999

0

G57_X Offset of the X axis coordinate system 4
G57_Z Offset of the X axis coordinate system 4

Setting unit	Data unit
Metric machine	mm
Inch machine	inch

-999.999~999.999

0

G58_X Offset of the X axis coordinate system 5
G58_Z Offset of the X axis coordinate system 5

Setting unit	Data unit
Metric machine	mm
Inch machine	inch

-999.999~999.999

0

G59_X Offset of the X axis coordinate system 6
G59_Z Offset of the X axis coordinate system 6

Setting unit	Data unit
Metric machine	mm
Inch machine	inch

-999.999~999.999

0

1	4	0
---	---	---

[Data unit]

[Data range]

Default value

1	4	6
1	4	7
1	4	8

[Data range]

Default value

1	4	9
1	5	0
1	5	1

[Data range]

Default value

1	5	3
---	---	---

[Data range]

Default value

1	5	4
---	---	---

[Data unit]

[Data range]

Default value

1	5	5
1	5	6
1	5	7

[Data unit]

[Data range]

Default value

+ or – limit of each input wear in TOOL OFFSET&WEAR interface

Setting unit	Data unit
Metric machine	mm
Inch machine	inch

0.001~99.9999

1

Y pulse output multiplication coefficient**4TH pulse output multiplication coefficient****5TH pulse output multiplication coefficient**

1 ~65536

1

Y pulse output division coefficient**4TH pulse output division coefficient****5TH pulse output division coefficient**

1 ~65536

1

current being used ladder numbe

0 ~15

1

Max. arc radius error

Setting unit	Data unit
Metric machine	mm
Inch machine	inch

0.0001~1

0.01

Y max. rapid traverse speed**4TH max. rapid traverse speed****5TH max. rapid traverse speed**

Setting unit	Data unit
Metric machine	mm/min
Inch machine	inch/min
Rotary axis	deg/min

0~90000

5000

1	5	8
1	5	9
1	6	0

Y acceleration/deceleration time constant value in rapid traverse
4TH acceleration/deceleration time constant value in rapid traverse
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

[Data unit]

Y backlash compensation value
4TH backlash compensation value
5TH backlash compensation value

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

[Data unit]

Y pitch error compensation interval
4TH pitch error compensation interval
5TH pitch error compensation interval

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

[Data range]

0 ~255

Default value

0

1	8	9
1	9	0
1	9	1

[Data unit]

Offset value of Y machine zero
Offset value of 4TH machine zero
Offset value of 5TH machine zero

Setting unit	Data unit
Metric machine	mm/min
Inch machine	inch/min
Rotary axis	deg/min

[Data range]

0~100

Default value

0

1	9	2
1	9	3
1	9	4

Positive max. travel of Y
Positive max. travel of 4th
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
1	9	3
1	9	4

Negative max. travel of Y
Negative max. travel of 4th
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
2	0	2
2	0	3
2	0	4
2	0	5
2	0	6
2	0	7
2	0	8
2	0	9
2	1	0
2	1	1
2	1	2

Y 1st reference point machine coordinates
4TH 1st reference point machine coordinates
5TH 1st reference point machine coordinates
Y 2nd reference point machine coordinates
4TH 2nd reference point machine coordinates
5TH 2nd reference point machine coordinates
Y 3rd reference point machine coordinates
4TH 3rd reference point machine coordinates
5TH 3rd reference point machine coordinates
Y 4th reference point machine coordinates
4TH 4th reference point machine coordinates
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

2	2	5
2	2	6
2	2	7

Axis name definition of Y
Axis name definition of 4th
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)

2	3	7
---	---	---

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

2	3	8
---	---	---

Arc interpolation control precision
--

[Data range]

0~0.5

Default value

0.03

2	4	0
2	4	1
2	4	2
2	4	3
2	4	4

X axis pitch error compensation ratio
Z axis pitch error compensation ratio
Y axis pitch error compensation ratio
4TH axis pitch error compensation ratio
5TH axis pitch error compensation ratio

[Data range]

0~9999.9999

Default value

0.001

2	4	5
---	---	---

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
2	4	7
2	4	8
2	4	9
2	5	0

Compensation step for X axis space with fixed frequency
Compensation step for Z axis space with fixed frequency
Compensation step for Y axis space with fixed frequency
Compensation step for 4TH axis space with fixed frequency
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
2	7	1
2	7	2
2	7	3
2	7	4

X axis offset of the origin of the external workpiece
Z axis offset of the origin of the external workpiece
Y axis offset of the origin of the external workpiece
4TH axis offset of the origin of the external workpiece
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
2	7	6
2	7	7
2	7	8
2	7	9
2	8	0
2	8	1
2	8	2
2	8	3
2	8	4
2	8	5
2	8	6
2	8	7
2	8	8
2	8	9
2	9	0
2	9	1
2	9	2

G54 Offset of the Y axis coordinate system 1
G54 Offset of the 4th axis coordinate system 1
G54 Offset of the 5th axis coordinate system 1
G55 Offset of the Y axis coordinate system 2
G55 Offset of the 4th axis coordinate system 2
G55 Offset of the 5th axis coordinate system 2
G56 Offset of the Y axis coordinate system 3
G56 Offset of the 4th axis coordinate system 3
G56 Offset of the 5th axis coordinate system 3
G57 Offset of the Y axis coordinate system 4
G57 Offset of the 4th axis coordinate system 4
G57 Offset of the 5th axis coordinate system 4
G58 Offset of the Y axis coordinate system 5
G58 Offset of the 4th axis coordinate system 5
G58 Offset of the 5th axis coordinate system 5
G59 Offset of the Y axis coordinate system 6
G59 Offset of the 4th axis coordinate system 6
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
2	9	4
2	9	5

Positive max. travel of Y(Second travel limit)
Positive max. travel of 4th(Second travel limit)
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
2	9	7
2	9	8

Negative max. travel of Y(Second travel limit)**Negative max. travel of 4th(Second travel limit)****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
3	0	6

L type time constant for the acceleration and deceleration fast X axis

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
3	0	8

L type time constant for the acceleration and deceleration fast X axis

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
3	1	0

L type time constant for the acceleration and deceleration fast X axis

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
3	1	2

E type time constant for the acceleration and deceleration fast X axis

E type time constant for the acceleration and deceleration fast Z axis

[Data unit] ms

[Data range] 1~4000

Default value 60

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
3	2	2
3	2	3

Line type time constant for the acce and dece in the thread cutting(gear1)

Line type time constant for the acce and dece in the thread cutting(gear2)

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
3	2	5
3	2	6

S type time constant for the acce and dece in the thread cutting(gear1)

S type time constant for the acce and dece in the thread cutting(gear2)

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
3	2	8
3	2	9

Spindle backlash compensation value(gear1)

Spindle backlash compensation value(gear2)

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

[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
3	4	6
3	4	7

L type time constant before the acce and dece fast Y axis
L type time constant before the acce and dece fast 4TH axis
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
3	4	9
3	5	0

S type time constant before the acce and dece fast Y axis
S type time constant before the acce and dece fast 4TH axis
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
3	5	2
3	5	3

L type time constant before the acce and dece fast Y axis
L type time constant before the acce and dece fast 4TH axis
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
3	5	5
3	5	6

E type time constant before the acce and dece fast Y axis
E type time constant before the acce and dece fast 4TH axis
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
3	5	8
3	5	9

Y axis direction and overshoot
4TH axis direction and overshoot
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

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 KY-980Tc 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 KY-980Tc 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):

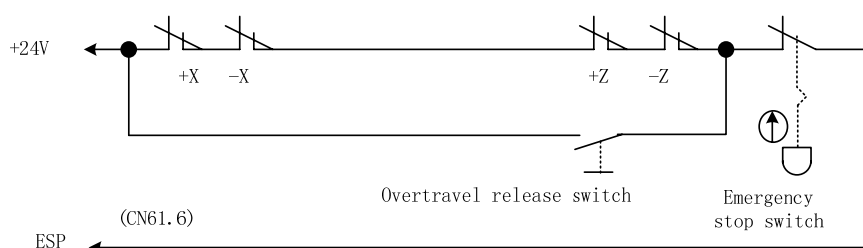


Fig. 4-1

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

BIT2, BIT1, BIT0 (YALM, ZALM, XALM separately corresponds to Y, Z, X) of bit parameter No.009 for our 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 BIT2, BIT1 and BIT0 (DIRY, DIRZ, DIRX separately corresponds to Y, Z, X) of bit parameter No.008.

The manual move direction can be set by BIT2, BIT1, BIT0 (YVAL, ZVAL, XVAL separately corresponds to 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:

$$\frac{CMR}{CMD} = \frac{\delta \times 360}{\alpha \times L} \times \frac{Z_M}{Z_D}$$

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 GSK980TDb Z axis, 0.0005mm for X axis of GSK980TDb)

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 $\alpha = 0.075^\circ$, 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, No.155: X, Z, Y axis rapid traverse rate;

Data parameter No.024, No.025, No.158: linear ACC&DEC time constant of X, Z, Y 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.

4.5 Mechanical (machine) zero adjustment

Related signal

DECX: X axis deceleration signal;

DECY: Y axis deceleration signal;

DECZ: Z axis deceleration signal;

PCX: X axis zero signal;

PCY: Y axis zero signal;

PCZ: Z axis zero signal;

DGN DATA

0	0	0						DECZ	DECY	DECX
Interface pin								CN61.12	CN61.32	CN61.4

Control PAR

K	2	2		DECY	DECZ	DECX				
---	---	---	--	------	------	------	--	--	--	--

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						MZRY	MZRZ	MZRX
---	---	---	--	--	--	--	--	------	------	------

MZRX=1: Select the zero direction is negative;

=0: Select the zero direction is positive.

DATA PAR

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.

Adjust the relevant parameters based on the active level of the connection signal, zero return type and direction applied:

BIT6, BIT5, BIT4 (ZMX, ZMZ, ZMY) of the K parameter No.022: Valid level of deceleration signal when X,Z,Y return to machine zero point

BIT5 (ZMOD) of the bit parameter No.006: X, Z, Y axis machine zero return mode B or C selection(B:behind block C:front block).

BIT0 of the bit parameter No.007: Return zero mode selection:(0:no 1:yes) have a-turn signal

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, (MZRZ, MZRZ, MZRY) 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

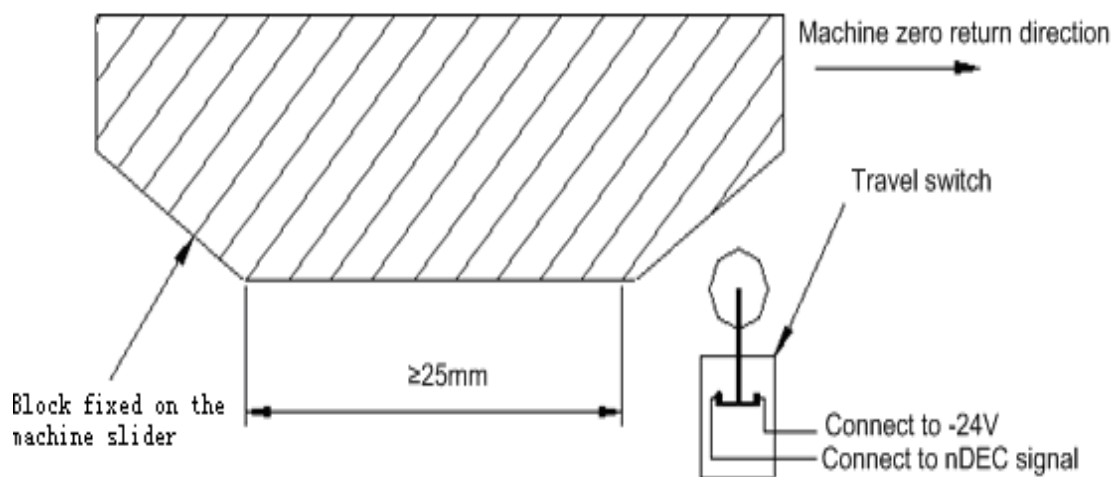


Fig. 4-2

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.006 BIT5(ZMOD) =0

Bit parameter No.007 BIT0(ZPLS) =1

Data parameter No.033=200

Data parameter No.183 BIT0(MZRX) , BIT1(MZRZ) , BIT2(MZRY) =0

- ② The connection to stepper motor: schematic diagram of using a proximity switch taken as both deceleration signal and zero signal

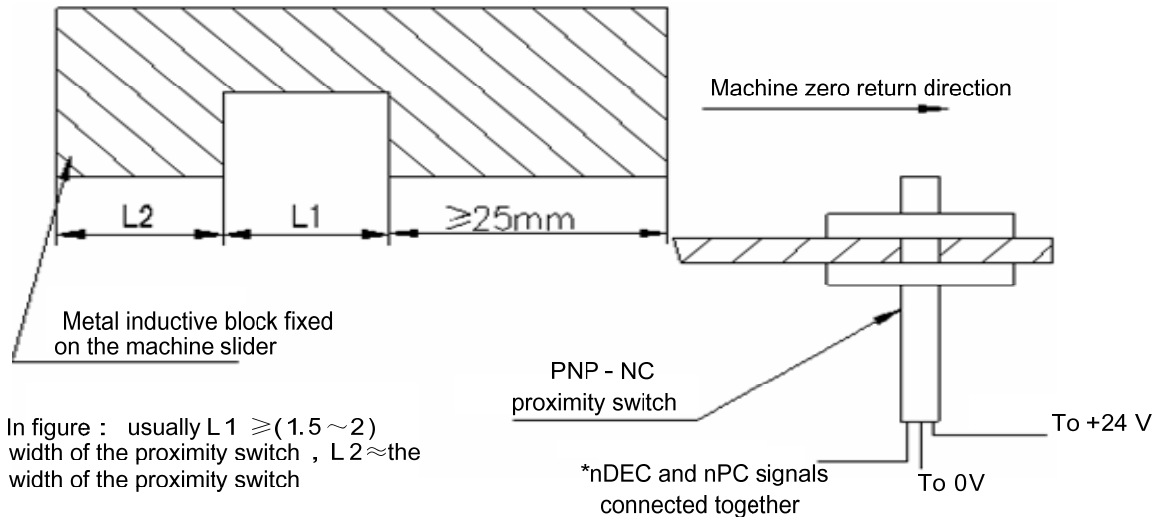


Fig. 4-3

When matching the stepper motor, the parameter settings are as follows:

Bit parameter No.006 BIT5(ZMOD) =0

Bit parameter No.007 BIT0(ZPLS) =0

Data parameter No.033=200

Data parameter No.183 BIT0(MZRX), BIT1(MZRZ), BIT2(MZRY) =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.

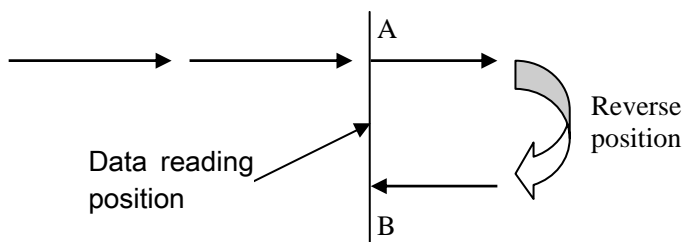


Fig. 4-4 Schematic map of backlash measuring method

- 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

KY980Tc 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 started, 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 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 KY980Tc 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 axis
- 5) Offset range: $0 \sim \pm 127 \mu\text{m}$ for each offset point
- 6) Offset interval: $1000 \sim 9999999 \mu\text{m}$;
- 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			SCRW						
---	---	---	--	--	------	--	--	--	--	--	--

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, No.186, 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.

6.3.3 Offset interval

Pitch offset interval: No.102, No.103, No.183;

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

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.

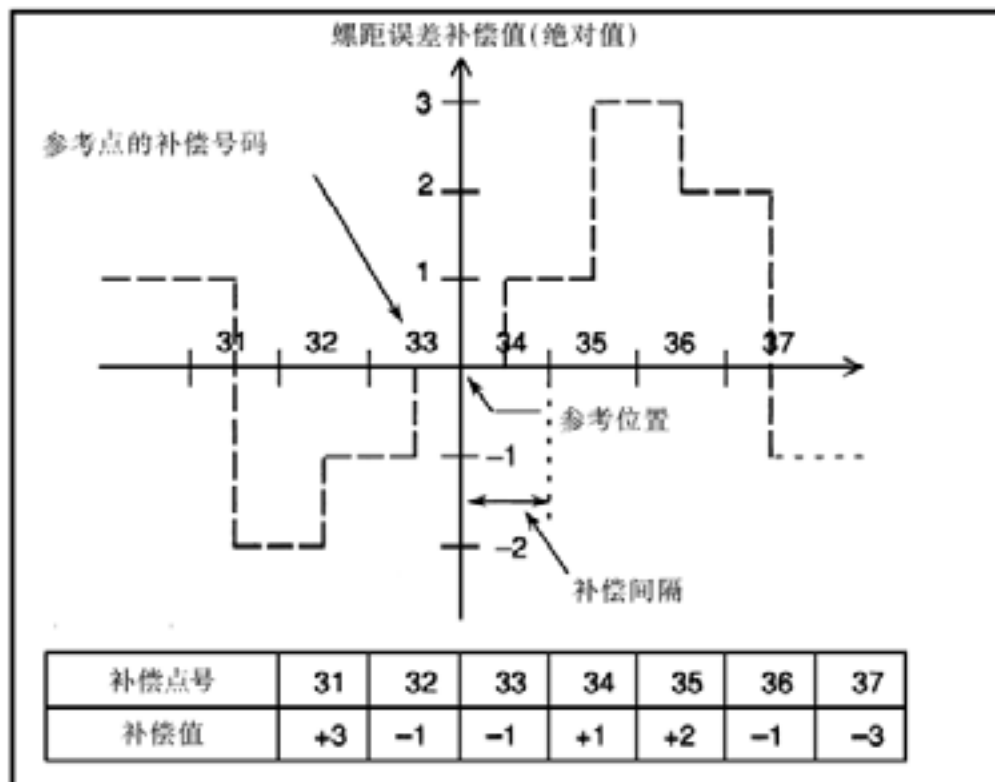
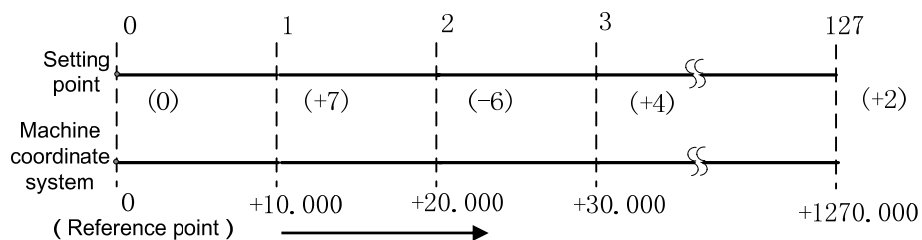
Offset No.	X	Z	Y
000
001	5	-2	3
002	-3	4	-1
...
255

6.4 Notes of offset setting

- ① The setting and alteration of pitch offset can only be done at the authority of password level 2.
- ② After the parameter of pitch offset is set, only the machine zero is returned could the offset be done.

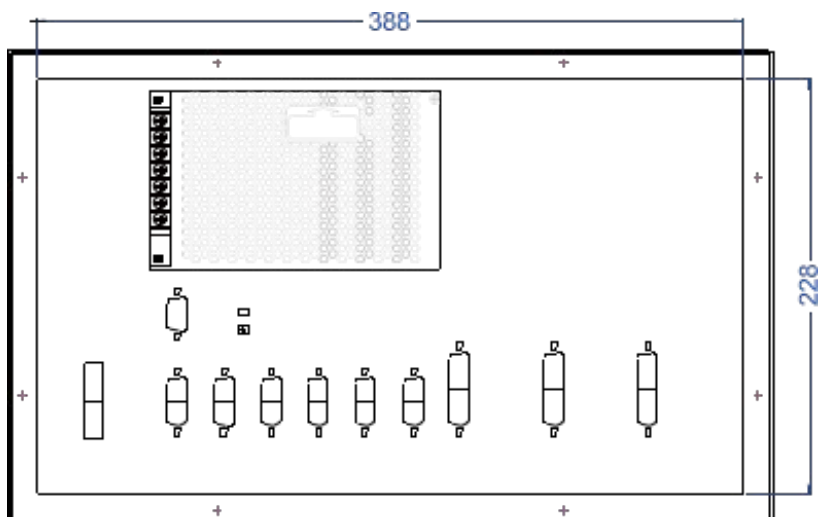
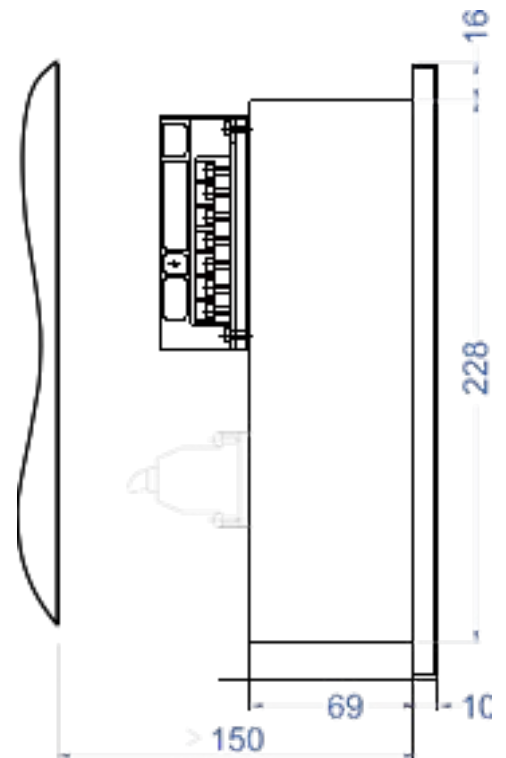
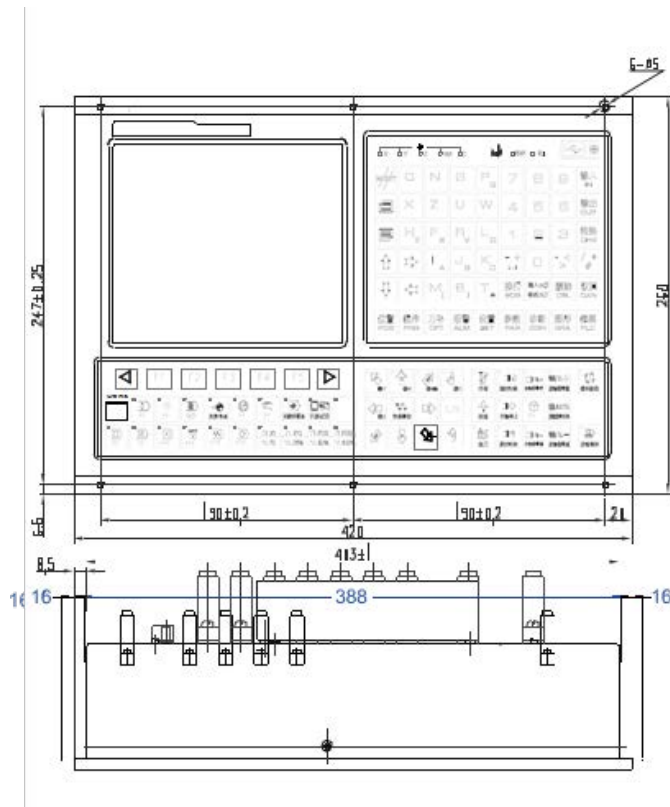
6.5 Setting examples of offset parameters

- ① Data parameter No.99(pitch error origin) =33, Data parameter No.103 (offset interval)=10.000mm
When the pitch error origin is set to 33:



Appendix

Appendix 1 KY980Tc contour dimension



Appendix 2 Additional panel dimensions

