

**Declaration!**

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

**Warning !**

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

**Caution !**

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

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

## Safety notes

### ■ Transportation and storage

- Do not pile up the packing boxes over 6 layers.
- Never climb the packing box, neither stand on it, nor place heavy objects on it.
- Do not move or drag the product by the cables connected to it.
- Avoid impact or scratch to the panel and screen.
- Packing box should be protected from dampness, insolation and drench.

### ■ Open-package inspection

- Confirm the product is the one you purchased after opening the package.
- Check whether the product is damaged during transportation.
- Confirm all the elements are complete without damage by referring to the list.
- If there is incorrect product type, incomplete accessories or damage, please contact us in time.

### ■ Connection

- Only qualified personnel can connect and inspect the system.
- The system must be earthed. The earth resistance should not be greater than  $0.1\Omega$ , and a neutral wire (zero wire) cannot be used as an earth wire.
- The connection must be correct and secured. Otherwise, the product may be damaged or unexpected results may occur.
- Connect the surge absorbing diode to the product in the specified direction; otherwise the product may be damaged.
- Turn off the power before inserting or unplugging a plug, or opening the electric cabinet.

### ■ Troubleshooting

- Turn off the power supply before troubleshooting or replacing components.
- Overhaul the system when there is a short circuit or overload, and do not restart it until the trouble is removed.
- Do not turn ON/OFF the product frequently, and the ON/OFF interval should be 1 minute at least.

## **BOOK I PROGRAMMING**

This part gives an introduction to the specification, product portfolio, parameter configuration, instruction codes as well as program format.

## **BOOK II OPERATION**

This part gives an introduction to the operation of the machining center CNC system of GSK 990MA .

## **APPENDIX**

This part gives an introduction to the use of the machining center CNC system and its accessories of GSK990MA.

# **CNCmakers**

---

[www.CNCmakers.com](http://www.CNCmakers.com)

# CONTENT

<b>I</b>	<b>OVERVIEW.....</b>	<b>1</b>
1	Overview .....	2
<b>II</b>	<b>PROGRAMMING.....</b>	<b>3</b>
1	General.....	4
1.1	Tool movement along workpiece contour —interpolation .....	4
1.2	Feed—Feed function .....	5
1.3	Cutting feedrate, spindle speed function .....	6
1.4	Operation instruction—miscellaneous function.....	6
1.5	Tool selection for various machining—Tool function .....	6
1.6	Tool figure and tool motion by program .....	7
1.6.1	Tool length compensation .....	7
1.6.2	Tool radius compensation .....	7
1.7	Tool movement range—stroke.....	8
2	Part Program Composition .....	9
2.1	Program composition .....	9
2.1.1	Program name .....	9
2.1.2	Sequence number and block .....	10
2.1.3	Instruction word.....	10
2.2	General structure of a program .....	11
2.2.1	Subprogram edit.....	12
2.2.2	Subprogram call.....	12
2.2.3	Program end .....	13
3	Programming Fundamentals .....	14
3.1	Controlled axis .....	14
3.2	Axis name .....	14
3.3	Coordinate system .....	14
3.3.1	Machine coordinate system .....	14
3.3.2	Reference point.....	14
3.3.3	Workpiece coordinate system .....	15
3.3.4	Absolute programming and relative programming .....	16
3.4	Mode and non-mode .....	17
3.5	Decimal point programming .....	18
4	Preparatory Function: G code .....	19
4.1	Classification of G code .....	19
4.2	Simple G code .....	21
4.2.1	Rapid positioning G00.....	21

4.2.2	Linear interpolation G01 .....	23
4.2.3	Circular (helical) interpolation G02/G03.....	24
4.2.4	Absolute/ incremental programming G90/G91.....	28
4.2.5	Dwell (G04).....	29
4.2.6	Unidirectional positioning (G60).....	30
4.2.7	System parameter online modification (G10).....	31
4.2.8	Workpiece coordinate system G54~G59.....	32
4.2.9	Additional workpiece coordinate system.....	34
4.2.10	Machine coordinate system selection G53 .....	35
4.2.11	Floating coordinate system G92 .....	35
4.2.12	Plane selection G17/G18/G19 .....	37
4.2.13	Polar coordinate system setup/cancel G16/G15.....	38
4.2.14	Scaling in plane G51/G50.....	40
4.2.15	Coordinate system rotation G68/G69 .....	43
4.2.16	Skip function G31 .....	47
4.2.17	Inch/metric conversion G20/G21 .....	48
4.2.18	Arbitrary angle chamfering/corner rounding.....	49
4.3	Reference point G code .....	50
4.3.1	Reference point return G28 .....	50
4.3.2	2 <sup>nd</sup> , 3 <sup>rd</sup> , 4 <sup>th</sup> reference point return G30 .....	52
4.3.3	Automatic return from reference point G29.....	52
4.3.4	Reference point return check G27 .....	53
4.4	Canned cycle G code .....	53
4.4.1	Rough milling of inner circular groove G22/G23 .....	58
4.4.2	Fine milling cycle within a circle G24/G25 .....	61
4.4.3	Outer circle finish milling cycle G26/G32 .....	62
4.4.4	Rectangular groove rough milling G33/G34.....	64
4.4.5	Inner rectangular groove finish milling cycle G35/G36 .....	66
4.4.6	Rectangle outside finish milling cycle G35/G36.....	67
4.4.7	High-speed peck drilling cycle G73 .....	69
4.4.8	Drilling cycle, spot drilling cycle G81 .....	71
4.4.9	Drilling cycle, counterboring cycle G82.....	72
4.4.10	Drilling cycle with chip removal G83 .....	74
4.4.11	Right-handed tapping cycle G84 .....	76
4.4.12	Left-handed tapping cycle G74 .....	78
4.4.13	Fine boring cycle G76.....	79
4.4.14	Boring cycle G85 .....	81
4.4.15	Boring cycle G86 .....	83
4.4.16	Boring cycle, back boring cycle G87.....	85
4.4.17	Boring cycle G88 .....	87
4.4.18	Boring cycle G89 .....	88
4.4.19	Left-handed rigid tapping G74.....	90
4.4.20	Right-handed rigid tapping G84.....	92

4.4.21	Peck drilling rigid tapping (chip removal) cycle .....	94
4.4.22	Canned cycle cancel G80 .....	96
4.5	Tool compensation G code .....	98
4.5.1	Tool length compensation G43, G44, G49 .....	98
4.5.2	Tool radius compensation G40/G41/G42 .....	102
4.5.3	Explanation of tool radius compensation .....	109
4.5.4	Corner offset circular interpolation (G39) .....	125
4.5.5	Tool offset value and number input by program (G10) .....	126
4.6	Feed G code .....	127
4.6.1	Feed mode G64/G61/G63 .....	127
4.6.2	Automatic override for inner corners (G62) .....	127
4.7	Macro G code .....	130
4.7.1	Custom macro .....	130
4.7.2	Macro variables .....	130
4.7.3	Custom macro call .....	135
4.7.4	Custom macro function A .....	136
4.7.5	Custom macro function B .....	141
<b>5</b>	<b>Miscellaneous Function M code .....</b>	<b>148</b>
5.1	M codes controlled by PLC .....	149
5.1.1	Forward and reverse rotation instructions (M03, M04) .....	149
5.1.2	Spindle stop (M05) .....	149
5.1.3	Cooling on and off (M08, M09) .....	149
5.1.4	A axis release and clamping (M10, M11) .....	149
5.1.5	Tool release and clamping (M16, M17) .....	150
5.1.6	Spindle orientation (M19) .....	150
5.1.7	Tool search instruction (M21, M22) .....	150
5.1.8	Magazine rotation instruction (M23, M24) .....	150
5.1.9	Rigid tapping (M29) .....	150
5.1.10	Lubricating on and off (M32, M33) .....	150
5.1.11	Helical chip remover on and off (M35, M36) .....	150
5.1.12	Mirror image instructions (M40, M41, M42, M43) .....	150
5.1.13	Spindle blowing on and off (M44, M45) .....	150
5.1.14	Auto tool change start and end (M50, M51) .....	150
5.1.15	Tool judging after tool change (M53) .....	151
5.2	M codes used by program .....	151
5.2.1	Program end and return (M30, M02) .....	151
5.2.2	Program dwell (M00) .....	151
5.2.3	Program optional stop (M01) .....	151
5.2.4	Subprogram calling (M98) .....	151
5.2.5	Program end and return (M99) .....	152
<b>6</b>	<b>S codes for Spindle Function .....</b>	<b>153</b>

6.1	Spindle analog control .....	153
6.2	Spindle switch volume control.....	153
6.3	Constant surface speed control (G96/G97) .....	153
<b>7</b>	<b>Feed Functions: F code.....</b>	<b>156</b>
7.1	Traverse.....	156
7.2	Cutting feedrate .....	156
7.2.1	Feed per minute (G94) .....	157
7.2.2	Feed per revolution (G95) .....	157
7.3	Tangential speed control.....	158
7.4	Feedrate override keys .....	158
7.5	Auto acceleration/deceleration .....	158
7.6	Acceleration/deceleration for corner of a block .....	159
<b>8</b>	<b>Tool Function .....</b>	<b>161</b>
8.1	Tool function .....	161
<b>III</b>	<b>OPERATION.....</b>	<b>162</b>
<b>1</b>	<b>Operator Panel.....</b>	<b>163</b>
1.1	Panel layout.....	163
1.2	Explanation of the panel function.....	163
1.2.1	LCD area .....	163
1.2.2	Edit area .....	163
1.2.3	Screen operation keys .....	164
1.2.4	Control area.....	165
<b>2</b>	<b>System Power On/Off and Safety Operations.....</b>	<b>169</b>
2.1	System power on .....	169
2.2	System power off .....	169
2.3	Safety operations.....	170
2.3.1	Reset operation .....	170
2.3.2	Emergency stop.....	170
2.3.3	Feed hold.....	171
2.4	Cycle start and feed hold .....	171
2.5	Overtravel protection .....	171
2.5.1	Hardware overtravel protection.....	171
2.5.2	Software overtravel protection .....	172
2.5.3	Release of the overtravel alarm .....	172
2.6	Stroke detection .....	172
<b>3</b>	<b>Interface Display and Data Modification and setting .....</b>	<b>176</b>
3.1	Position display.....	176
3.1.1	Four types of position display .....	176

3.1.2	Display of run time, part count, programming speed and override, actual speed etc. ....	178
3.1.3	Relative coordinate clearing and mediating .....	179
3.2	Program display .....	179
3.3	System display .....	181
3.3.1	Display. Modification and setting for parameter .....	182
3.3.1.2	Modification and setting of the parameter values .....	182
3.3.2	Display, modification and setting for macro variables .....	183
3.3.3	Display, modification and setting for offset .....	184
3.3.4	Display, modification and setting for screw pitch offset .....	186
3.4	PMC display .....	186
3.5	Setting display .....	188
3.5.1	Setting page .....	188
3.5.2	Parameter and program on-off page - .....	189
3.5.3	Coordinate setting interface .....	190
3.5.4	Data processing interface .....	192
3.5.5	Password authority interface .....	193
3.6	Graphic display .....	194
3.7	Diagnosis display .....	195
3.7.1	Diagnosis data display .....	196
3.7.2	Signal viewing .....	197
3.8	Alarm display .....	198
3.9	Help display (this function is invalid temporarily ) .....	199
<b>4</b>	<b>Manual Operation.....</b>	<b>200</b>
4.1	Coordinate axis movement .....	200
4.1.1	Manual feed .....	200
4.1.2	Manual rapid traverse .....	200
4.1.3	JOG feedrate and manual rapid traverse speed selection .....	200
4.1.4	Manual intervention .....	201
4.1.5	Workpiece alignment .....	202
4.2	Spindle control .....	204
4.2.1	Spindle CCW .....	204
4.2.2	Spindle CW .....	204
4.2.3	Spindle stop .....	204
4.2.4	Spindle auto gear shift .....	204
4.3	Other manual operations .....	205
4.3.1	Cooling control .....	205
4.3.2	Lubricating control .....	205
4.3.3	Work light control .....	205
<b>5</b>	<b>Step Operation.....</b>	<b>206</b>
5.1	Step feed .....	206



5.1.1	Selection of moving amount .....	206
5.1.2	Selection of moving axis and direction .....	206
5.1.3	Step feed explanation .....	206
5.2	Step interruption.....	207
5.3	Auxiliary control in Step mode .....	207
<b>6</b>	<b>MPG Operation .....</b>	<b>208</b>
6.1	MPG feed .....	208
6.1.1	Moving amount selection .....	208
6.1.2	Selection of moving axis and direction .....	208
6.1.3	Explanation of MPG feed.....	209
6.2	Control in MPG interruption .....	209
6.2.1	MPG interruption operation.....	209
6.2.2	Relation of MPG interruption with other functions.....	210
6.3	Auxiliary control in MPG mode.....	211
6.4	MPG drive function .....	211
<b>7</b>	<b>Auto Operation .....</b>	<b>212</b>
7.1	Selection of the auto run programs.....	212
7.2	Auto run start .....	212
7.3	Auto run stop .....	213
7.4	Auto running from an arbitrary block.....	214
7.5	Dry run .....	215
7.6	Single block running .....	215
7.7	Running with machine lock .....	216
7.8	Running with M.S.T. lock .....	216
7.9	Feedrate and rapid override in Auto run .....	216
7.10	Spindle override in Auto run .....	217
7.11	Background edit in Auto run .....	218
<b>8</b>	<b>MDI Operation .....</b>	<b>219</b>
8.1	MDI instructions input .....	219
8.2	Run and stop of MDI instructions.....	220
8.3	Words modification and clearing of MDI instructions .....	220
8.4	Modes changing .....	221
<b>9</b>	<b>Machine Zero Operation.....</b>	<b>222</b>
9.1	Conception of machine zero .....	222
9.2	Steps for machine zero .....	223
9.3	Machine zero steps by program.....	224
<b>10</b>	<b>Edit Operation .....</b>	<b>225</b>
10.1	Program edit .....	225

10.1.1	Program creation .....	225
10.1.2	Deletion of a single program .....	231
10.1.3	Deletion of all programs .....	232
10.1.4	Copy of a program .....	232
10.1.5	Copy and paste of blocks.....	233
10.1.6	Cut and paste of block .....	233
10.1.7	Replacement of the blocks.....	234
10.1.8	Rename of a program .....	234
10.1.9	Program restart .....	234
10.2	Program management .....	235
10.2.1	Program directory search.....	235
10.2.2	Number of the program stored .....	236
10.2.3	Memory capacity .....	236
10.2.4	Viewing of the program list.....	236
10.2.5	Program lock.....	237
<b>11</b>	<b>Communication .....</b>	<b>238</b>
11.1	Serial communication .....	238
11.1.1	Program start .....	238
11.1.2	Function introduction.....	239
11.1.3	Serial port data transmission.....	240
11.1.4	Serial port DNC on-line machining.....	245
11.2	USB communication.....	247
11.2.1	General and precautions .....	247
11.2.2	USB part program operation steps.....	247
11.2.3	DNC processing operation steps .....	248
11.2.4	U flash disk system exit .....	249
<b>APPENDIX 1</b>	<b>.....</b>	<b>250</b>
<b>GSK990MA PARAMETER LIST</b>	<b>.....</b>	<b>250</b>
1	Bit parameter .....	251
2	Number parameter .....	271
<b>APPENDIX 2</b>	<b>.....</b>	<b>298</b>

# I      **OVERVIEW**

# 1 Overview

## **This manual is comprised by following parts:**

### **I Overview**

It describes the chapter structure, system model available, relative instructions and the note.

### **II Programming**

It describes G functions and the programming format, characteristics and restrictions by NC language.

### **III Operation**

It describes the manual and auto operation, program input/output and editing methods.

### **Appendix**

It describes parameter list, alarm list and programming data table.

The manual is used for GSK990MA CNC system.

## **II            PROGRAMMING**

## 1

## General

## 1.1 Tool movement along workpiece contour —interpolation

## 1) Tool movement along a straight line

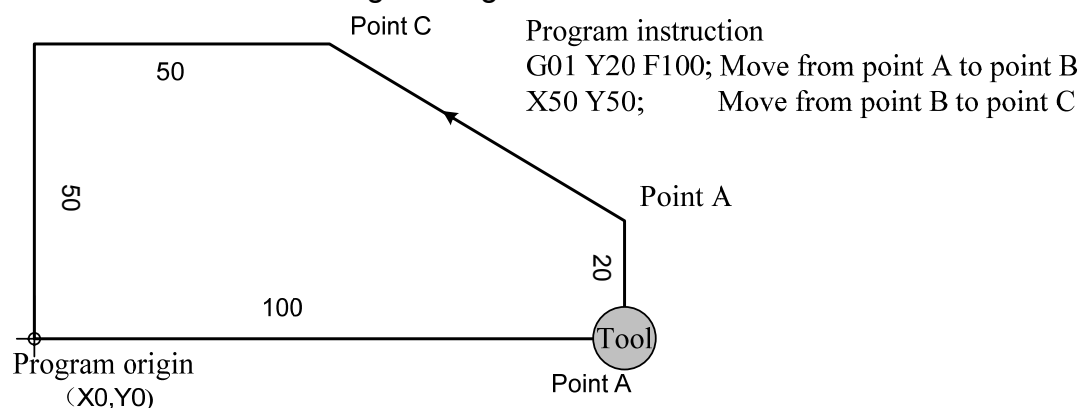


Fig. 1-1-1

## 2) Tool movement along an arc

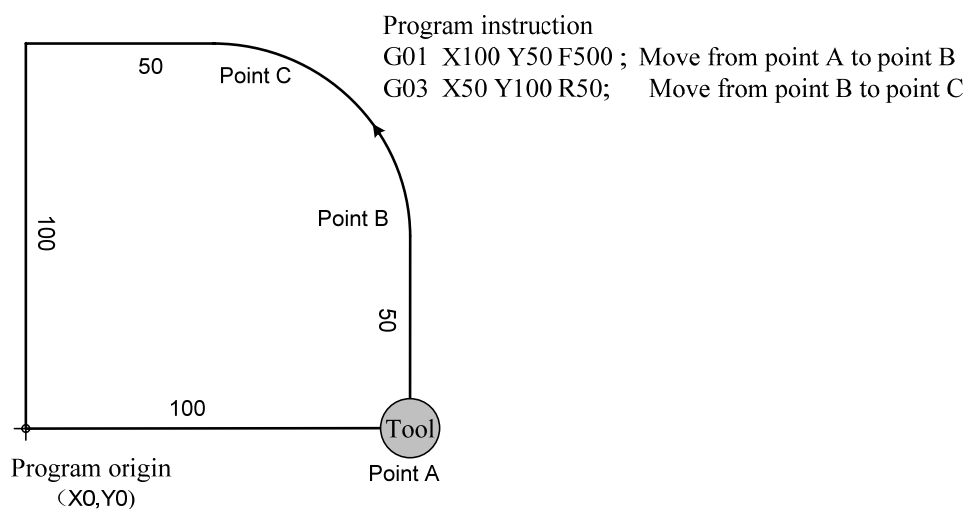


Fig. 1-1-2

The tool linear and arc motion function is called interpolation.

The programming instructions such as G01, G02 are called preparatory function, which is used for interpolation for CNC device.

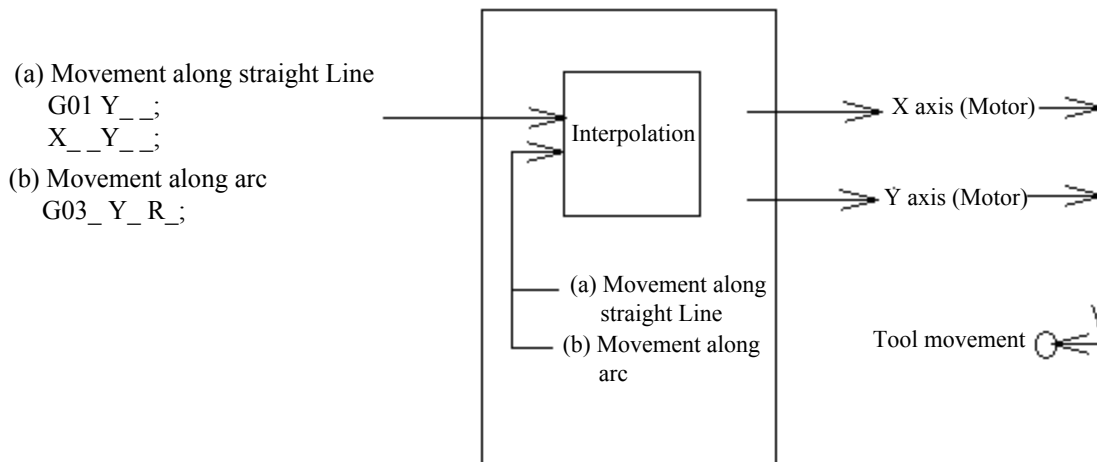


Fig. 1-1-3

**Note** For some machines, it is the worktable moving other than tool moving in practice. It is assumed that the tool moves relative to the workpiece in this manual. Refer to the machine actual movement direction in practice to protect against personnel hurt and machine damage.

## 1.2 Feed——Feed function

The feedrate specification is called feed function.

### 1. Rapid traverse

The rapid traverse is used to specify the rapid speed when G00 is used for positioning.

The rapid traverse speed of each axis is set by parameters, so it is unnecessary to specify it in the program.

### 2. Cutting feedrate

Moving a tool at a specified speed to cut a workpiece is called feed. The feedrate is specified with numerical values. E.g., the program code is F150 when the tool is moved at the speed of 150m/min.

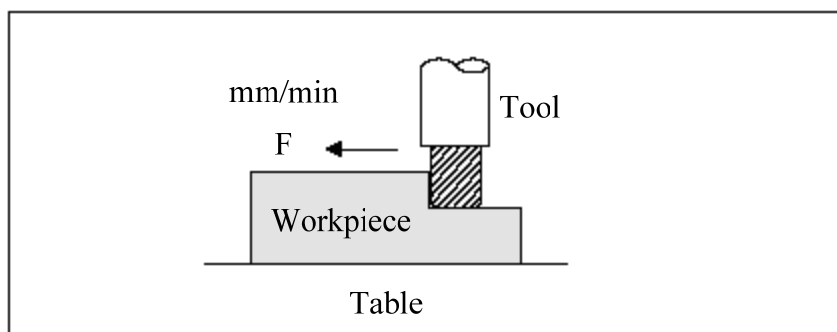
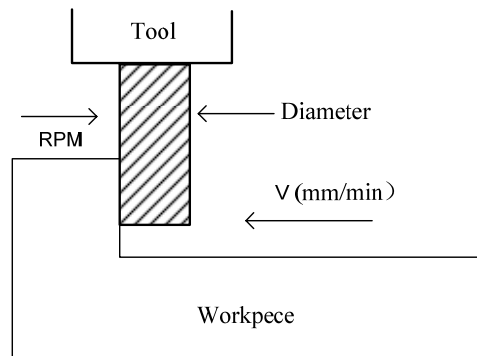


Fig. 1-2-1

### 1.3 Cutting feedrate, spindle speed function



**Fig.1-3-1**

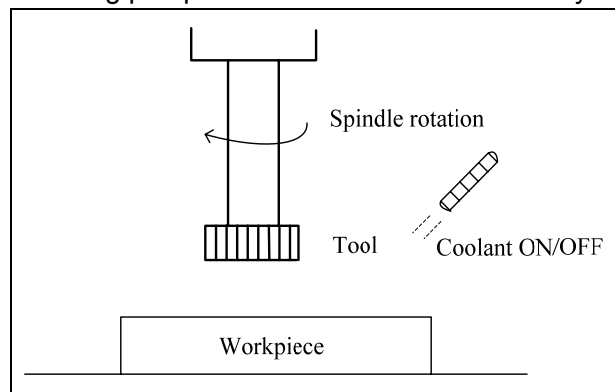
The speed of tool relative to workpiece in cutting is called cutting feedrate. It can be instructed by spindle speed RPM(r/min) by CNC.

Example: If the tool diameter is 10mm, cutting linear speed is 8 m/min, the spindle speed is about 255RPM according to  $N=1000V/\pi D$ , so the instruction is: S255

Instructions related to spindle speed are called spindle speed function.

### 1.4 Operation instruction——miscellaneous function

When the workpiece is to be machined, to make the spindle run and supply coolant, the machine spindle motor and cooling pump switches must be controlled by actual requirement.



**Fig. 1-4-1**

The programs or machine on-off actions controlled by system NC instructions are called miscellaneous functions, which are instructed by M code.

Example: If M03 is instructed, the spindle rotates clockwise by the speed specified. (Clockwise direction means the direction viewed from the spindle -Z negative direction.)

### 1.5 Tool selection for various machining——Tool function

It is necessary to select a proper tool when drilling, tapping, boring, milling, etc. is performed. When a number is assigned for each tool and the number is specified in the program, the corresponding tool is selected.



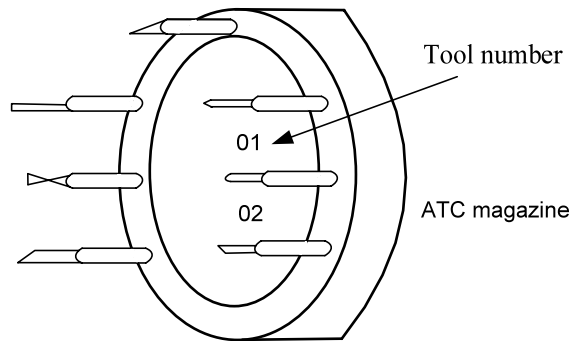


Fig. 1-5-1

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

## 1.6 Tool figure and tool motion by program

### 1.6.1 Tool length compensation

Usually several tools are used for machining one workpiece. If instructions such as G0 Z0 are executed in a same coordinate system, because tools have different tool lengths, the distances from tool end to workpiece are different. It is very troublesome to change the program frequently.

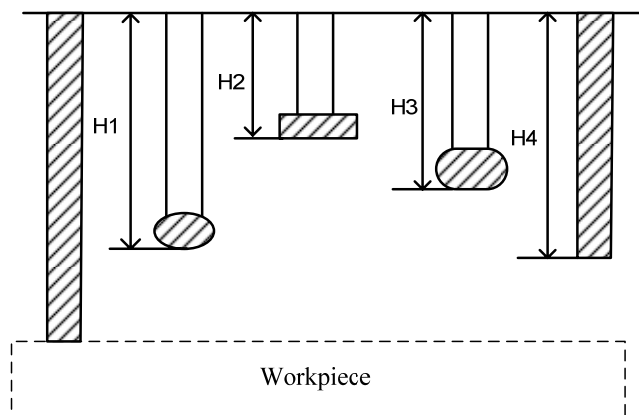


Fig. 1-6-1-1

Therefore, the length of each tool used should be measured in advance. By setting the difference between the length of the standard tool and the length of each tool in the CNC (usually the 1st tool), machining can be performed without altering the program even when the tool is changed. After the tool positioning in Z axis (e.g. G0Z0), the distances of the tool end to the workpiece are identical. This function is called tool length compensation.

### 1.6.2 Tool radius compensation

Because a tool has a radius, if the tool goes by the path given by program, the workpiece will be cut off a part for a radius wide. To simplify the programming, the program can be ran by CNC around the workpiece with the tool radius deviated, while the transient path of the intersections of the lines or the arcs can be processed automatically by system.

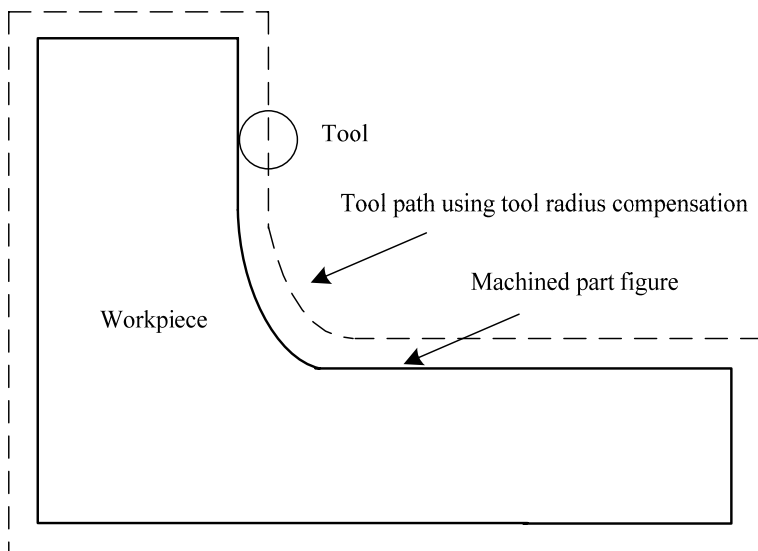


Fig.1-6-2-1

If diameters of tools are stored in the CNC tool compensation list, the tool can be moved by tool radius apart from the machining part figure by calling different radius compensation according to program. This function is called tool radius compensation.

## 1.7 Tool movement range—stroke

The overtravel limit switches are fixed at the positive and negative maximum stroke of the machine X, Y, Z axis respectively. If the overtravel occurs, the moving axis slows down and stops after it touches the limit switch, and the overtravel alarm is issued. This function is usually called hardware limit.

The parameter setting can specify the safe tool running range, if the tool exceeds the range, the system stops all the axes moving with overtravel alarm given. This function is called stroke verification, namely, the software limit.

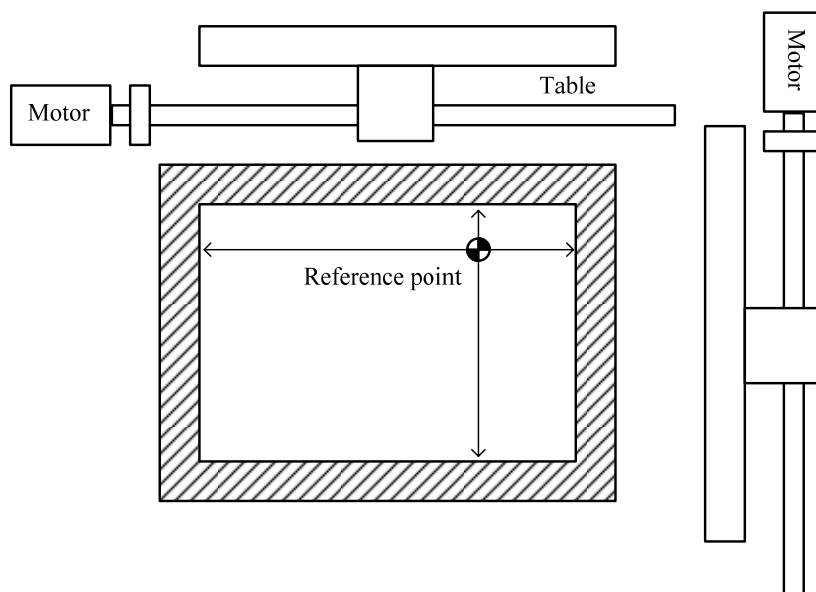


Fig. 1-7-1

## 2 Part Program Composition

### 2.1 Program composition

A program is composed by many blocks which are formed by words. The blocks are separated by the end code (LF for ISO, CR for EIA). In this manual the end code is represented by “;” character.

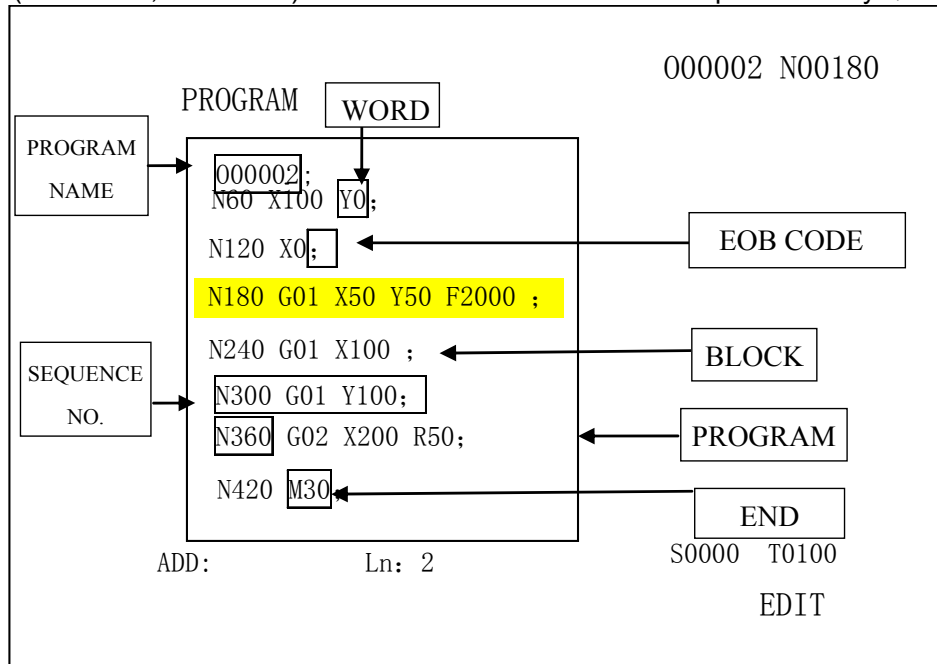


Fig. 2-1-1 Program structure

The set instructions to control the CNC machine tool to machine the parts are called program. After the program edited is entered into the CNC system, the system controls the tool to move along straight line, arc or make the spindle run or stop by these instructions. And the instructions should be edited by the machine actual movement sequence. The program structure is shown in Fig.2-1-1.

#### 2.1.1 Program name

In this system the system memory may store many programs. In order to differentiate these programs, address O with five figures behind it is headed in the beginning of the program. And it is shown in Fig. 2-1-1-1.

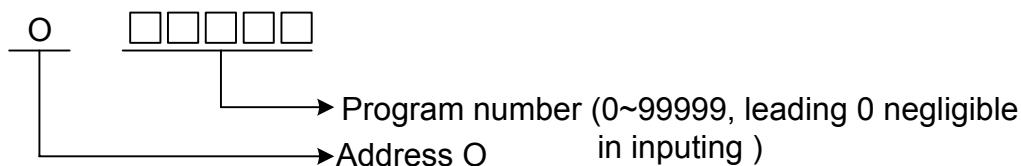


Fig.2-1-1-1 Program name composition

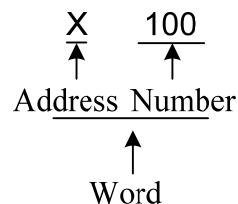
### 2.1.2 Sequence number and block

A program consisted of many instructions, and an instruction unit is called block (see **Fig. 2-1-1**). The blocks are separated by program end code (see **Fig. 2-1-1**). In this manual the block end code is represented by character“; ”.

Address N with 4 figures sequence number behind it can be used at the beginning of the block (see **Fig. 2-1-1**), and the leading zero can be omitted. The sequence of the sequence number (insertion set by bit parameter No. 0 # 5) can be arbitrary, and the intervals between them can be inequal (set by Parameter P210). Sequence number can be either in all blocks, or in some important blocks. But by common machining sequence, the number should be arranged by ascending. That the sequence number is placed in important part of the program is for convenience. (e.g. in tool changing, or worktable indexed to a new plane).

### 2.1.3 Instruction word

Word (Fig. 2-1-3-1) is a factor to block composition. It is formed by an address and figures behind it (sometimes +, - added before figures)



**Fig.2-1-3-1 Word composition**

The address is a character from English alphabetic table which defines the meaning of the figure behind it. In this system, the usable addresses and their meaning as well as value range are shown as **Table2-1-3-1**:

Sometimes an address has a different meaning for different preparatory function.

If 2 or more identical addresses appear in an instruction, the alarm for it will be set by parameter **NO. 32#6**.

**Table 2-1-3-1**

Address	Range	Meaning
O	0~99999	Program name
N	0~99999	Sequence number
G	00~99	Preparatory function
X	-99999.999~99999.999 (mm)	X coordinate address
	0.001~9999.999 (s)	Dwell time
Y	-99999.999~99999.999 (mm)	Y coordinate address
Z	-99999.999~99999.999 (mm)	Z coordinate address
R	-99999.999~99999.999 (mm)	Arc radius/angle displacement
	-99999.999~99999.999 (mm)	R level in canned cycle
I	-99999.999~99999.999 (mm)	vector of arc center to start point in X axis
J	-99999.999~99999.999 (mm)	vector of arc center to start point in Y axis
K	-99999.999~99999.999 (mm)	vector of arc center to start point in Z axis
F	0~99999 (mm/min)	Feed in a minute
	0.001~500(mm/r)	Feed in a revolution
S	0~99999 (r/min)	Spindle speed
	00~04	Multi-gear spindle output

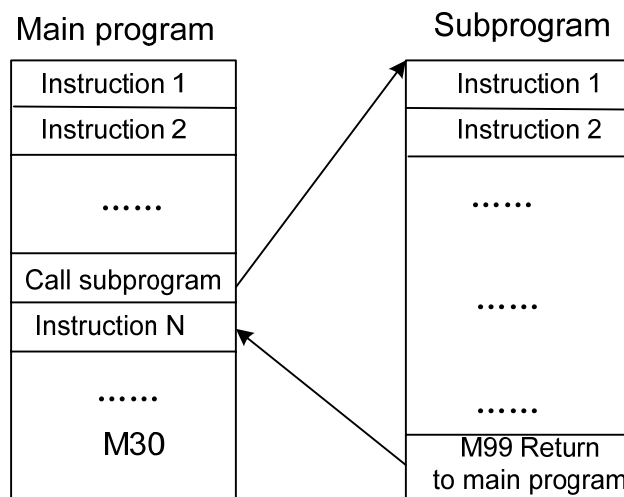
Address	Range	Meaning
T	0~9999	Tool function
M	00~99	Miscellaneous function output, program executing process, subprogram calling
P	1~99999.9999 (ms)	Dwell time
	1~99999	Subprogram number calling
Q	-99999.9999~99999.9999 (mm)	Cutting depth or hole bottom offset in canned cycle
H	01~99	Operator for G65
	00~256	Length offset number
D	00~256	Radius offset number

Special attention should be paid that the limits in table 2-1-3-1 are all for CNC device, but not for machine tool. Therefore, programming should be done on a basis of good understanding of the programming limitation of machine builder manual besides this manual.

**Note:** each word should not exceed 79 characters.

## 2.2 General structure of a program

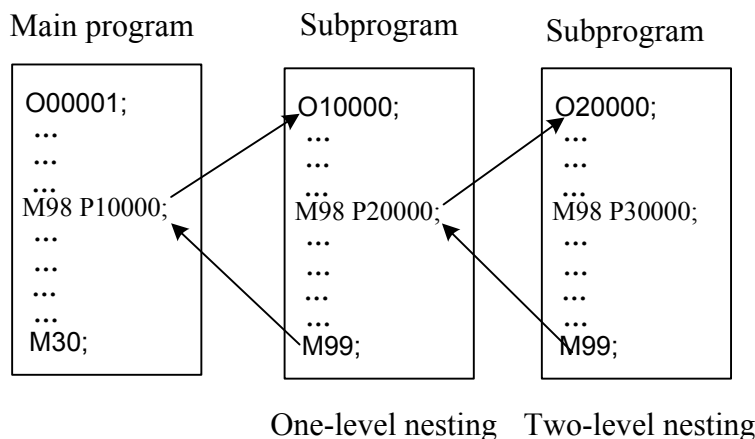
The program is classified for main program and subprogram. Generally, the CNC system is acutated by the main program. If the main program contains the subprogram call, the CNC system acts by the subprogram. If the subprogram contains the instruction of returning to main program, the CNC system returns to the main program to go on execution. The program execution sequence is shown as **Fig.2-2-1**.



**Fig. 2-2-1**

The structure of the subprogram is same as that of the main program.

If there are fixed sequence blocks occurring repeatedly in a program, it can be taken as a subprogram which can be stored in the memory in advance with no need to be edited repeatedly. So it can simplify the program. The subprogram can be called in Auto mode, usually by M98 in the main program. And the subprogram called can also call other subprograms. The subprogram called from the main program is called the 1<sup>st</sup> level subprogram. 4 levels subprogram at most can be called in a program (**Fig.2-2-2**). The last block in the subprogram must be the returning instruction M99. After M99 execution, the control returns to next block following the block that calls the subprogram in the main program to go on execution. If the main program end is M99, the program execution can be repeated.

**Fig. 2-2-2** Two-level subprogram nesting

A single subprogram call instruction can be continuously and repeatedly used to call a subprogram up to 9999 times.

### 2.2.1 Subprogram edit

Write out a subprogram by following format:

```

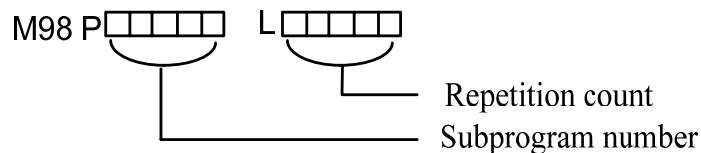
O  □□□□□ ; Subprogram number
    .....
    .....
    .....
    M99;      Subprogram end
  
```

**Fig. 2-2-1-1**

Write out the subprogram number behind the address O at the subprogram beginning, and the M99 instruction at the subprogram end (M99 format as above).

### 2.2.2 Subprogram call

The subprogram is called out for execution by the main program or the subprogram. The instruction format is as following:

**Fig. 2-2-2-1**

- If the repeat time is omitted, the default is 1.  
Example M98 P1002L5 ; (It means No.1002 subprogram is continuously called for 5 times.)
- Execution sequence of subprogram call from main program

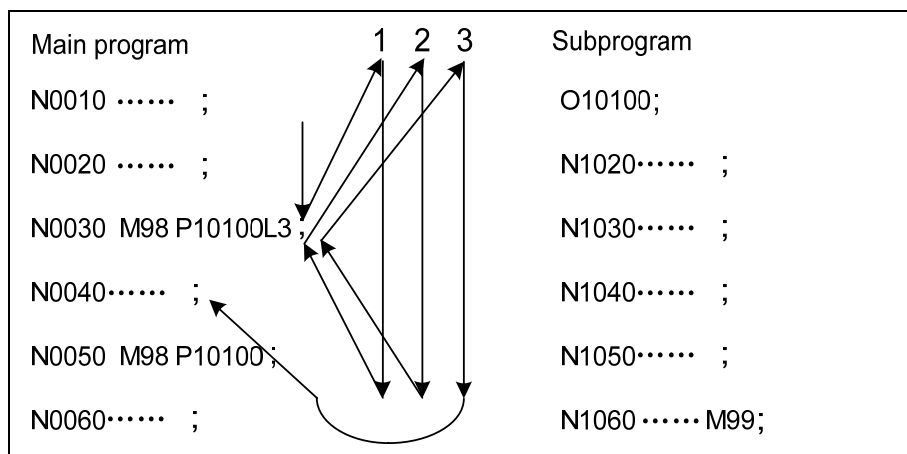


Fig. 2-2-2-2

Subprogram call from subprogram are identical with that from main program.

- Note**
- 1、Alarm (PS 078) occurs if subprogram number specified by address P is not found.
  - 2、No. 90000~99999 subprograms are the system reserved programs, if they are called, they can be executed, and can be displayed.

### 2.2.3 Program end

The program begins with program name, ends with M02, M30 or M99 (see Fig. 2-2-2-2). For the end code M02,,M30 or M99 detected in program execution: if M02, M30 specifies the end, the program finishes and reset; and M30 can be set by bit parameter N0.33#4 for returning to the program beginning, and M02 can be set by bit parameter N0.33#2 for returning to the program beginning. if M99 specifies the end, the control returns to the program beginning to restart the program; if M99 、M02 and M30 is at the end of the subprogram, the control returns to the program that calls the subprogram and go on executing the following block.

# 3 Programming Fundamentals

## 3.1 Controlled axis

Table 3-1-1

Item	GSK990MA
Basic controlled axes	3 axes (X, Y, Z)
Extended controlled axes (total)	4 axes

## 3.2 Axis name

The 3 primary axis names are always X, Y, or Z. And the controlled axes are set by data parameter **P005**. The additional axis names are set by data parameter **P006** accordingly, such as A, B, C.

## 3.3 Coordinate system

### 3.3.1 Machine coordinate system

A special point on machine used as machine benchmark is called machine zero, which is set by the machine builder. The coordinate system set by machine zero taken as origin is called machine coordinate system. It is set up by manual machine zero return after power on. Once set, it remains unchanged till the power off, system reset or emergency stop.

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

### 3.3.2 Reference point

There is a special point on CNC machine tool for tool change and coordinate system setup, which is called reference point. It is a fixed point in machine coordinate system set by machine builder. By reference point return, the tool can easily move to this position. Generally this point in CNC milling system coincides with the machine zero, while the reference point of Machine Center is usually the tool change point.



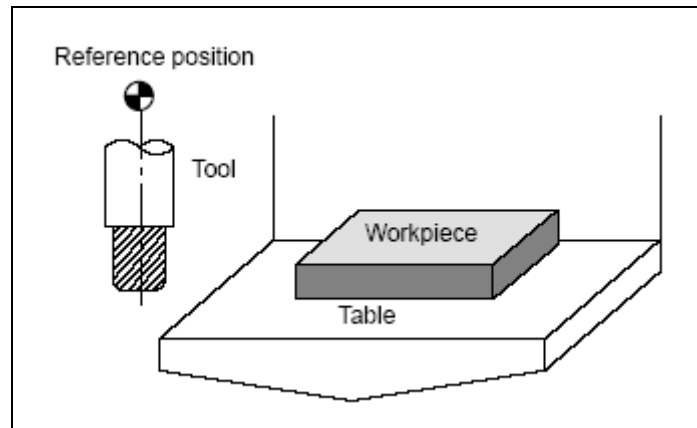


Fig. 3-3-2

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

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

### 3.3.3 Workpiece coordinate system

The coordinate system used for workpiece machining is called workpiece coordinate system (or part coordinate system), which is preset by CNC system (to set workpiece coordinate system).

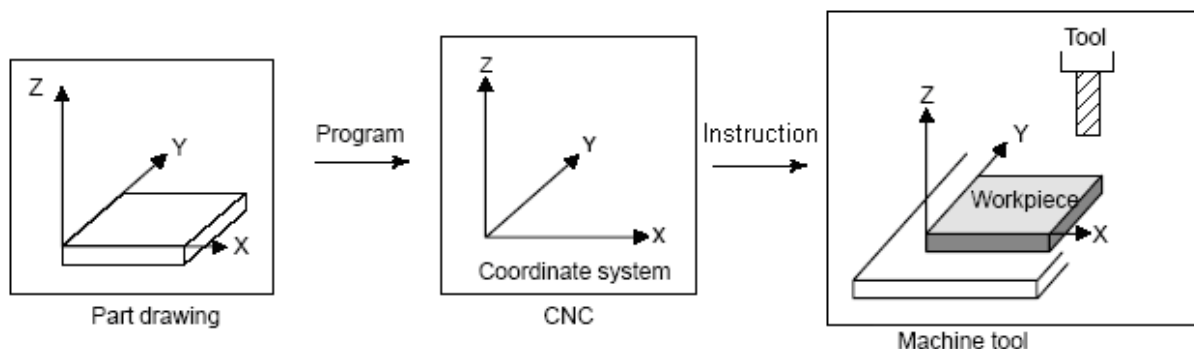


Fig. 3-3-3-1

In order to make the tool to cut the workpiece to the figure on drawing by instruction program according to drawing in the workpiece coordinate system specified by CNC, the relation of the machine coordinate system and the workpiece coordinate system must be determined.

The method to determine the relation of these two coordinate systems is called alignment. It can be done by different methods such as part figure, workpiece quantity.

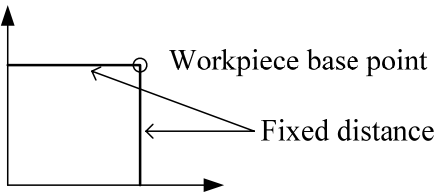
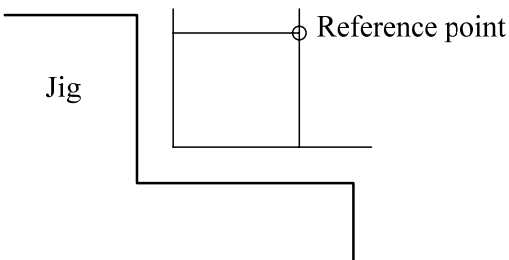
I) By workpiece base point	II) When part is fixed on jig
	
<p>To align the tool center to the workpiece base point, specify the workpiece coordinate system by CNC at this position, and the workpiece coordinate system coincides with the programming coordinate system</p>	<p>Because the tool center can't be located at the workpiece base point, locate the tool to a position (or reference point) that has a distance to the base point, set the workpiece coordinate system by this distance (e.g. G92)</p>

Fig. 3-3-3-2

Workpiece coordinate system should be set by each processing program (to select a workpiece coordinate system). The workpiece coordinate system set can be changed by moving its origin.

There are two methods to set the workpiece coordinate system:

1. By G92, see 4.2.11 for details.
2. By G code from 54 to 59, see 4.2.8 for details.

### 3.3.4 Absolute programming and relative programming

There are absolute and relative definitions to define the axis moving. The absolute definition is the method of programming by the axis moving final point, which is called absolute programming. The relative definition is the method of programming by the axis moving, which is called relative programming (call incremental programming).

#### 1) Absolute coordinate

It is the target position coordinate in the specified workpiece coordinate system, namely the position the tool to move to.

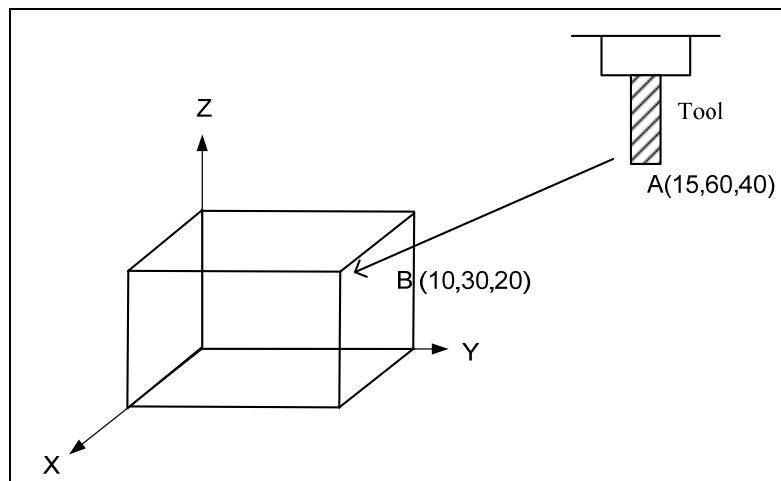


Fig.3-3-4-1

Move the tool from point A to point B, using the B coordinate in G54 workpiece coordinate system, the instruction is as following:

G90 G54X10 Y30 Z20 ;

## 2) Incremental coordinate

It is the target position coordinate relative to the current position by taking the current position as the origin.

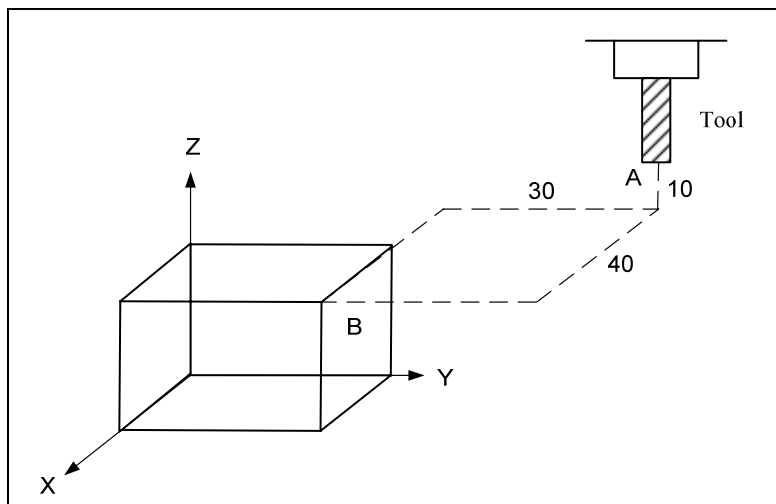


Fig. 3-3-4-2

For traversing the tool from point A to point B, the instruction is as following:

G0 G91 X-40 Y-30 Z-10;

## 3.4 Mode and non-mode

The mode means that the address value set by a block is effective till it is reset by another block. Another significance of it is that if a functional word is set, it doesn't need to be input again if it is used in the following blocks.

### ➤ e.g. for following program:

G0 X100 Y100; (rapid positioning to the location X100 Y100)

X20 Y30; (rapid positioning to the location X120 Y30, G0 specified by mode can be omitted)

G1 X50 Y50 F300; (interpolate to location X50 Y50 by straight line with the feedrate 300mm/min G0→G1 )

X100; (interpolate to location X100 Y50 by straight line with the feedrate 300mm/min , G1, Z50,F300 are all specified by mode and can be omitted )

G0 X0 Y0; (rapid positioning to the location X0 Y0)

The initial state is the default state after the system power-on. See table 4-1-2.

### ➤ For following program:

O00001

X100 Y100; (rapid positioning to the location X100 Y100, G0 is the initial state)

G1 X0 Y0 F100; (interpolate to location X0 Y0 by straight line with the feedrate 100mm/min, G98 is the initial power-on state )

Non-mode means that the relevant address value is effective only in the block contains this address, if it is used in following blocks, it must be specified again. e.g. G functional instructions of 00 group in Table 4-1-2.

Refer to **Table 3-4-1** for mode and non-modal description for functional word.

**Table 3-4-1 Mode and non-mode for functional instruction**

Mode	Modal G function	A group of G functions that can be cancelled by each other, once executed, they are effective till they are cancelled by other G functions in the same group.
	Modal M function	A group of M functions that can be cancelled by each other, once executed, they are effective till they are cancelled by other G functions in the same group.
Non-mode	Non-modal G function	They are only effective in the block they are specified and cancelled at the block end.
	Non-modal M function	They are only effective in the block they are specified.

### 3.5 Decimal point programming

Numerical values can be entered with a decimal point. A decimal point can be used when entering a distance, time, or speed. Decimal points can be specified with the following addresses:

X, Y, Z, A, B, C, I, J, K, R, P, Q, and F.

**Explanation:**

- 1、 The decimal point programming are set by bit parameter NO.33#1. If bit parameter NO.33#1=1, the programming value unit is mm, inch, or deg; if bit parameter NO.33#1=0, the programming value unit is the min. moving unit which is set by bit parameter NO.5#1.
- 2、 The decimal part that is less than the min. input incremental unit should be omitted.

**Example:**

X9.87654; When the min. input incremental unit is 0.001mm, it should be X 9.876.  
When the min. input incremental unit is 0.0001mm, it should be X 9.8765.

# 4 Preparatory Function: G code

## 4.1 Classification of G code

Preparatory function is represented by G code with the number behind it, which defines the meaning of the block that contains it. G codes are divided by the following two types:

**Table 4-1-1**

Classification	Meaning
Non-modal G code	Only effective in the block in which it is specified
modal G code	Effective till another G code of the same group is specified

**Example** G01 and G00 are modal G code in the same group.

G01 X \_\_\_ ;  
 Z \_\_\_\_\_ ; G01 effective  
 X \_\_\_\_\_ ; G01 effective  
 G00 Z\_\_\_; G00 effective

**Note** Refer to system parameter list (modal list) for details.

**Table 4-1-2 G codes and their functions**

G code	Group	Instruction format	Function
*G00	01	G00 X_Y_Z_	Positioning (rapid traverse)
G01		G01 X_Y_Z_F_	Linear interpolation(cutting feed)
G02		G02 X_Y_ R_ F_;	Circular interpolation CW
G03		G03 X_Y_ I_J_ F_;	Circular interpolation CCW
G04	00	G04 P_ or G04 X_	Dwell, exact stop
G10	00	G10L_; N_P_R_	Programmable data input
*G11	00	G11	Programmable data input cancel
*G12	16	G12 X_Y_Z_ I_J_K_	Storage stroke detection on
G13		G13 X_Y_Z_ I_J_K_	Storage stroke detection off
*G15	11	G15	Polar coordinate instruction cancel
G16		G16	Polar coordinate instruction
*G17 G18 G19	02	Write in with other program in block, used for circular interpolation and tool radius compensation	XY plane selection ZX plane selection YZ plane selection
G20	06	Specified by a single block at the program beginning before the coordinate system set	Inch input
*G21			Metric input
G22	09	G22_X_Y_Z_R_I_L_W_Q_V_D_F_K	CCW inner circular groove rough milling
G23		G23_X_Y_Z_R_I_L_W_Q_V_D_F_K	CCW inner circular groove rough milling
G24		G24 X_Y_Z_R_I_J_D_F_K	CCW fine milling within a circle
G25		G25 X_Y_Z_R_I_J_D_F_K	CW fine milling within a circle

G26	00	G26 X_Y_Z_R_I_J_D_F_K			CCW outer circle finish milling cycle
G27		X_Y_Z_		Reference point return detection	
G28				Reference point return	
G29				Return from reference point	
G30				2 <sup>nd</sup> ,3 <sup>rd</sup> , 4 <sup>th</sup> reference point return	
G31				Skip function	
G32	09	G32 X_Y_Z_R_I_J_D_F_K			CCW Outer circle finish milling cycle
G33		G33 X_Y_Z_R_I_L_W_Q_V_U_D_F_K			CW rectangular groove rough milling
G34		G34_X_Y_Z_R_I_L_W_Q_V_U_D_F_K			CCW rectangular groove rough milling cycle
G35		G35_X_Y_Z_R_I_J_L_U_D_F_K			CW inner rectangular groove finish milling cycle
G36		G36_X_Y_Z_R_I_J_L_U_D_F_K			CCW inner rectangular groove finish milling cycle
G37		G37_X_Y_Z_R_I_J_L_U_D_F_K			CW rectangle outside finish milling cycle
G38		G38_X_Y_Z_R_I_J_L_U_D_F_K			CCW rectangle outside finish milling cycle
G39	00	G39	I_J_ ; I_J_ ; J_K_ or G39		Corner offset circular interpolation
*G40	07	G17	G40 G41 G42	X_Y_	Tool radius compensation cancel
G41		G18		X_Z_	Left-hand tool radius compensation
G42		G19		Y_Z_	Right-hand tool radius compensation
G43	08	G43		Z_	Positive tool length compensation
G44		G44			Negative tool length compensation
*G49		G49			Tool length compensation cancel
*G50	12	G51			Scaling cancel
G51		G51 X_Y_Z_P_			Scaling
G53	00	Write into the program			Machine coordinate system selection
*G54	05	Write into the block with other program, usually placed at the program beginning			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
G60	00	G60 X_Y_Z_F_			Unidirectional position
G61	14	G61			Exact stop mode
G62		G62			Automatic corner override
G63		G63			Tapping mode
*G64		G64			Cutting mode
G65	00	G65 H_P# i Q# j R# k			Macro program instruction
G68	13	G68 X_Y_R_			Coordinate system rotation
*G69		G69			Coordinate system rotation cancel
G73	09	G73 X_Y_Z_R_Q_F_;			Peck drilling cycle
G74		G74 X_Y_Z_R_P_F_;			Lef-hand tapping cycle
G76		G76 X_Y_Z_R_P_F_K_;			Fine boring cycle
*G80		Write into the block with other program			Canned cycle cancel
G81		G81 X_Y_Z_R_F_;			Drilling cycle(spot drilling cycle)
G82		G82 X_Y_Z_R_P_F_;			Drilling cycle (counter boring cycle)
G83		G83 X_Y_Z_R_Q_F;			Peck drilling cycle

G84		G84 X_Y_Z_R_P_F_;	Tapping cycle
G85		G85 X_Y_Z_R_F_;	Boring cycle
G86		G86 X_Y_Z_R_F_;	Drilling cycle
G87		G87 X_Y_Z_R_Q_P_F_;	Back boring cycle
G88		G88 X_Y_Z_R_P_F_;	Boring cycle
G89		G89 X_Y_Z_R_P_F_;	Boring cycle
*G90	03	Write into the block with other program	Absolute programming
G91			Incremental programming
G92	00	G92 X_Y_Z_	Coordinate system set
*G94	04	G94	Feed per minute
G95		G95	Feed per revolution
G96	15	G96S_	Constant surface speed control (cutting speed)
*G97		G97S_	Constant surface speed control cancel (cutting speed)
*G98	10	Write into the block with other program	Return to initial point in canned cycle
G99			Return to point R level (in canned cycle)

- Note:**
- 1、if modal instruction and non-modal instruction share the same block, the non-modal instruction is in priority, and the corresponding mode should be changed according to the other modal instructions in the block, but the modal instructions are not executed .
  - 2、For the G code with \* sign, when the power is switched on, the system is in the state of this G code.
  - 3、G codes except G10, G11 in 00 group are all non-modal G code.
  - 4、Alarm occurs if G code not listed in this table is used or G code without the selection function is specified.
  - 5、G codes from different groups can be specified in a block, but 2 or more G codes from the same group can't be specified in a block, otherwise alarm or tool abnormality occurs.
  - 6、If G codes from 01 group and 09 group share the same block, the G code from 01 group will be taken. In canned cycle, if G code from 01 group is specified, the canned cycle will be cancelled automatically and the system turns into G80 state.
  - 7、G codes are represented by group numbers respectively according to their types. All G codes can be cleared by bit parameter No.35#0~7 and No.36#0~7 setting at system reset and emergency stop.
  - 8、if rotation scaling instruction and the instruction of 01 group or that of 09 group share the same block, the rotation scaling instruction will be taken, and the 01 group or 09 group modes should be changed. If the rotation scaling instruction and instruction of 00 group share the same block, alarm occurs.

## 4.2 Simple G code

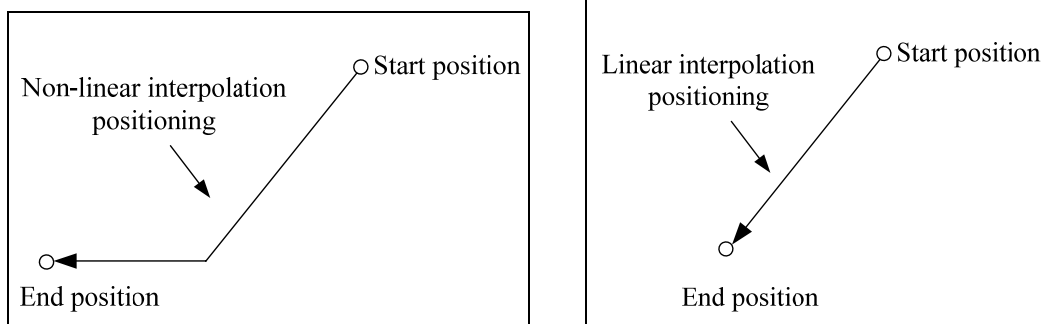
### 4.2.1 Rapid positioning G00

**Format:** G00 X\_Y\_Z\_

**Function:** G00 instruction moves the tool to the position in the workpiece system specified with an absolute or an incremental instruction at a traverse speed by linear interpolation. It is set by bit parameter NO.12#1 and uses one of the following two paths.

( e.g. **Fig. 4-2-1-1**).

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



**Fig. 4-2-1-1**

**Explanation:**

- 1 After G00 is executed, the system changes the tool current move mode for G00 mode. The G00 (parameter value is 0) or G01 (parameter value is 1) default mode can be set by bit parameter No.031#0 while the power is switched on.
- 2 The tool doesn't move if positioning parameter is not specified, and the system only change the current tool move mode for G00.
- 3 G00 is identical with G0.
- 4 G0 speed for X, Y, Z axis is set by data parameter P88~P92.

**Restrictions**

The traverse speed is set by parameter, if F is specified in G0 instruction, it is used for the following cutting feedrate. For example:

G0 X0 Y10 F800; rapid traversing by system parameter set

G1 X20 Y50; by F800 feedrate

The rapid feedrate is adjusted by the key on operator panel with following override : F0, 25, 50, 100%, see Fig. 4-2-1-2. The speed for F0 is set by number parameter P93, and they are used by all axes.



**Fig. 4-2-1-2 Rapid feedrate override key**

**Note:** attention for the positions of the workpiece and worktable when programming should be paid to protect against the tool collision.



### 4.2.2 Linear interpolation G01

**Format:** G01 X\_ Y\_ Z\_ F\_

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

**Explanation:**

- 1 X\_ Y\_ Z\_ are the final point coordinate which concerns the coordinate system, refer to 3.3.1~3.3.3 sections.
- 2 The feedrate specified by F is effective till the new F code is specified. The feedrate by F code is got by an interpolation along a line, if F code is not specified in program, the feedrate uses the default value when the power is on.(see number parameter P87 for the setting)

Program example (see **Fig. 4-2-2-1**)

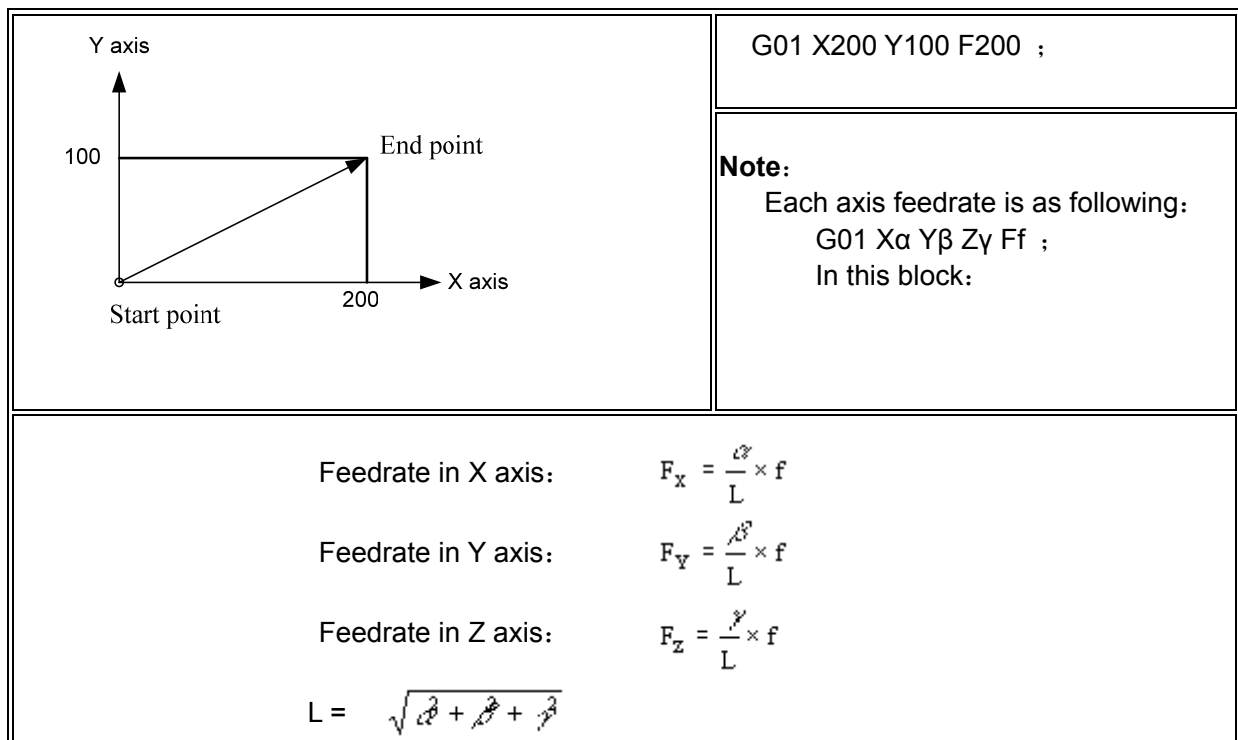


Fig. 4-2-2-1

**Note:**

1. The instruction parameters except F are all positioning parameter. And the upper limit of the feedrate F can be set by number parameter P94. If the actual feedrate (using override) exceeds the upper limit, it is restricted to the upper limit and its unit is mm/min. The lower limit of the feedrate F can be set by number parameter P95. If the actual feedrate (using override) exceeds the lower limit, it is restricted to the lower limit and its unit is mm/min.
2. If the positioning parameter behind G01 is not specified, the tool doesn't move, and the system only changes the tool current mode for G 01 mode. The system default mode at power-on can be set for G00 (value is 0) or G01 (value is 1) by altering the system bit parameter NO.31#0.

### 4.2.3 Circular (helical) interpolation G02/G03

#### A Circular interpolation G02/G03

Prescriptions for G02/G03:

The plane circular interpolation means that the arc path is to be finished by the specified rotation and radius or circle center from the start point to the end point in the specified plane.

Because the arc path can't be defined only by the start point and the end point, other conditions are needed:

- Arc rotation direction (G02, G03)
- Circular interpolation plane (G17, G18, G19)
- Circle center coordinate or radius, which gives two programming format: Circle center coordinate I, J, K or radius R programming

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

The circular interpolation can be done by the following instructions to make the tool to go along an arc, it is shown as follows:

Arc in XY plane

```
G17 G02 X_Y_ R_ F_;
      G03 I_J_
```

Arc in ZX plane

```
G18 G02 X_Z_ R_ F_;
      G03 I_K_
```

Arc in YZ plane

```
G19 G02 Y_Z_ R_ F_;
      G03 J_K_
```

**Table 4-2-3-1**

Item	Content	Instruction	Description
1	To specify plane	G17	Arc specification on XY plane
		G18	Arc specification on ZX plane
		G19	Arc specification on YZ plane
2	To specify rotation direction	G02	CW
		G03	CCW
3	G90 Final position	Two axes of X,Y, Z axis	End point coordinate in workpiece coordinate system
	G91	Two axes of X,Y, Z axis	Coordinate of end point relative to start point
4	Distance from start point to circle center	Two axes of I,J, K axis	Coordinate of circle center relative to start point
	Arc radius	R	Arc radius
5	Feedrate	F	Arc tangential speed

CW and CCW mean the directions viewed from the positive Z (or Y, Z) axis to the negative in the right-hand Cartesian coordinate system regarding to XY ( or ZX, YZ) plane , as shown in **Fig. 4-2-3-1**.

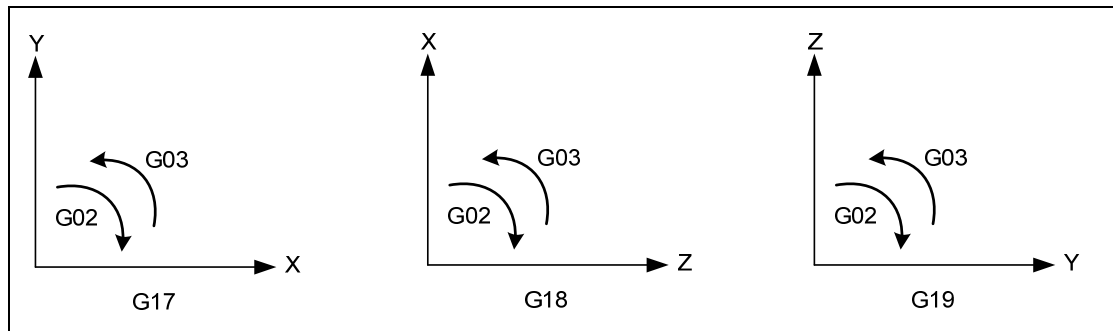


Fig. 4-2-3-1

The default plane mode at power-on can be set by bit parameters **NO.31#1, #2, #3**.

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

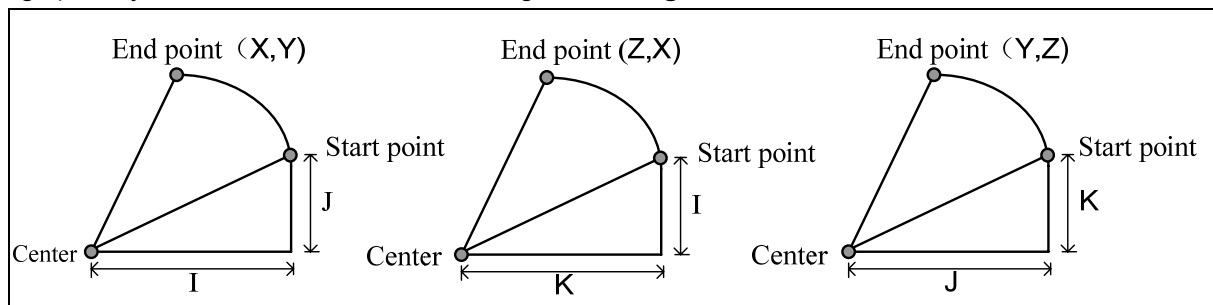


Fig. 4-2-3-2

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

G02 X\_ Y\_ R\_ ;  
G03 X\_ Y\_ R\_ ;

- Two arcs can be drawn out as following, one arc is more than 180°, the other is less than 180°. The radius of the arc more than 180° should be specified by a negative value.

(e.g. Fig. 4-2-3-3)

as arc ① is less than 180°  
G91 G02 X60 Y20 R50 F300 ;  
as arc ② is more than 180°  
G91 G02 X60 Y20 R-50 F300 ;

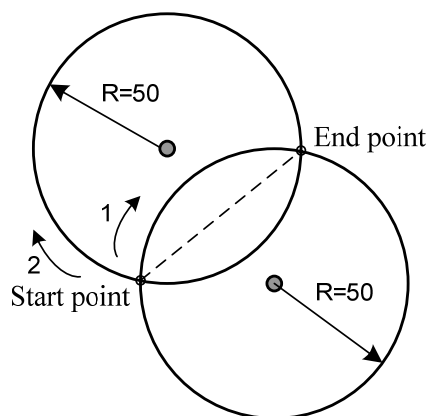


Fig. 4-2-3-3

- 2 The arc equal to  $180^\circ$  can be programmed either by I, J, K, or by R.

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

Equal to G90 G0 X0 Y0; G2 X20 R10 F100;

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

**Note** For the arc  $180^\circ$ , the positive or negative value of R doesn't affect the arc path.

- 3 The arc equal to  $360^\circ$  can only be programmed by I, J, K.

(Program example)

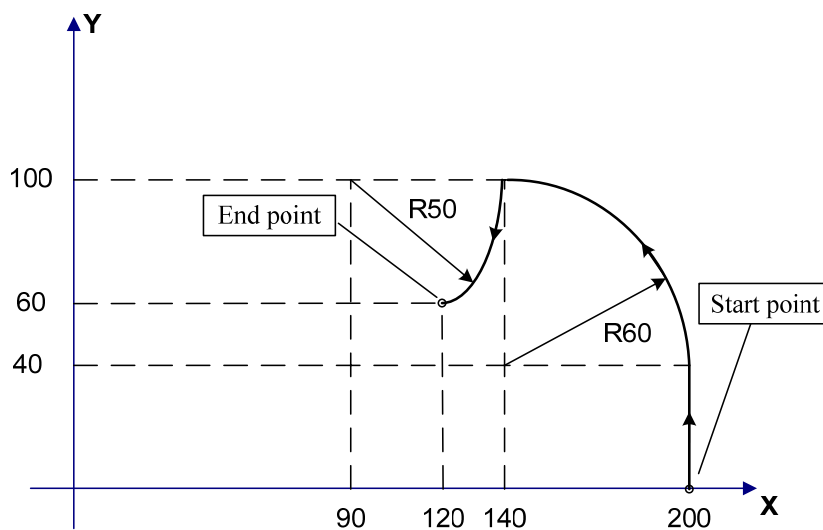


Fig. 4-2-3-4

The tool path programming for Fig. 4-2-3-4 is as following:

1. Absolute programming

G90 G0 X200 Y40 Z0;

G3 X140 Y100 R60 F300;

G2 X120 Y60 R50;

Or

G0 X200 Y40 Z0;

G90 G3 X140 Y100 I-60 F300;

G2 X120 Y60 I-50;

## 2. Incremental programming

```
G0 G90 X200 Y40 Z0;
```

```
G91 G3 X-60 Y60 R60 F3000;
```

```
G2 X-20 Y-40 R50;
```

Or

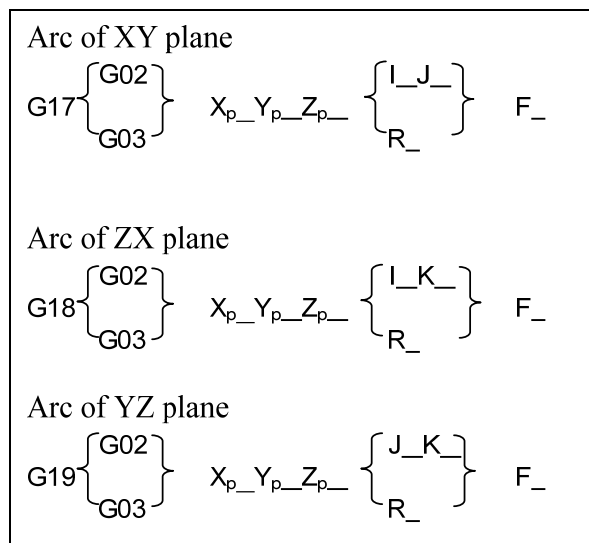
```
G0 G90 X200 Y40 Z0;
```

```
G91 G3 X-60 Y60 I-60 F300;
```

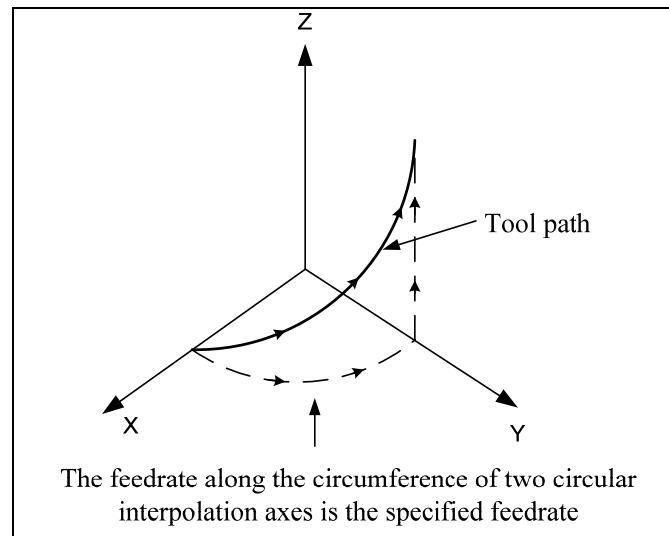
```
G2 X-20 Y-40 I-50;
```

**Restriction**

1. If address I, J, K and R are specified together in program, the arc specified by R is in priority and others are ignored.
2. If both arc radius parameter and the parameter from the start point to the circle center are not specified, error message will be issued by system.
3. If the circle is to be interpolated, only the parameters I, J, K from start point to circle center but the parameter R can be specified.
4. Attention should be paid to the coordinate plane selection when the circular interpolation is being done.
5. If X, Y, Z are all omitted, i.e. the start point and the final point coincides, as well as R is specified (e.g. G02R50), the tool doesn't move.

**B Helical interpolation****Format:** G02/G03**Fig. 4-2-3-5**

**Function:** It is used to move the tool to a position specified from current position by a feedrate specified by parameter F in a helical path.

**Explanation:****Fig. 4-2-3-6**

The first two bits of the instruction parameter are positioning parameter. The parameter words are the two axes name (X, Y or Z) in current plane. These two positioning parameters specify the position the tool is to go to. The third parameter word of the instruction parameter is a linear axis except the circular interpolation axis. Its value is the helical height. The significance and restriction for other instruction parameters are identical with circular interpolation.

If the circle can't be machined by the system specified instruction parameter, the system will give error message. And the system changes the current tool moving mode for G02/G03 mode.

Feedrate along the circumference of two circular interpolation axes are specified

The feedrate along the circumference of two circular interpolation axes is specified. The specification method is to simply add a moving axis which is not a circular interpolation axis. The feedrate along a circular arc is specified by F instruction. Thus the feedrate of the linear axis is as follows:

$$F * \frac{\text{Length of linear axis}}{\text{Length of circular arc}}$$

The feedrate should be ensured that the linear axis feedrate are not beyond any limit.

**Restriction:** Attention should be paid to the coordinate plane selection set when the helical interpolation is being done.

#### 4.2.4 Absolute/ incremental programming G90/G91

**Format:** G90/G91

**Function:** There are 2 instructions for axis moving, the absolute instruction and the incremental instruction. The absolute instruction is a method of programming by the axis moving end point coordinate, which is concerned with coordinate system. Refer to section 3.3.1~3.3.4.

The incremental instruction is a method of programming by the axis relative moving. The incremental value is irrelevant with the coordinate system concerned, it only uses moving direction and distance of the end point relative to the start point.

The absolute instruction and the incremental instruction are specified by G90 and G91 respectively.

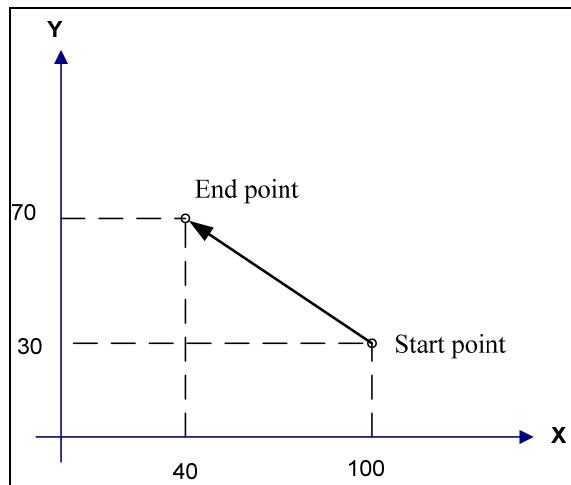


Fig. 4-2-4-1

For the moving from start point to end point in Fig. 4-2-4-1, the programming by absolute instruction G90 and incremental instruction G91 are as follows:

G90 G0 X40 Y70;

or G91 G0 X-60 Y40 ;

The action can be performed by both programming methods that can be expediently used by operator.

**Explanation:**

- No instruction parameter. It can be written into the block with other instructions.
- G90 and G91 are the same group mode, i.e. if G90 is specified while G91 not, the mode is G90(default). If G91 specified while G90 not, the mode is G91.

**System parameter**

G90 mode ( parameter is 1) or G91 ( parameter is 1) mode specified for the default positioning parameter at power-on can be set by bit parameter **NO: 31#4**.

## 4.2.5 Dwell (G04)

**Format:** G04 X\_ or P\_

**Function:** The dwell is executed by G04, and the execution of next block is delayed by the time specified. In addition, a dwell can be specified to make an exact stop check in cutting mode G64.

Table 4-2-5-1 Value range of dwell time (instructed with X)

Least moving unit	Value range	Unit of dwell time
No.5#1=0	0.001~9999.999	S or rev
No.5#1=1	0.0001~9999.9999	

Table 4-2-5-2 Value range of dwell time (instructed with P)

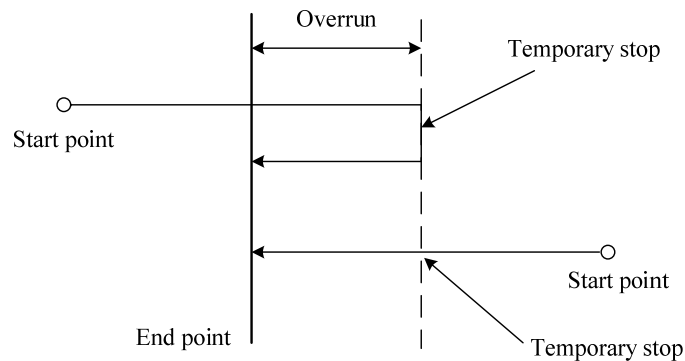
Least moving unit	Value range	Unit of dwell time
No.5#1=0	1~99999.999	0.001s or rev
No.5#1=1	1~99999.9999	0.0001s or rev

**Explanation:**

- 1 G04 is non-modal instruction, which is only effective in current line.
- 2 If parameter X, P both appear, parameter X is effective
- 3 Alarm occurs if X, P value is set for negative.
- 4 Exact stop is not executed if neither X nor P is specified.

**4.2.6 Unidirectional positioning (G60)**

**Format:** G60 X\_ Y\_ Z\_ F\_



**Fig. 4-2-6-1**

**Function:** For accurate positioning to eliminate machine backlash, G60 can be used for accurate positioning in a direction.

**Explanation:**

G60 is non-modal code (the modal value can be set by bit parameter NO: 48#0) , which is only effective in a specified block.

For parameter X, Y, Z, they represent the end point coordinate in absolute programming; and moving distance of tool in incremental programming. When using unidirectional positioning in tool offset, the path of unidirectional positioning is the tool compensation path.

The overrun marked in above figure can be set by system parameter P335, P336, P337, P338, P339, and the dwell time can be set by parameter P334. The positioning direction can be defined by the set positive or negative overrun, refer to system parameter for details.

Example 1:

G90 G00 X-10 Y10;

G60 X20 Y25; (1)

If the system parameter P334 = 1, P335 = -8, P336 = 5; as for statement (1), the tool path is AB→dwell for 1s→BC



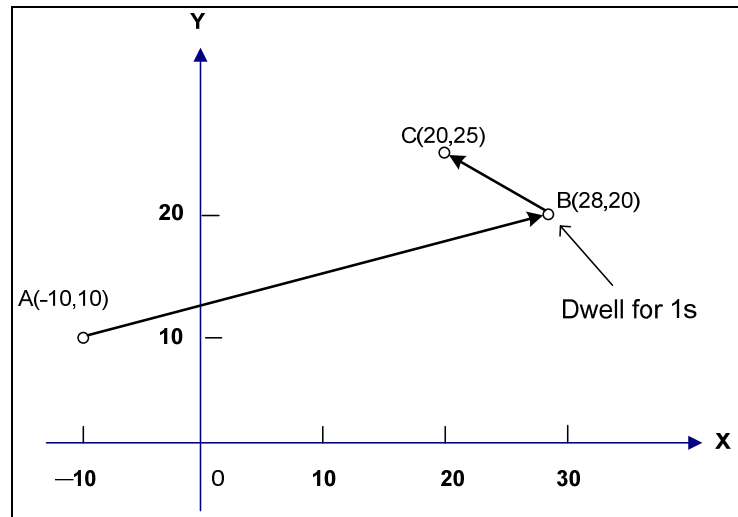


Fig. 4-2-6-2

System parameter:

Table 4-2-6-1

P334	Dwell time of unidirectional positioning (unit: mm)
P335	Overrun and unidirectional positioning direction in X axis (unit: mm)
P336	Overrun and unidirectional positioning direction in Y axis (unit: mm)
P337	Overrun and unidirectional positioning direction in Z axis (unit: mm)
P338	Overrun and unidirectional positioning direction in 4TH axis (unit: mm)

Note:

1. The sign of data parameter P335~P338 represents of the unidirectional positioning direction, and the value represents the overrun.
2. Overrun>0, positioning direction is positive.
3. Overrun<0, positioning direction is negative.
4. Overrun=0, do not perform unidirectional positioning.

#### 4.2.7 System parameter online modification (G10)

**Function:** It is used to set or modify the values of radius, length offset, external zero offset, workpiece zero offset, additional workpiece zero offset, number parameter, bit parameter and so on in program.

**Format:**

G10 L50 N\_P\_R\_;      Set or modify bit parameter  
 G10 L51 N\_R\_;      Set or modify number parameter  
 G11;      Parameter input mode cancel

**Parameter definition:**

N: Parameter number. Sequence number to be modified.  
 P: Parameter bit number. Bit number to be modified.  
 R: Value. Parameter value after it modified.

The values can also be modified by following instructions, refer to relative sections for details:

G10 L2 P\_X\_Y\_Z\_A\_B\_; Set or modify external zero offset or workpiece zero offset  
 G10 L10 P\_R\_; Set or modify length offset  
 G10 L11 P\_R\_; Set or modify length wear  
 G10 L12 P\_R\_; Set or modify radius offset  
 G10 L13 P\_R\_; Set or modify radius wear  
 G10 L20 P\_X\_Y\_Z\_A\_B\_; Set or modify additional workpiece zero offset

**Note:**

1. In parameter input mode, except annotation statement, other NC statement can't be specified.
2. G10 must be specified in a single block or the alarm occurs. It should be noted that the parameter input mode must be cancelled by G11 after G10 for program normal use.
3. The parameter value modified by G10 must be within the system parameter range. If not, alarm occurs.
4. The canned cycle mode must be cancelled prior to G10 execution, or alarm occurs.
5. Those parameters effective only by restarting after power-off can not be modified by G10.
6. On line modification for G20 and G21 is unavailable by G10.
7. When G10 modifies external zero offset, workpiece offset, additional workpiece zero offset or tool offset on line in G91 mode, the system adds the instruction offset to the current offset, when modifying them in G90 mode, it modifies by the instruction offset.
8. Cancel G10 mode when executing M00, M01, M02, M30, M99, M98 and M06.

#### 4.2.8 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. G54 (workpiece coordinate system 1) is selected automatically by system after machine zero return at power-on. The absolute position on display is the coordinate set in G54 coordinate system.
4. 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 (10, 10, 10) .

The corresponding machine coordinate for G55 coordinate system origin is (30, 30, 30) .  
When the program is executed by sequence, the absolute coordinate and the machine coordinate of the end point are shown as follows:

Table 4-2-8-1

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

5. The external workpiece zero offset or workpiece zero offset can be altered by G10, which is shown as following:

By instruction G10 L2 Pp X\_Y\_Z\_

P=0 : External workpiece zero offset

P=1 to 6 : Workpiece zero offset of workpiece coordinate system from 1 to 6

X\_Y\_Z\_ : For absolute instruction (G90) , it is workpiece zero offset of each axis  
For incremental instruction (G91) , it is workpiece zero offset set plusing each axis (the result is the new workpiece zero offset).

By G10 instruction, each coordinate system can be altered respectively.

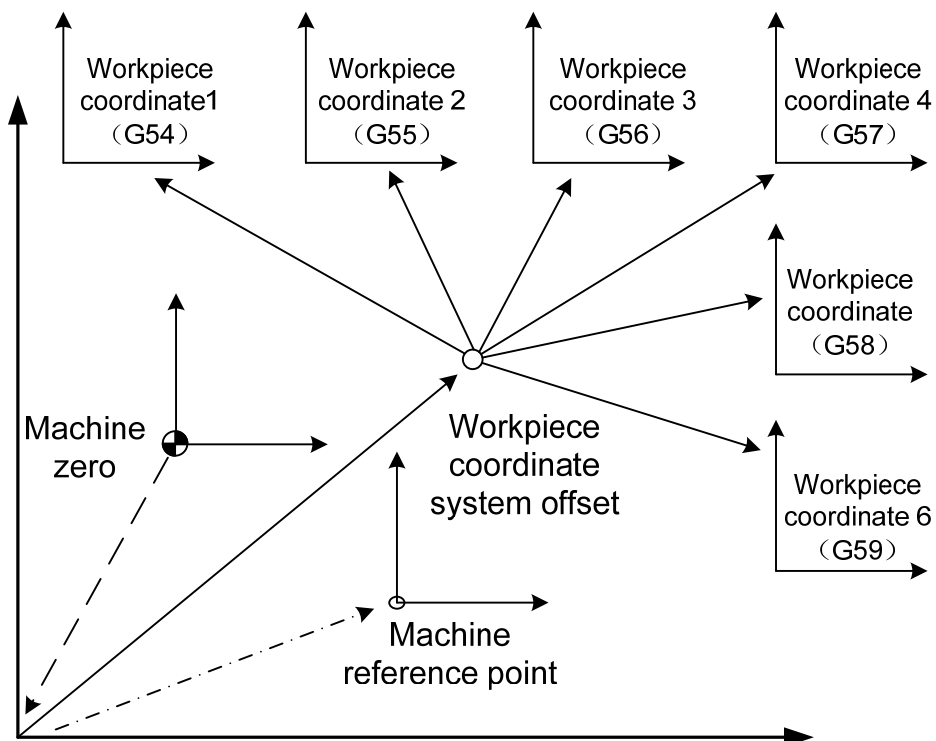


Fig.4-2-8-1

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 P10~14 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 P15~44. These 6 workpiece coordinate systems are set up by the distances from machine zero to each coordinate system origin.

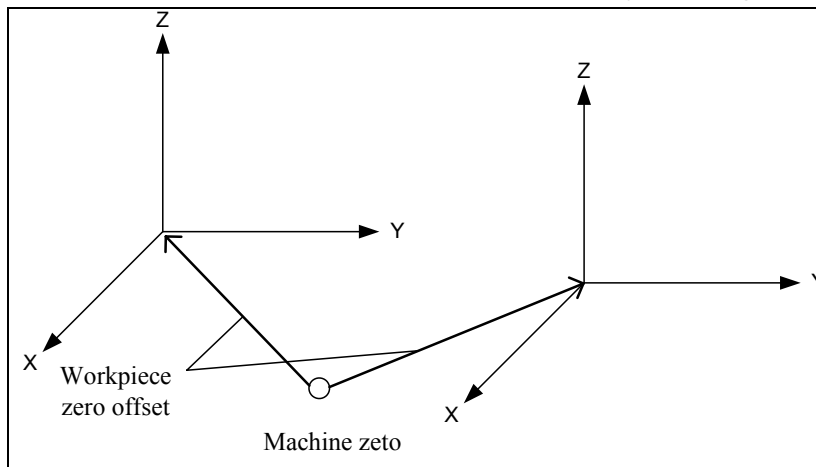


Fig. 4-2-8-2

**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.

### 4.2.9 Additional workpiece coordinate system

Except 6 workpiece coordinate systems (standard workpiece coordinate system) from G54 to G59, 50 additional workpiece coordinate systems can be used.

**Format:** G54 Pn  
Pn: specified additional workpiece coordinate system code  
Range : 1~50

The setting and restriction of the additional workpiece coordinate system are the same as that of workpiece coordinate system from G54 to G59.

The workpiece zero offset in additional workpiece coordinate system can be set by G10, as following:

By instruction G10 L20 Pn X\_Y\_Z\_

n=1 to 50: Additional workpiece coordinate system code

X\_Y\_Z\_ : Set axis address and offset value for workpiece zero offset.

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

For incremental instruction (G91), the new offset value can be got by adding the value specified to the current offset value.

By G10 instruction, each workpiece coordinate system can be changed respectively.

#### **4.2.10 Machine coordinate system selection G53**

**Format:** G53 X\_ Y\_ Z\_

**Function:** To rapidly position the tool to the corresponding coordinate location in the machine coordinate system.

**Explanation:**

- 1 While G53 is used in program, the instruction coordinate behind it should be the coordinate in the machine coordinate system and the machine will rapidly position to the location specified.
- 2 G53 is a non-modal instruction, which is effective in block containing it, and it doesn't affect the coordinate system defined before.

**Restriction**

Machine coordinate system selection G53

When the position in the machine coordinate system is specified, the tool rapidly traverses to this position. The G53 used for selecting machine coordinate system is a non-modal G code, which is only effective for the block specifying the machine coordinate system. Absolute G90 should be specified for G53; if G53 is specified in incremental mode (G91), G91 is neglected (G53 is still in G90 mode without changing G91 mode). The tool can be specified to move to a special position, e.g. G53 can be used in program to position the tool to the tool changing point.

**After power on**

Machine coordinate system must be set before G53 is specified after power on. Therefore, manual reference point return must be performed after power on (zero return in manual mode) or auto reference point return must be performed specified by G28. If an absolute position encoder is used, this operation is unneeded.

**Note: when G53 is specified, the tool radius compensation and tool length offset are cancelled temporarily and they will be restored in the next block.**

#### **4.2.11 Floating coordinate system G92**

**Format:** G92 X\_ Y\_ Z\_

**Function:** It is used to set floating workpiece coordinate system. The current tool absolute coordinate values in the new workpiece coordinate system are specified by 3 instruction parameters. And this instruction doesn't result in the axis movement.

**Explanation:**

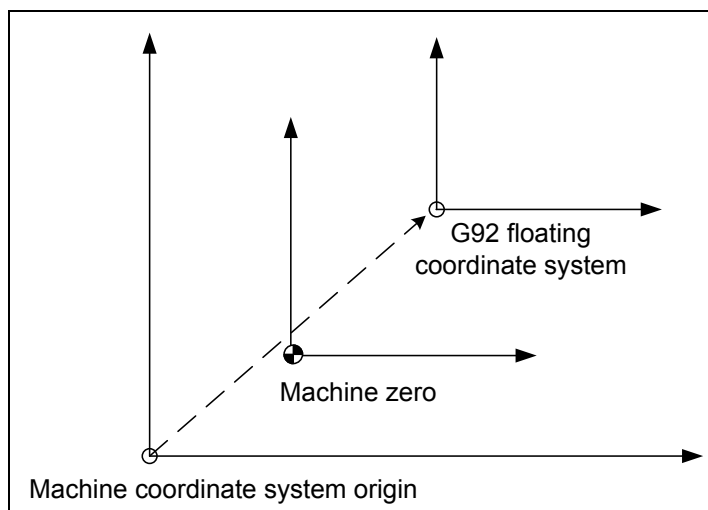


Fig. 4-2-11-1

- 1、As the figure shows, the origin of the G92 floating coordinate system is the value in machine coordinate system, which is irrelevant to the workpiece coordinate system, it can only be set up after the machine zero return.

G92 setting is effective in the following conditions:

- 1) Before system power off
- 2) Before workpiece coordinate system is called
- 3) Before machine zero return

The G92 floating coordinate system is usually used for the alignment of temporary workpiece machining and it will be lost after the power is off. And G92 is usually used at the program beginning or specified in MDI mode before the program auto run.

- 2、There are two methods for defining the floating coordinate system:

- (1) By tool nose:

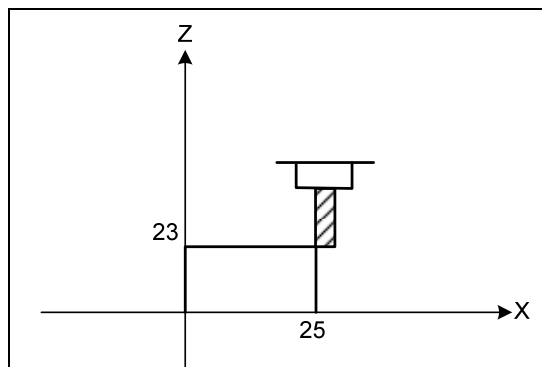


Fig. 4-2-11-2

As fig. 4-2-11-2 shows, for G92 X25.3 Z23, take the position the tool nose locates at as the point (X25.3, Z23) in the floating coordinate system,

- (2) By a fixed point in the arbor as a basic point:

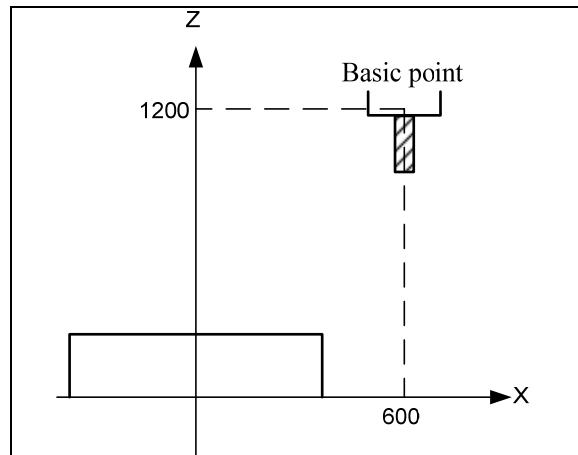


Fig. 4-2-11-3

As Fig. 4-2-11-3 shows, specify the workpiece coordinate system by block “G92 X600 Y1200”(by a basic point in the arbor as a start point). Regarding a basic point as the start point, if the motion is specified by the absolute value in the program, the basic point is moved to the specified position and it must be added the tool length compensation value, which is the difference of the basic point to the tool nose.

- Note:**
- 1 If G92 is used for coordinate system setting in tool offset, the coordinate system is the one set by G92 as to the tool length compensation without the offset value added.
  - 2 For tool radius compensation, the tool offset should be cancelled if G92 is used.
- Restriction**

#### 4.2.12 Plane selection G17/G18/G19

**Format:** G17/G18/G19

**Function:** For circular interpolation, tool radius compensation, drilling or boring, plane selection is needed, which can be selected by G 17/G18/G19.

**Explanation:**

It has no instruction parameter. The system default at power-on is G17 plane if parameter is not specified. It can also be set by bit parameter NO.31#1, #2, #3. The relation of the instruction and the plane is as following:

G17-----XY plane

G18-----ZX plane

G19-----YZ plane

Plane is not changed if G17, G18, G19 is not specified in the block.

For example:

G18 X\_ Z\_; ZX plane

G0 X\_ Y\_; Plane unchanged (ZX plane)

In addition the moving instruction is irrelevant to the plane selection. e.g. in the following instruction, Y axis is not in the ZX plane, so the Y axis moving is irrelevant to ZX plane.

G18Y\_;

**Annotation:** Only the canned cycle in G17 plane is available in this system at present. For

criterion or astringency, plane should be expressly defined in the corresponding block, especially in a system used by many users, which can avoid the incident or abnormality caused by programming error.

#### 4.2.13 Polar coordinate system setup/cancel G16/G15

**Format:** G16/G15

**Function:**

G16 is used for the setup of the polar coordinate system of the positioning parameter.

G15 is used for the cancellation of the polar coordinate system of the positioning parameter.

**Explanation:**

No command parameter.

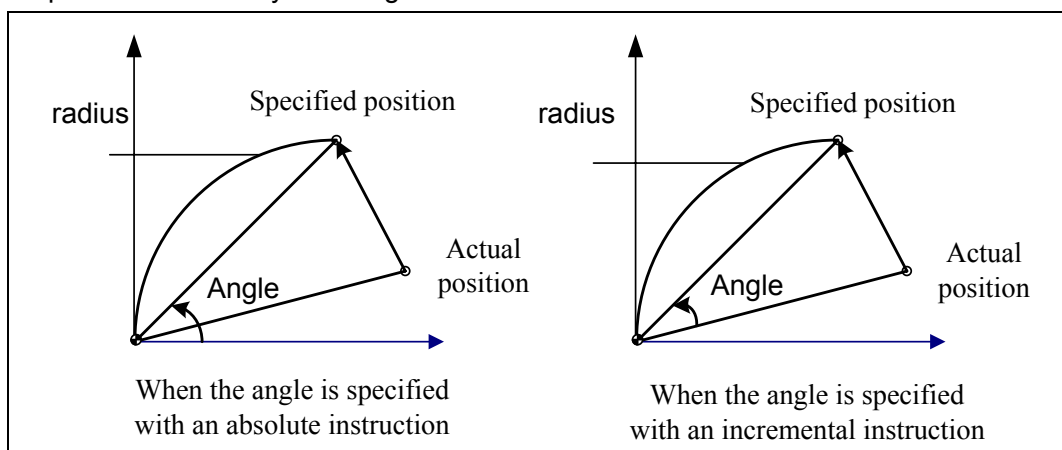
If G16 is set, the coordinate can be input by polar coordinate radius and angle. The positive of angle is the CCW direction of the 1<sup>st</sup> axis positive direction in a plane selected; while the negative is CW direction. Both the radius and angle can use the absolute or incremental instructions (G90 , G91) .

If G16 is used, the 1<sup>st</sup> axis of the positioning parameter of the tool moving command represents the polar radius in polar coordinate system, the 2<sup>nd</sup> axis of that represents the polar angle in polar coordinate system.

If G15 is specified, the polar coordinate system can be cancelled and the control returns to the Cartesian coordinate system.

**The definition of the polar coordinate system origin:**

- 1 In G90 absolute mode, if G16 is specified, the workpiece coordinate system origin is regarded as the polar coordinate system origin.



**Fig. 4-2-13-1**

- 2 In G91 incremental mode, if G16 is specified, the current point is regarded as the polar coordinate system origin.

**Example:** Bolt hole circle (the workpiece coordinate system zero point is set as the polar coordinate system origin, selecting X-Y plane)



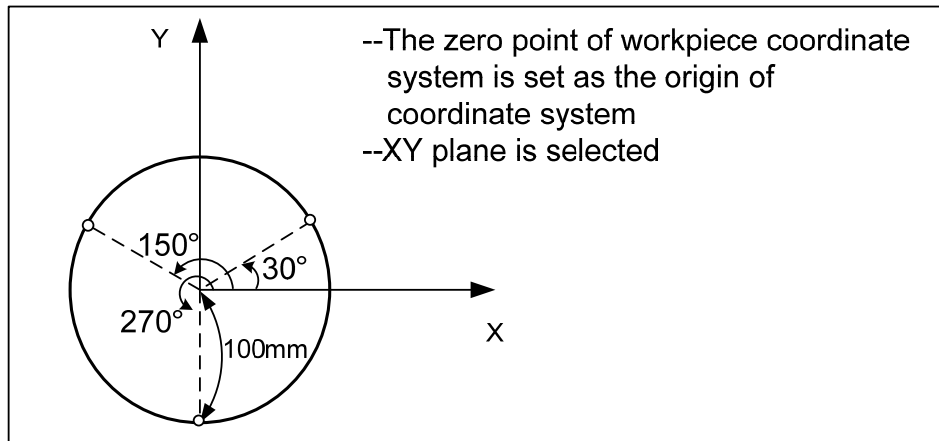


Fig. 4-2-13-2

- To specify angle and radius by absolute value  
 G17 G90 G16; To specify polar coordinate system and take the workpiece coordinate system zero point in X-Y plane as the polar coordinate system origin  
 G81 X100.0 Y30.0 Z-20.0 R-5.0 F200.0; To specify 100mm distance and 30°angle  
 Y150; To specify 100mm distance and 150°angle  
 Y270; To specify 100mm distance and 270°angle  
 G15 G80; To cancel the polar coordinate system
- To specify angle by incremental value, polar radius by absolute value  
 G17 G90 G16; To specify the polar coordinate system and take the workpiece coordinate system zero point in X-Y plane as the polar coordinate system origin  
 G81 X100.0 Y30.0 Z-20.0 R-5.0 F200.0; To specify 100mm distance and 30°angle  
 G91 Y120; To specify 100mm distance and 150°angle  
 Y120; To specify 100mm distance and 270°angle  
 G15 G80; To cancel the polar coordinate system

Moreover, when programming by polar coordinate system, the current coordinate plane setting should be considered. And the polar coordinate plane and the current coordinate plane are relevant. e.g. in G91 mode, if the current coordinate plane is specified by G17, the origin of it is defined by the X,Y axis components of the current tool position. If the current coordinate plane is specified by G18, the origin of it is defined by the Z, X axis components of the current tool position.

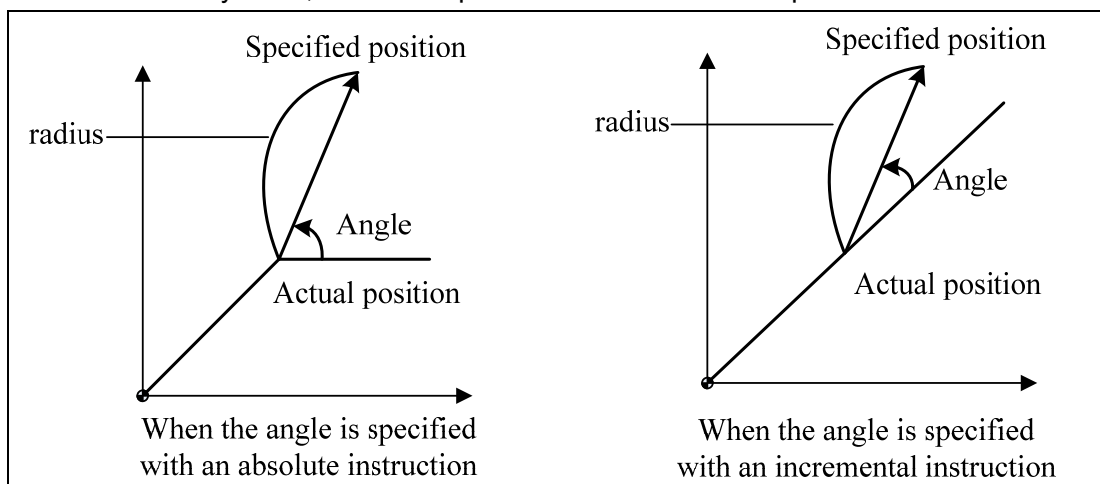


Fig. 4-2-13-3

If the positioning parameter of the 1<sup>st</sup> hole cycle after G16 instruction is not specified, the tool current position is the default positioning parameter of the hole cycle. The 1<sup>st</sup> canned cycle instruction after the current polar coordinate must be complete, or the tool moving will be wrong.

After G16 instruction, except the hole cycle, the words of the positioning parameter for tool moving involves with the special plane selection mode. While the polar coordinate system is cancelled by G15 which followed by a moving instruction, the tool current position is defaulted as the start point of the moving instruction.

#### 4.2.14 Scaling in plane G51/G50

##### Format:

G51 X\_ Y\_ Z\_ P\_ (X、Y、Z: Absolute instruction for scaling center coordinate, P: axis scaling by a same ratio)

... Scaling processing blocks

G50 Scaling cancel

or G51 X\_ Y\_ Z\_ I\_ J\_ K\_ (scaling by different ratios (I, J, K) by each axis)

... Scaling processing block

G50 Scaling cancel

##### Function:

G51 is used for the programming figure scaling in a same or different ratio by a position specified as the center. G51 is needed to be specified in a single block and cancelled by G50.

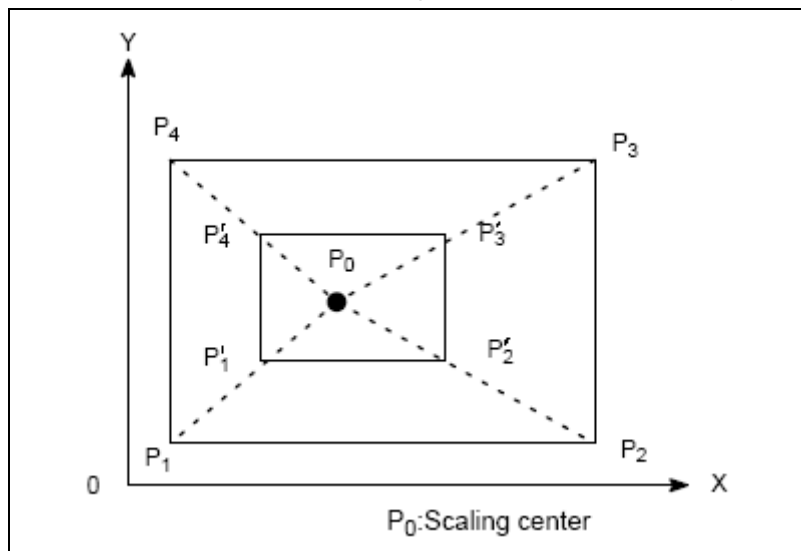


Fig. 4-2-14-1 Scaling (P1P2P3P4→ P'1'P'2'P'3'P'4 )

##### Explanation:

- 1 Scaling center: G51 can be specified with 3 positioning parameters X\_Y\_Z\_, which are optional. These positioning parameters are used to specify the scaling center of G51. If they are not specified, the tool current position will be specified as the scaling center. Whether the positioning mode is absolute or incremental, the scaling center is specified by the absolute positioning mode. Moreover, in polar coordinate system G16 mode, the parameters in G51 are expressed by Cartesian coordinate system.

**Example:**

G17 G91 G54 G0 X10 Y10;

G51 X40 Y40 P2;     Though in incremental mode, the scaling center means the absolute coordinate (40,40) in G54 coordinate system

G1 Y90;                     By incremental mode as for parameter Y

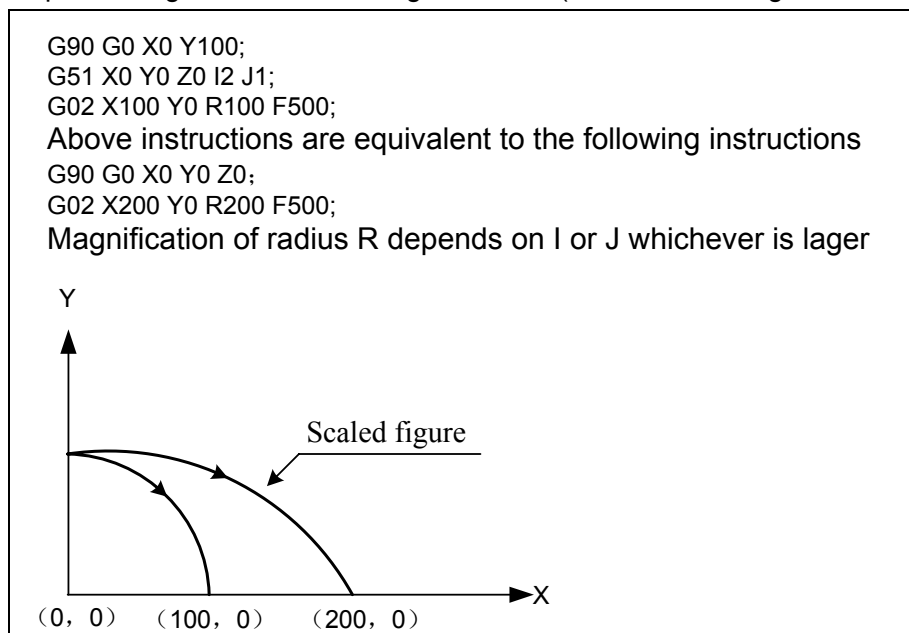
- 2    Scaling: whether the current mode is G90 or G91, the scaling are always expressed by absolute mode.

Except specified in program, the scaling can also be specified in parameters. The number parameters P331~335 correspond to the scaling ratios of X, Y, Z, respectively. If no scaling is specified, the number parameter P330 can be used for scaling setting.

If the parameter P or I, J, K value specified are negative, the mirror image is made for the corresponding axis.

- 3    Scaling setting: The effectiveness of the single axis scaling is set by bit parameter NO.47#3, the effectiveness of the axis scaling mirror image is set by bit parameter NO.47#6, and the ratio unit of it is set by bit parameter NO.47#7.
- 4    Scaling cancellation: After the scaling is cancelled by G50 followed by a moving instruction, if the coordinate rotation is cancelled by default, the current tool position is regarded as the start point of this moving instruction.
- 5    In scaling mode, G codes for reference point return (G27~G30 etc.) and coordinate system specification (G52~G59, G92 etc.) can't be specified. If needed, they should be specified after the scaling is cancelled.
- 6    Even different scalings are specified for circular interpolation and axes, the ellipse path can't be made by tool.

If the scaling ratios of the axes are different and the circular interpolation are programmed by R, the interpolation figure is shown as Fig. 4-2-14-2, (below the scaling ratio of X is 2, that of Y is 1)

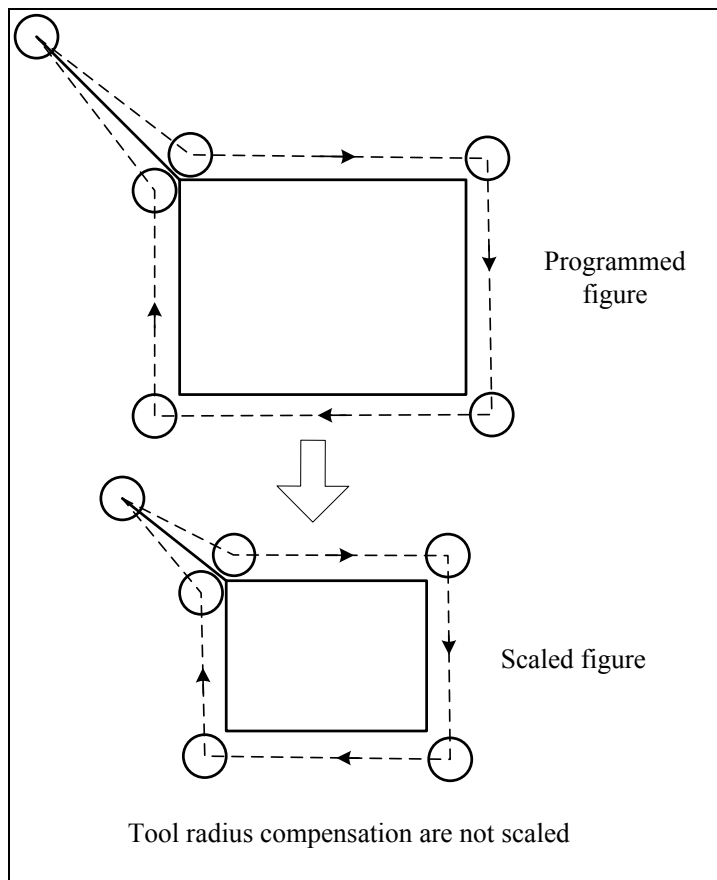


**Fig. 4-2-14-2     Scaling of circular interpolation 1**

The axes scaling ratio are different, if the arc is failed when the circular interpolation is

programmed by I, J, K, alarm for it occurs by the system.

- 7 Scaling is ineffective for the tool radius compensation, tool length compensation and tool offset, which is shown in Fig. 4-2-14-3.



**Fig. 4-2-14-3 Scaling of tool radius interpolation**

Example for mirror image program:

Main program

G00 G90;

M98 P9000;

G51 X50.0 Y50.0 I1 J-1;

M98 P9000;

G51 X50.0 Y50.0 I-1 J-1;

M98 P9000;

G51 X50.0 Y50.0 I-1 J1;

M98 P9000;

G50;

M30;

Subprogram

O9000

G00 G90 X60.0 Y60.0;

G01 X100.0 F100;

```
G01 Y100;
G01 X60.0 Y60.0;
M99;
```

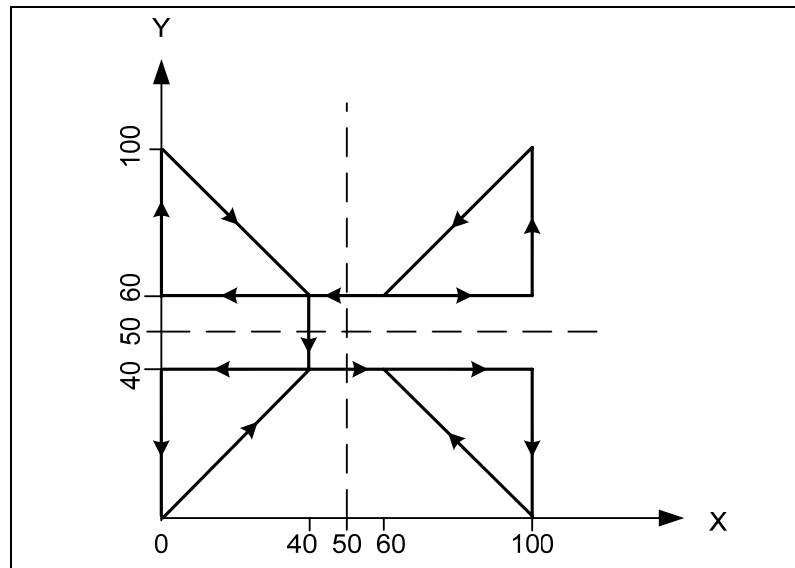


Fig. 4-2-14-4

**Restriction**

- 1 The moving scaling of Z axis is ineffective in following canned cycles:
  - 1) The cut-in value Q and retraction value d of peck drilling cycle (G83, G73)
  - 2) Fine boring cycle (G76) .
  - 3) Offset value Q of X axis and Y axis in back boring cycle (G87) .
- 2 In JOG mode, the traverse distance can't be increased or decreased by scaling.

**Note:**

- 1 **The position is displayed by scaling coordinates.**
- 2 **The result for an axis performing mirror image in a specified plane is as follows:**

- 1) Circular instruction..... reverse direction of rotation
- 2) Tool radius compensation C..... reverse direction of offset
- 3) Coordinate system rotation.....reverse direction of rotation angle

**4.2.15 Coordinate system rotation G68/G69**

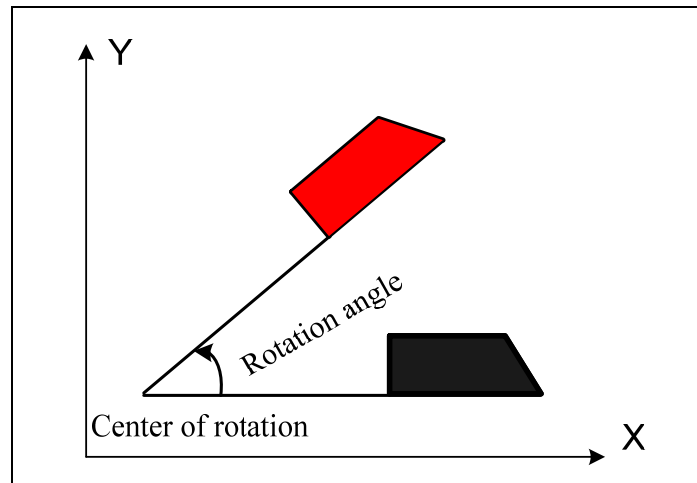
When a workpiece consist of some identical shapes, this function can be used for programming by preparing a subprogram for the shape unit, then calling it by rotation function.

**Format:**

```
G17 G68 X_ Y_ R_
or G18 G68 X_ Z_ R_
or G19 G68 Y_ Z_ R_
G69
```

**Function:** G68 is used for the programming shape in plane rotating by a center point specified as

an origin. G69 is used for cancellation of coordinate system rotation.



**Fig. 4-2-15-1**

Function: G68 makes the figure programmed in the plane take the specified center as the origin to rotate. G69 is used for the coordinate system rotation cancellation

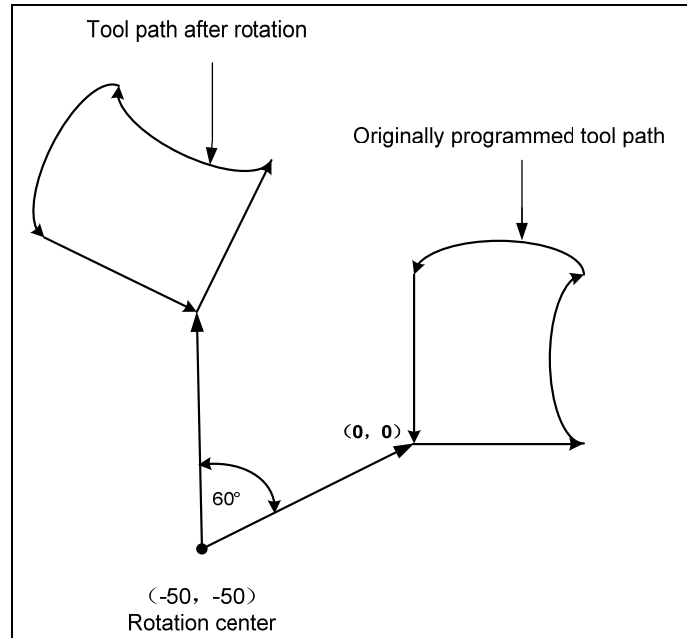
**Explanation:**

- 1 G68 is an optional parameter with 2 positioning parameters that are used for specifying the rotation center. If the rotation center is not specified, the tool current position is regarded as the center by system. The positioning parameters are relevant to the current coordinate plane, while X, Y for G17; Z, X for G18; Y, Z for G19.
- 2 Whether the current positioning mode is absolute or incremental, the rotation center can only be specified by absolute positioning of Cartesian coordinate system.  
G68 can be followed by a command parameter R, the value of the parameter is the angle to be rotated. The positive value is for CCW rotation and the angle unit is degree. If no rotation angle is specified in this function, the angle will be set by number parameter P329.
- 3 In G91 mode, the rotation angle by increment is set by bit parameter NO: 47#0 (rotation angle of coordinate system, 0: by absolute instruction; 1: by G90/91 instruction ).
- 4 When the system is in rotation mode, plane selection is not allowed, or errors will be shown. Attention should be paid in programming.
- 5 In coordinate system rotation mode, G codes for reference point return (G27~G30 etc.) and coordinate system specification (G52~G59, G92 etc.) can't be specified. They should be specified after the scaling is cancelled if needed.
- 6 After coordinate system rotation, the tool radius compensation, tool length compensation, tool offset and other compensation operation will be performed.
- 7 If coordinate system rotation is performed in scaling mode(G51), the rotation center coordinate values will be scaled. but the rotation angle is not scaled, when the moving instruction is given, the scaling will be executed first, then the coordinate system rotation.

**Example 1: Rotation**

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

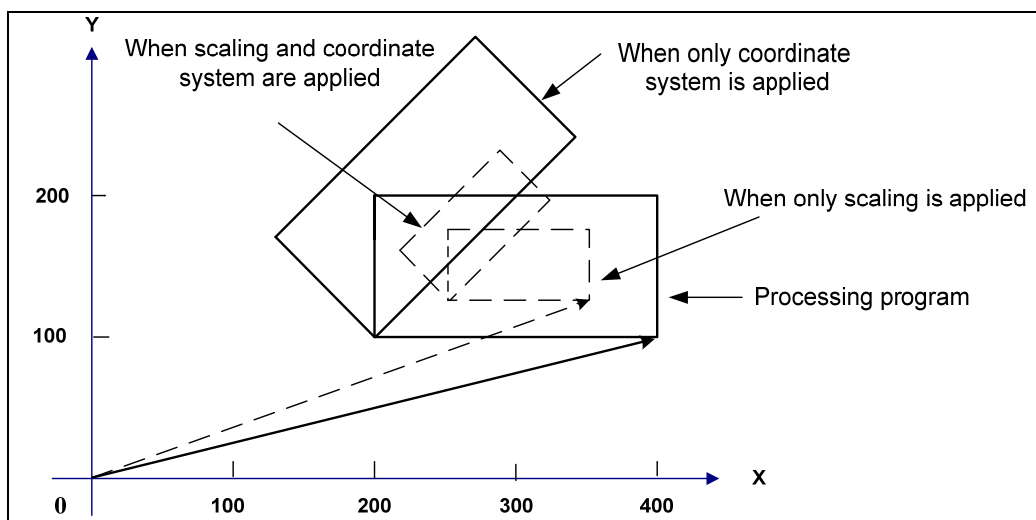
G3 X-100 I-50 J-50;  
G01 Y-100;  
G69;  
M30;



**Fig. 4-2-15-2**

**Example 2: Scaling and rotation**

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



**Fig. 4-2-15-3**

## Example 3 : Repetition of G68

By program (main program)

```

G92 X0 Y0 Z20 G69 G17;
M3 S1000;
G0Z2 ;
G51 X0 Y0 I1.2 J1.2;
G42 D01;                (offset setting)
M98 P2100 (P02100);      (subprogram call)
M98 P2200L7;             (calling for 7 times)
G40;
G50;
G0 G90 Z20;
X0Y0;
M30;

```

Subprogram 2200

```

O2200 G68 X0 Y0 G91 R45.0;  (relative rotation angle)
G90;
M98 P2100;                  (subprogram O2200 calling subprogram O2100)
M99;

```

Subprogram O2100

```

O2100 G90 G0 X0 Y-20;       (Right-hand tool compensation setup)
G01Z-2 F200;
X8.284;
X14.142 Y-14.142;
M99;

```

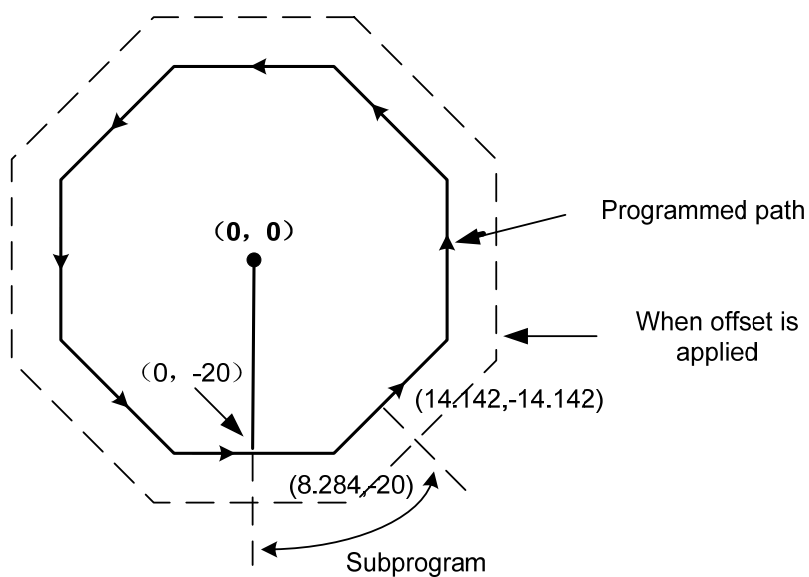


Fig. 4-2-15-4



#### 4.2.16 Skip function G31

**Format:** G31 X\_Y\_Z\_

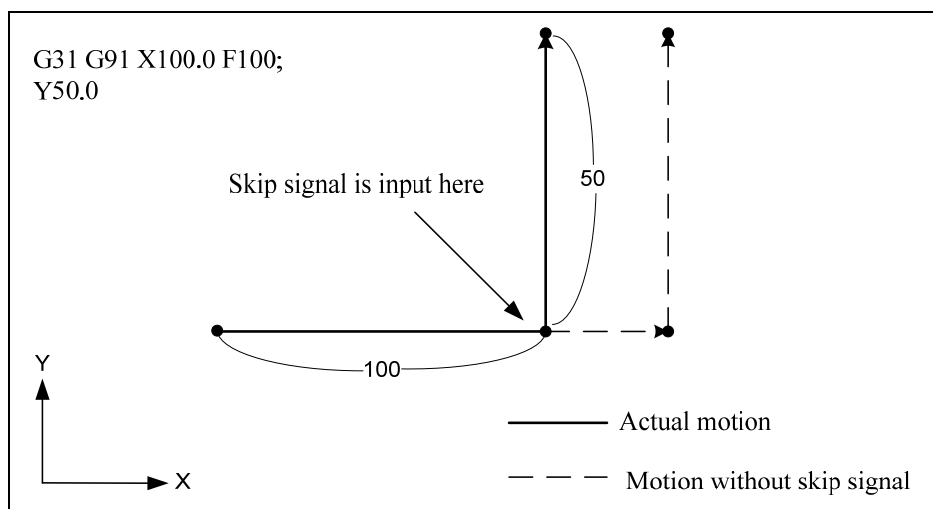
**Function:** The linear interpolation can be specified like G01 after G31 instruction. During the execution of G31, the current instruction execution will be interrupted to execute next block if an external skip signal is entered. While the working end point is specified not by programming but by signals from machine, this function can be used (e.g. used for grinding). It can also be used for measuring the workpiece dimensions.

**Explanation:**

- 1、G31 is a non-modal G code that is only effective in a specified block.
- 2、Alarm occurs if G31 is given during the tool radius compensation. The tool radius compensation should be cancelled before G31 instruction.

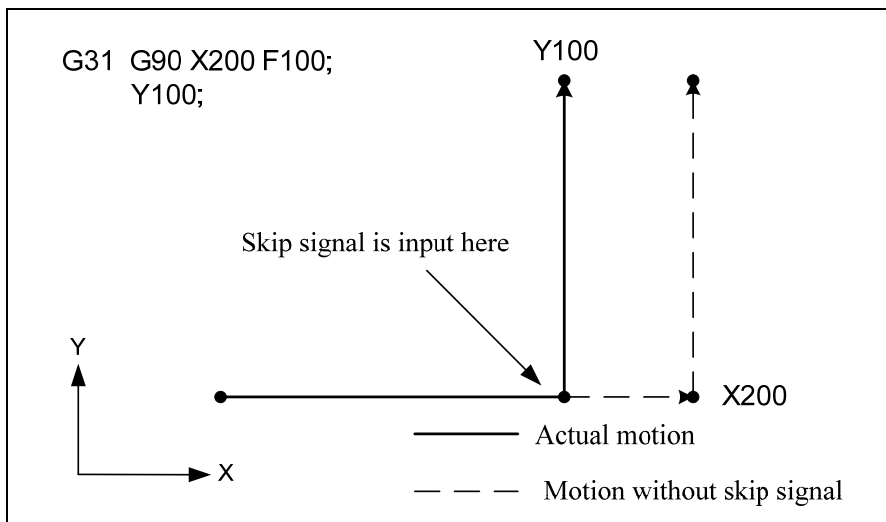
**Example:**

The block after G31 is a single axis moving specified by incremental values, as Fig. 4-2-16-1 shows:



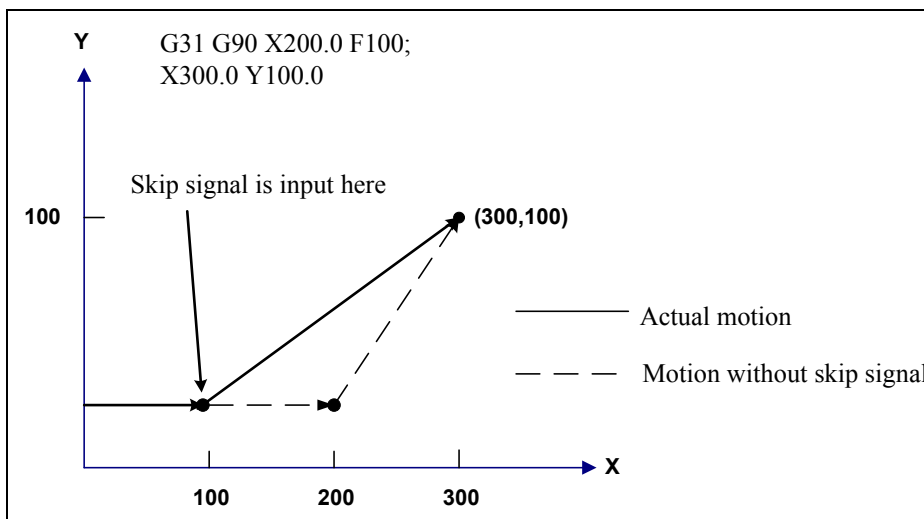
**Fig. 4-2-16-1 A single axis moving specified by incremental values of next block**

The block after G31 is a single axis moving specified by absolute values, as Fig. 4-2-16-2 shows:



**Fig. 4-2-16-2 Single axis moving specified by absolute values of next block**

The block after G31 is 2-axis moving specified by absolute values, as Fig. 4-2-16-3 shows:



**Fig. 4-2-16-3 2-axis moving specified by absolute values of next block**

**Note:** The setting can be done by bit parameter NO: 02#6 [skip signal SKIP, (0:1, 1:0)].

#### 4.2.17 Inch/metric conversion G20/G21

**Format:** G20: input by inch system

G21: input by metric system

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

**Explanation:**

Change the unit of the following item after the inch/metric conversion:

Feedrate specified by F code

Position instruction

Workpiece zero offset value

Tool compensation value

Scale unit of MPG

Moving distance in incremental feeding

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

- Note:**
- 1 Inch/metric conversion can't be executed during the program execution.
  - 2 The tool compensation value must be preset by the minimum incremental input unit when inch system is converted to metric system or the reverse.
  - 3 For the 1<sup>st</sup> G28 instruction, the running from the intermediate point is the same as the JOG reference point return when inch system is converted to metric system or the reverse.
  - 4 When the minimum incremental input unit is different from the minimum command unit, the maximum error that is not accumulated is the half of the minimum command unit.
  - 5 The inch/metric system for program input can be set by bit parameter NO: 00#2.
  - 6 The inch/metric system for program output can be set by bit parameter NO:03#0.

#### 4.2.18 Arbitrary angle chamfering/corner rounding

**Format:** L\_: chamfering  
R\_: corner rounding

**Function:** When the above instruction is added to the end of a block that specifies linear interpolation (G01) or circular interpolation (G02, G03), a chamfering or corner rounding is automatically done in the machining. Blocks specifying chamfering and corner rounding can be specified consecutively.

**Explanation:**

- 1、Blocks specifying chamfering and corner rounding can only be inserted between the linear interpolation blocks.
- 2、The chamfering after L is used to specify the distance from the virtual corner point to the start and the end point. The virtual corner point is the corner point that exists if chamfering is not performed. As the following figure shows:

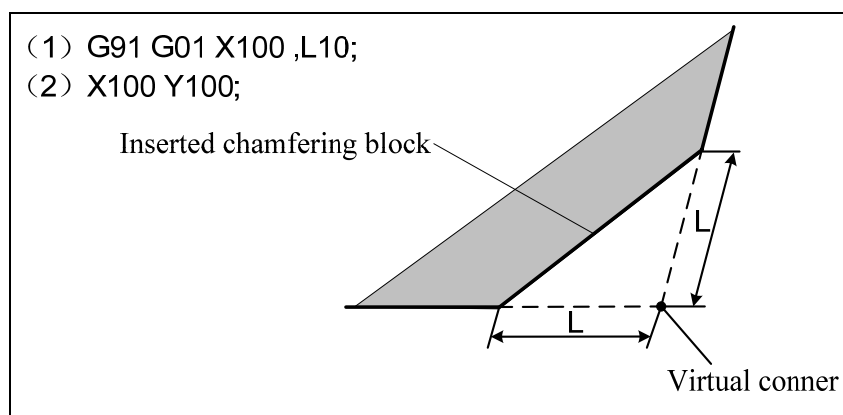


Fig. 4-2-18-1

- 3、The corner rounding after R is used to specify the radius for corner. As the following figure shows:

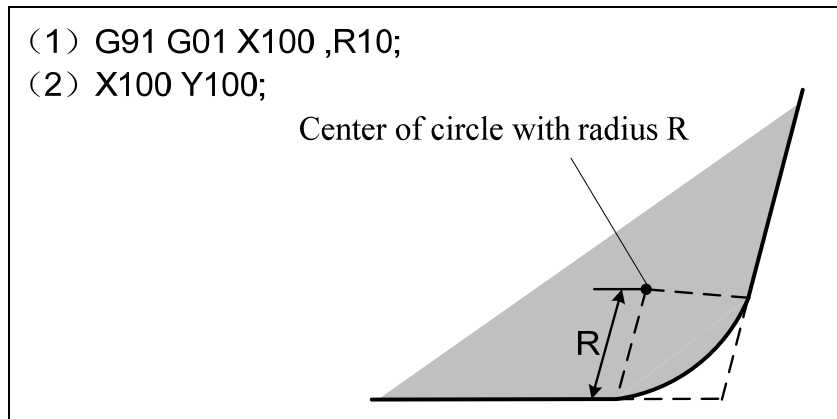


Fig. 4-2-18-2

**Restriction**

- 1 Chamfering and corner rounding can only be performed in the specified plane, and these functions can't be performed for parallel axes.
- 2 If the inserted chamfering or corner rounding block causes the tool to go beyond the original interpolation move range, alarm is issued.
- 3 Corner rounding can't be specified in a threading block.
- 4 The chamfering and corner rounding value can't be negative, or alarm is issued.

**4.3 Reference point G code**

The reference point is a fixed point on a machine tool to which the tool can easily be moved by the reference point return function. There are 3 instructions for reference point as is shown in Fig. 4.3.1.1, the tool can be automatically moved to the reference point via an intermediate point along an axis specified by G28; or from the reference point automatically to a specified point via an intermediate point along a specified axis by G29.

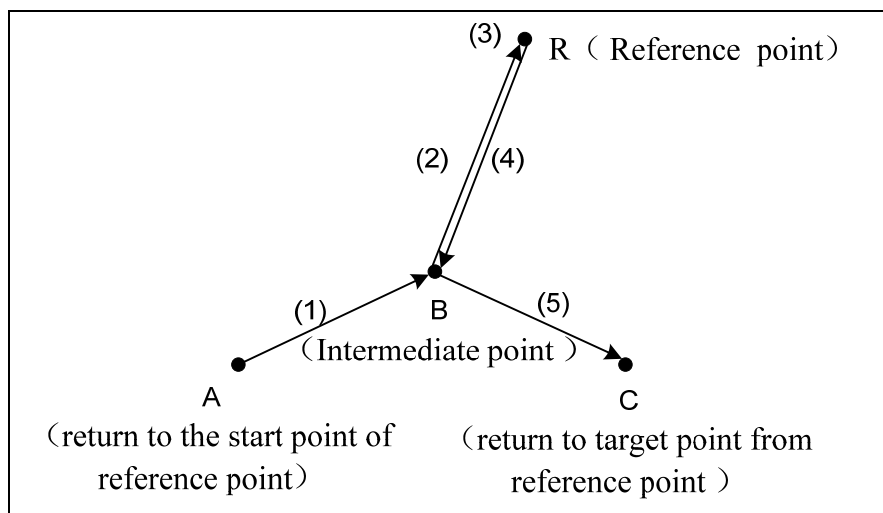


Fig. 4-3-1

**4.3.1 Reference point return G28**

**Format:** G28 X\_ Y\_ Z\_

**Function:** It is used for the operation to return to the reference point (a special point on machine)

via an intermediate point.

#### Explanation:

##### Intermediate point:

An intermediate point is specified by an instruction parameter in G28, which can be expressed by absolute or incremental instructions. During the execution of this block, the coordinate value of the intermediate point of the axis specified is stored that is to be used for the G29(returning from the reference point) instruction.

**Note:** The coordinate value of the intermediate point is stored in the CNC system. Only the axis coordinate value specified by G28 is stored each time, for the other axes not specified by G28, the coordinate values specified by G28 before are used. If the intermediate point defaulted by the system is not ensured by user when using G28 instruction, it is better to specify all the axes. Take a consideration by N5 block in the following example 1.

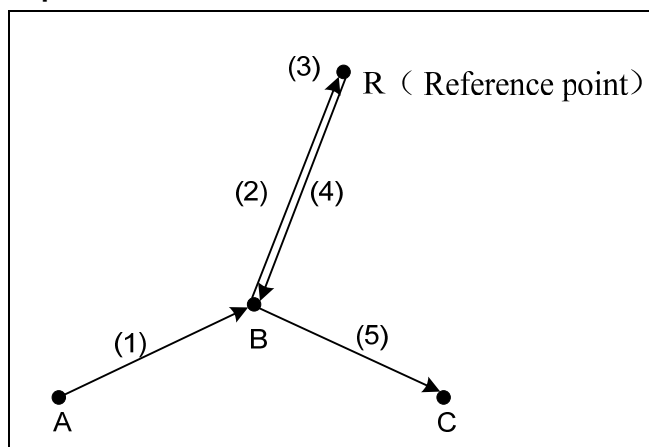


Fig. 4-3-1-1

- 1 The action of the G28 block can be analyzed as following: (refer to Fig.4-3-1-1):
  - (1) Positioning to the intermediate point of the specified axis from the current position (point A→point B) at a traverse speed.
  - (2) Positioning to the reference point from the intermediate point (point B →point R) at a traverse speed.
- 2 G28 is a non-modal instruction which is only effective in current block.
- 3 The combined reference point return of a single axis or multiple axes is available in this system. And the intermediate point coordinate is saved by system during the workpiece coordinate system change.

#### Example 1:

N1 G90 G54 X0 Y10;

N2 G28 X40 ; Set the intermediate point of X axis for X40 in G54 workpiece coordinate system, and return to reference point via point (40,10) , i.e. reference point return of single X axis

N3 G29 X30 ; Return to the point (30, 10) via point (40,10) from reference point, i.e. target point return of single X axis

N4 G01 X20;

N5 G28 Y60 ; Intermediate point(X40, Y60), which is substituted by X40 specified by G28 before due to it is not specified in X axis.

**Note:** The intermediate point is not (20, 60) .

N6 G55; Due to workpiece coordinate system change, the intermediate point (40, 60) in G54 workpiece coordinate system is changed for (40, 60) in G55 workpiece coordinate system.

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

The G28 instruction can automatically cancel the tool compensation and this instruction is only used in automatic tool change mode (changing tool at the reference point after reference point return). So the tool radius compensation and tool length compensation should be cancelled before using this instruction. See the 1<sup>st</sup> reference point setting in number parameter P45~P49.

### 4.3.2 2<sup>nd</sup>, 3<sup>rd</sup>, 4<sup>th</sup> reference point return G30

There are 4 reference points in machine coordinate system. In a system without an absolute-position detector, the 2<sup>nd</sup>, 3<sup>rd</sup>, 4<sup>th</sup> reference point return functions can be used only after the auto reference point return (G28) or manual reference point return is performed.

**Format:** G30 P2 X\_ Y\_ Z\_; the 2<sup>nd</sup> reference point return (P2 can be omitted)

G30 P3 X\_ Y\_ Z\_; the 3<sup>rd</sup> reference point return

G30 P4 X\_ Y\_ Z\_; the 4<sup>th</sup> reference point return

**Function:** It is used for the operation of returning to the specified point via the intermediate point specified by G30 from the reference point.

**Explanation:**

- 1 X\_ Y\_ Z\_; Instruction for specifying the intermediate point (absolute/ incremental)
- 2 The specification and restriction for G30 instruction is the same as G28 instruction. See number parameter P50~64 for the 2<sup>nd</sup>, 3<sup>rd</sup>, 4<sup>th</sup> reference point setting.
- 3 The G30 code can also be used together with G29 code (return from reference point), whose setting and restriction are identical with G28 code.

### 4.3.3 Automatic return from reference point G29

**Format:** G29 X\_ Y\_ Z\_

**Function:** It is used for the operation of returning to a specified point via the intermediate point specified by G28, G30 from the reference point (or current point).

**Explanation:**

- 1 The action of the G29 block can be analyzed as following: (refer to Fig.4-3-1-1):
  - (1) Positioning to the intermediate point (point R→point B) specified by G28, G30 from the reference point at a traverse speed.
  - (2) Positioning to a specified point from the intermediate point (point B →point C) at a traverse speed.
- 2 G29 is a non-modal instruction which is only effective in current block. Usually return from reference point should be specified immediately after G28, G30 instruction.
- 3 The optional parameters X,Y and Z in G29 instruction are used for specifying the target point (i.e. point C in Fig. 4-3-1-1) from the reference point, which can be expressed by absolute or

incremental instruction. The instruction specifies the incremental value from the intermediate point in incremental programming. If an axis is not specified it means the axis has no moving relative to the intermediate point. The G29 instruction followed by an axis is a single axis return with no action taken by other axes.

**Example 1**

```
G90 G0 X10 Y10;  
G91 G28 X20 Y20;      Reference point return via the intermediate point(30, 30)  
G29 X30;              Return to (60, 30) from the reference point via the intermediate  
                      point(30, 30).
```

**Note: The component in X axis should be 60 in incremental programming.**

The intermediate point of G29 instruction is assigned by G28, G30. Refer to G28 explanation for the definition, criterion and system default of the intermediate point.

#### 4.3.4 Reference point return check G27

**Format:** G27 X\_ Y\_ Z\_

**Function:** It is used for the reference point return check; the reference point is specified by X\_ Y\_ Z\_ (absolute/incremental instruction).

**Explanation:**

1. G27 instruction positions the tool at a traverse speed. If the tool reaches the reference point, the reference point return indicator lights up. However, if the position reached by the tool is not the reference point, an alarm is issued.
2. In machine lock mode, even G27 is specified and the tool has automatically returned to the reference point, the indicator for return completion doesn't light up.
3. In an offset mode, the position to be reached by the tool with G27 instruction is the position obtained by adding the offset. Therefore, if the position with the offset added is not the reference point, the indicator does not light up, and an alarm is issued. Usually the tool offset should be cancelled before G27 instruction.
4. The coordinate position of X, Y and Z specified by G27 is the position in the machine coordinate system.

#### 4.4 Canned cycle G code

Canned cycle make it easier for the programmer to creat programs. With a canned cycle, a machining operation by multiple blocks can be realized by a single block which contains G function. (In this system only canned cycle in G17 plane is available)

**The general process of canned cycle:**

A canned cycle consists of a sequence of 6 operations, as Fig. 4-4-1 shows:

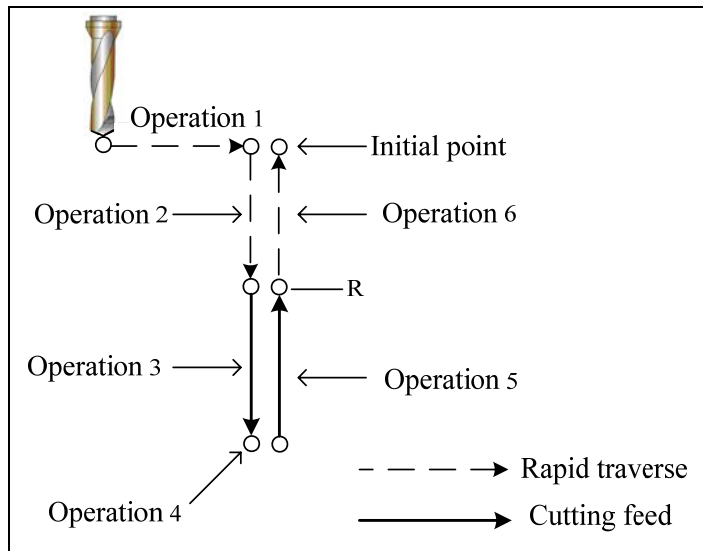


Fig. 4-4-1

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

The hole machining can be performed in Z axis if positioned in XY plane. It defines that a canned cycle operation is determined by 3 types. They are all specified by G code.

- 1) Data type
  - G90 absolute mode; G91 incremental mode
- 2) Return point plane
  - G98 initial level; G99 R level
- 3) Groove machining type
  - G22、G23、G24、G25、G26、G32、G33、G34、G35、G36、G37、G38。
- 4) Hole machining type
  - G73, G74, G76, G81~G89

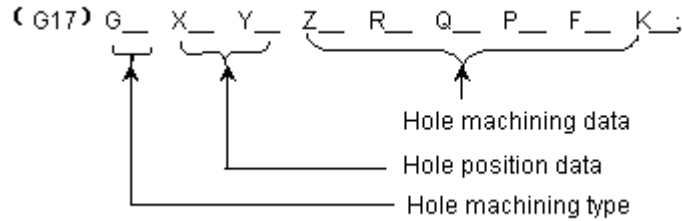
#### Initial level and R level

**Initial level** It is the absolute position where the tool locates in Z axis before the canned cycle.

**R level** It is also called safe plane, it is a position in Z axis when the traverse is switched to the feeding in canned cycle, which is usually positioned at a distance from the workpiece surface to prevent the tool from colliding with the workpiece and provide a sufficient distance to finish the acceleration. The instructions of G73/G74 /G76/G81~G89 specify all the data( hole location data, hole machining data, repetition) , by which a block is constituted.

The format for hole machining is shown as following:





Therein, the significance of the hole location data and machining data is as following Table 4-4-1:

Table 4-4-1

Designation	Parameter word	Explanation
Hole machining	G	Refer to Table 4-4-3, note the restrictions above.
Data for hole location	X, Y	The hole location is specified by absolute value or incremental value and the control is identical to the G00 positioning.
Data for hole machining	Z	As Fig. 4.4.2(A) shows, the distance from point R level to the hole bottom is specified by incremental value, or the hole bottom coordinate is specified by absolute value. And the feedrate is the speed specified by F in operation 3; while in operation 5, it is a traverse speed or a speed specified by F code due to the different machining type.
	R	In Fig. 4.4.2(B), the distance from the initial level to point R level is specified by incremental value or point R level coordinate is specified by absolute value. The speeds in operation 2 and 6 are both traverse.
	Q	It is used to specify the cut-in value or the parallel moving value in G76 or G87.
	P	It is used to specify the dwell time at the hole bottom. The canned cycle instruction can be followed by a parameter P_, which specifies the dwell time after the tool reaches the Z plane. The time unit is ms. The min. value of the parameter can be set by number parameter P281, and the max. value by number parameter P282.
	F	It is used to specify the cutting feedrate.
	K	The repetition is specified in parameter K_, which is effective only in the specified block. It can be omitted and the default is one time. The max. drilling times are 99999. If a negative value is specified, it executes by absolute values. If zero is specified, the mode is changed without drilling operation.

### Restriction

- The canned cycle is modal instruction, which is effective till it is cancelled by a G code.
- G80 and G codes in 01 group are used for cancelling canned cycle.
- The processing data once specified in canned cycle are effective till the canned cycle is cancelled. Therefore, after all the processing data required for hole machining are specified in the beginning of the canned cycle, only the data to be changed is needed to be respecified in the following canned cycle.

### Note

- 1 The feedrate specified by F remains effective even the canned cycle is cancelled.
- 2 Note: scaling for Z axis (cutting axial direction) is ineffective in canned cycle.
- 3 In single mode, the canned cycle has 3 stage working type, positioning→R level→initial level.
- 4 In canned cycle, the data of hole machining and hole position will be eliminated if the system is reset. The instance of data retained and eliminated is shown as following table:

Table 4-4-2

No.	Designation of data	Explanation
①	G00X_M3;	
②	G81X_Y_Z_R_F_;	Specify values for Z, R, F in the beginning.
③	Y_;	G81, Z-R-F- can be omitted due to the identical hole machining mode and data specified in ②. Drill the hole for the length Y once by G81.
④	G82X_P_;	Move in X axis relative to hole ③. Do the hole machining by G82 and data Z, R, F specified in ② and P in ④.
⑤	G80X_Y_	Hole machining is not performed. Cancel all the hole data.
⑥	G85X_Z_R_P_;	Because all data are cancelled in ⑤, Z, R needs to be respecified and F that remains can be omitted. P is saved but not needed in this block.
⑦	X_Z_;	It is a hole machining with a different Z value to ⑥. And there is moving only in X axis.
⑧	G89X_Y_;	Do the hole machining by G89 according to the data Z specified in ⑦, R, P in ⑥ and F in ②.
⑨	G01X_Y_;	Cancel the hole machining mode and data.

#### A Absolute instruction and incremental instruction in canned cycle G90/G91

The change of G90/G91 along drilling axis is shown as Fig. 4-4-2. (Usually it is programmed by G90, if it is programmed by G91, Z and R are regarded as negative values.)

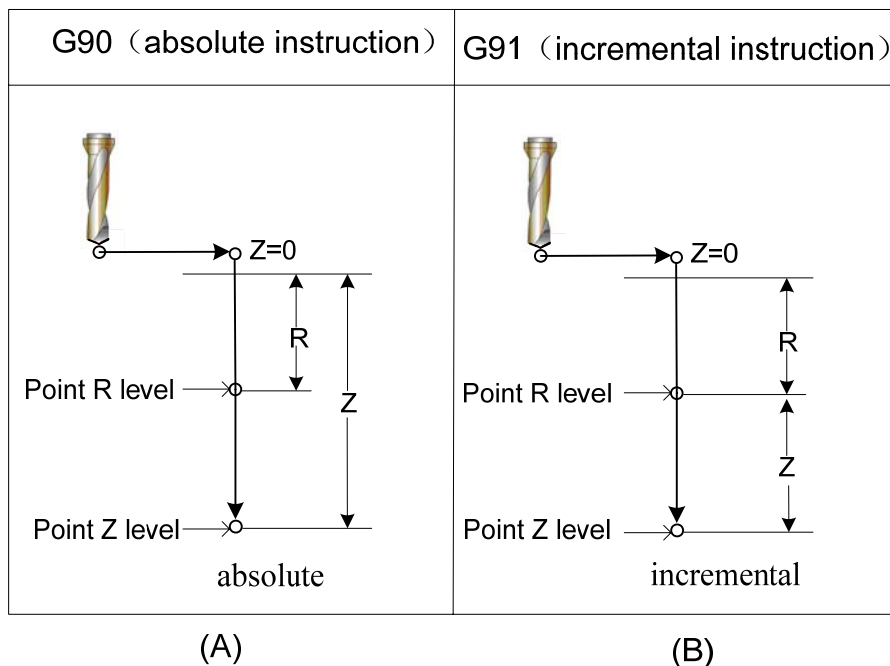


Fig. 4-4-2

#### B Return to initial level in canned cycle G98/G99

After the tool reaches the bottom of a hole, it may return to the point R level or the initial level. These operations can be specified by G98 and G99.

Generally, G99 is used for the 1<sup>st</sup> drilling operation and G98 is used for the last drilling operation. The initial level does not change even drilling is performed in G99 mode. The following figure illustrates the operation of G98 and G99.

G98 is the system default mode.

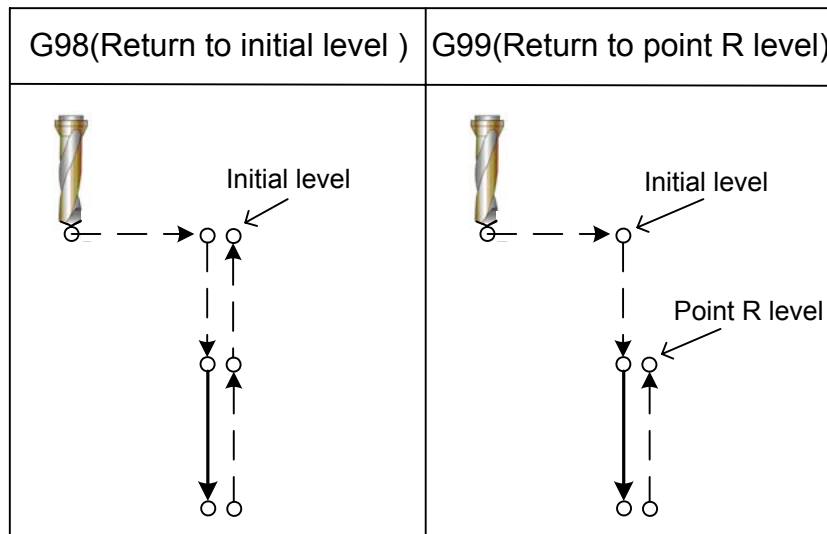


Fig. 4-4-3

The following symbols are used for the canned cycle illustration:

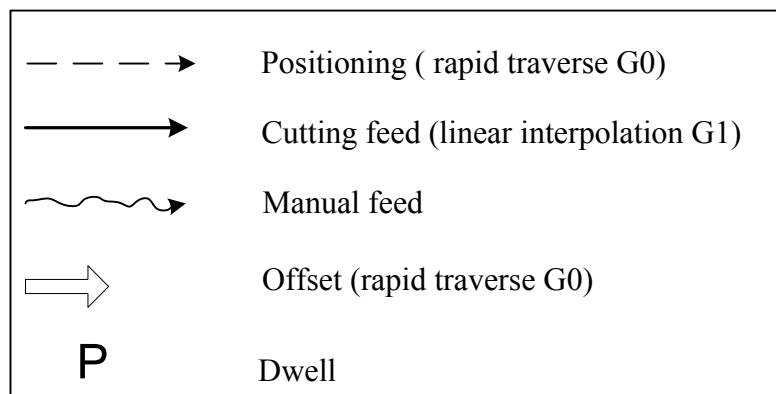


Fig. 4-4-4

Canned cycle comparison table (G22~G89)

Table 4-4-3

G code	Drilling (-Z direction)	Operation at the hole bottom	Retraction(+Z direction)	Application
G22	Cutting feed		Rapid feed	CCW inner circular groove rough milling
G23	Cutting feed		Rapid feed	CW inner circular groove rough milling
G24	Cutting feed		Rapid feed	CCW finish-milling within a circle cycle
G25	Cutting feed		Rapid feed	CW finish-milling within a circle cycle
G26	Cutting feed		Rapid feed	CCW outer circle finish-milling cycle
G32	Cutting feed		Rapid feed	CW outer circle finish-milling cycle
G33	Cutting feed		Rapid feed	CCW rectangle groove rough milling
G34	Cutting feed		Rapid feed	CW rectangle groove rough milling
G35	Cutting feed		Rapid feed	CCW inner rectangle groove finish-milling cycle

G36	Cutting feed		Rapid feed	CW inner rectangle groove finish-milling cycle
G37	Cutting feed		Rapid feed	CCW rectangle outside finish-milling cycle
G38	Cutting feed		Rapid feed	CW rectangle outside finish milling cycle
G73	Intermittent feed		Rapid feed	High-speed peck drilling cycle
G74	Cutting feed	Dwell→spindle CW	Cutting feed	Counter tapping cycle
G76	Cutting feed	Oriented spindle stop	Rapid feed	Fine boring
G80				Cancel
G81	Cutting feed		Rapid feed	Drilling, spot drilling
G82	Cutting feed	Stop	Rapid feed	Drilling, counterboring
G83	Intermittent feed		Rapid feed	Peck drilling cycle
G84	Cutting feed	Dwell → spindle CCW	Cutting feed	Tapping
G85	Cutting feed		Cutting feed	Boring
G86	Cutting feed	Spindle stop	Rapid feed	Boring
G87	Cutting feed	Spindle CCW	Rapid feed	Boring
G88	Cutting feed	Dwell → spindle CCW	JOG	Boring
G89	Cutting feed	Dwell	Rapid feed	Boring

**Restriction**

During canned cycle positioning, tool radius offset (D) is ignored.

**4.4.1 Rough milling of inner circular groove G22/G23****Format:****G22****G98/G99**      **X\_ Y\_ Z\_ R\_ I\_ L\_ W\_ Q\_ V\_ D\_ F\_ K\_****G23**

**Function:** They are used for circular interpolations from the circle center by helical type till the circular groove programmed is machined.

**Explanation:**

G22: CCW inner circular groove rough milling

G23: CW inner circular groove rough milling

X、Y: The start point position within X, Y plane

Z: Machining depth, absolute position in G90, position to R reference level in G91

R: R reference level, absolute position in G90, position to start point of this block in G91

I: Circular groove radius, it should be over the current tool radius

L: Cut width increment within XY plane, less than tool diameter but more than 0;

W: Initial cut depth in Z axis, which is the distance below R reference level and it

is over 0 (if the initial cut depth exceeds the groove bottom, it should machine by this bottom) ;

Q: Cut depth of each feed;

V: Distance to the end surface at rapid tool traverse, which is over 0;

D: Tool compensation number, ranging within 0 ~ 256, D0 is defaulted for 0. The current tool

diameter value is got by the given number.

K: Repetitions.

#### Cycle process:

- (1) Rapid to a location in XY plane;
- (2) Rapid down to R level;
- (3) To cut W depth downward by cutting feedrate;
- (4) From center outward to mill a circle surface with a radius I helically by a L increment each time;
- (5) Z axis rapidly returns to R level;
- (6) X, Y axes rapidly position to the circle center;
- (7) Z axis rapid downward to a location with a distance V to the end surface;
- (8) To cut a (Q+V) depth downward in Z axis;
- (9) Repeat the actions from (4) ~ (8) till the total depth of circle surface is finished;
- (10) Return to initial level or R level according to G98 or G99 instruction.

#### Instruction path:

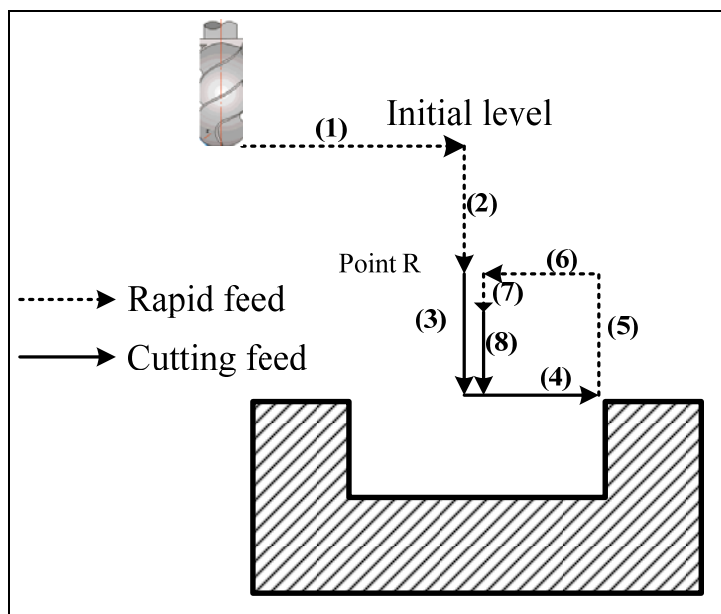


Fig. 4-4-1-1

G22: CCW inner circle groove rough milling

G23: CW inner circle groove rough milling

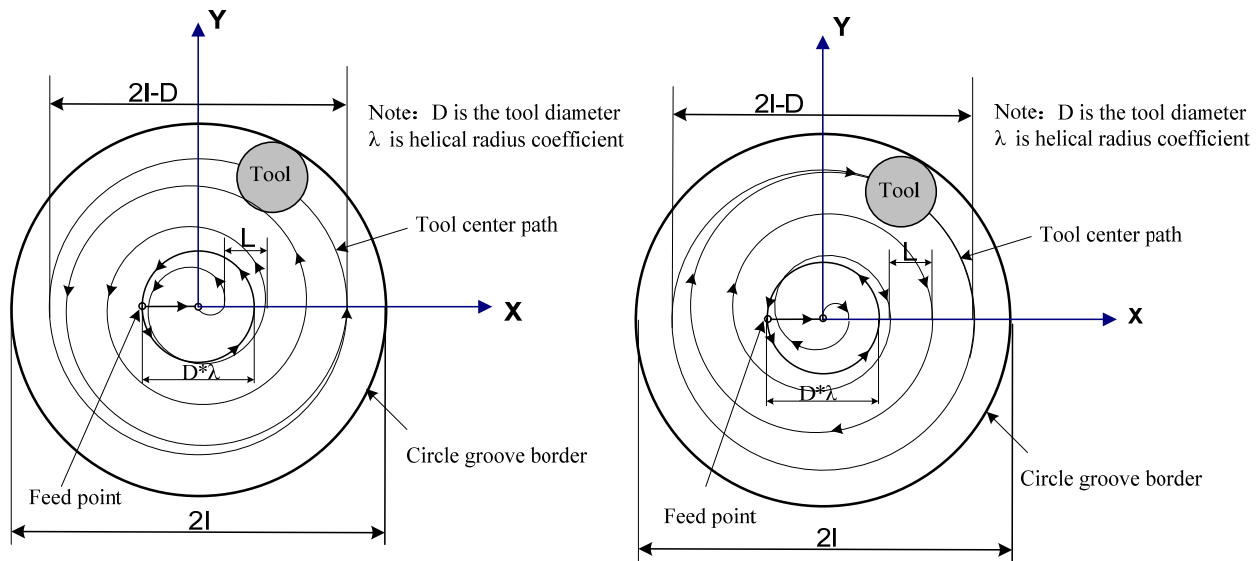


Fig. 4-4-1-2

**Note:** 1、The NO: 12#1 should be set to 1 when the instruction is used.

2、The helical radius coefficient in groove cycle should be over 0, if the programmed speed is over F15, it feeds by the speed of F15, if the programmed speed is less than F15, it feeds by the programmed one.

**Example:** To rough mill a groove within a circle by canned cycle G22 instruction, which is as follows:

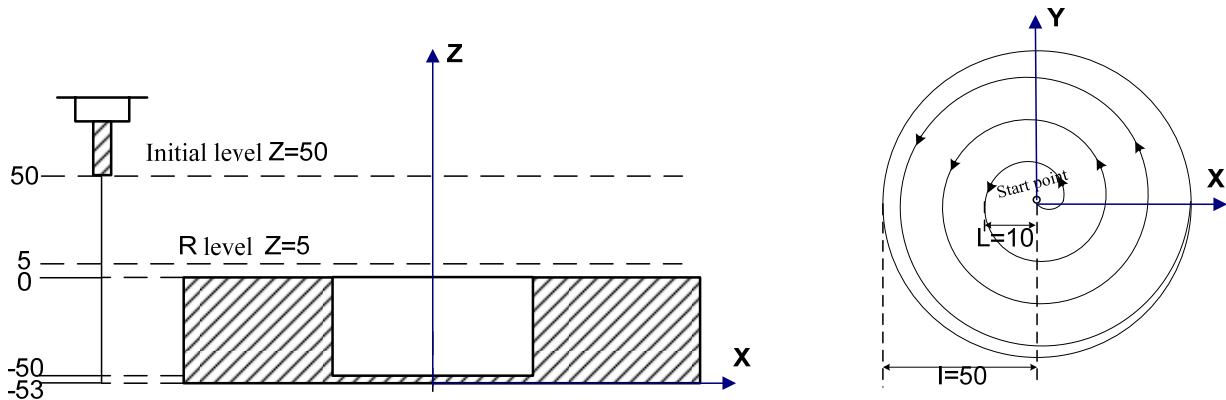


Fig. 4-4-1-3

G90 G00 X50 Y50 Z50; (G00 rapid positioning)

G99 G22 X25 Y25 Z-50 R5 I50 L10 W20 Q10 V10 F800;

(Groove rough milling cycle within a circle)

G80 X50 Y50 Z50; (Canned cycle cancel and return from R level)

M30;

**Restriction:**

Cancellation: G codes of 01 group (G00 to G03), G60 modal G code (bit parameter NO: 48#0 is set to 1) and G22/G23 cannot be specified in a same block, or G22/G23 will be cancelled.

Tool offset: The tool radius offset in canned cycle is omitted.

#### 4.4.2 Fine milling cycle within a circle G24/G25

**Format:**

**G24**  
**G98/G99**      **X\_ Y\_ Z\_ R\_ I\_ J\_ D\_ F\_ K\_**  
**G25**

**Function:** They are used to fine mill a circle by a radius I and direction specified and the tool returns after milling.

**Explanation:**

G24: CCW fine milling within a circle

G25: CW fine milling within a circle

X、Y: The start point position within X, Y plane

Z: Machining depth, absolute position in G90, position to R reference level in G91

R: R reference level, absolute position in G90, position to start point of this block in G91

I: Milling circle radius, ranging within 0 mm ~9999.999mm, use absolute value if it is a negative one;

J: Distance of fine milling start point to circle center, ranging with 0 mm ~9999.999mm, use absolute value if it is a negative one;

D: Tool compensation number, ranging within 0 ~128. D0 is defaulted for 0. The tool diameter value is obtained by the given number.

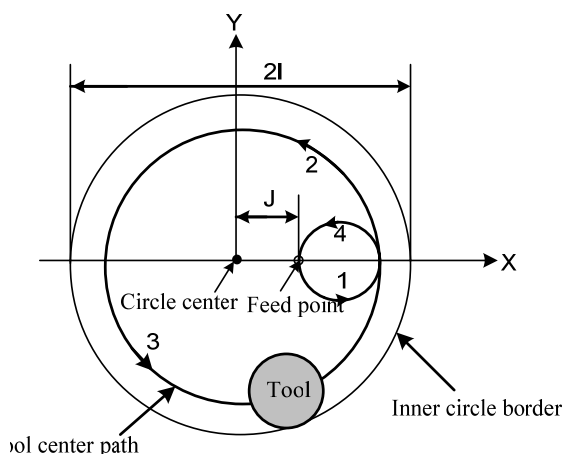
K: Repetitions

**Cycle process:**

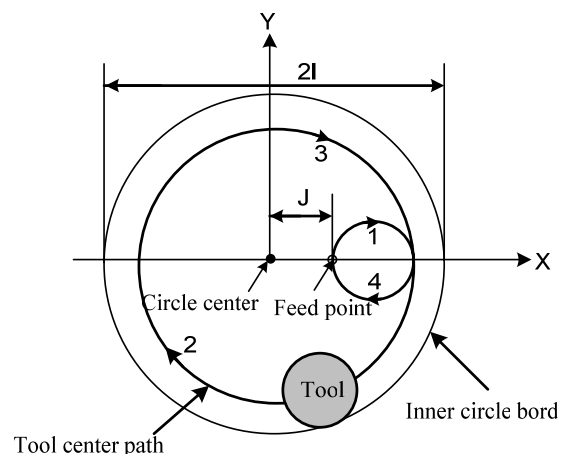
- (1) Rapid to a location within XY plane;
- (2) Rapid down to R level;
- (3) Feed to the hole bottom;
- (4) To interpolate by the transition arc 1 from the start point;
- (5) To make circular interpolation for the whole circle by inner arc path of finish-milling.
- (6) To make circular interpolation by transition arc 4 and return to the start point;
- (7) Return to the initial level or R level according to G98 or G99 instruction.

**Instruction path:**

24: CCW finish milling within a circle cycle



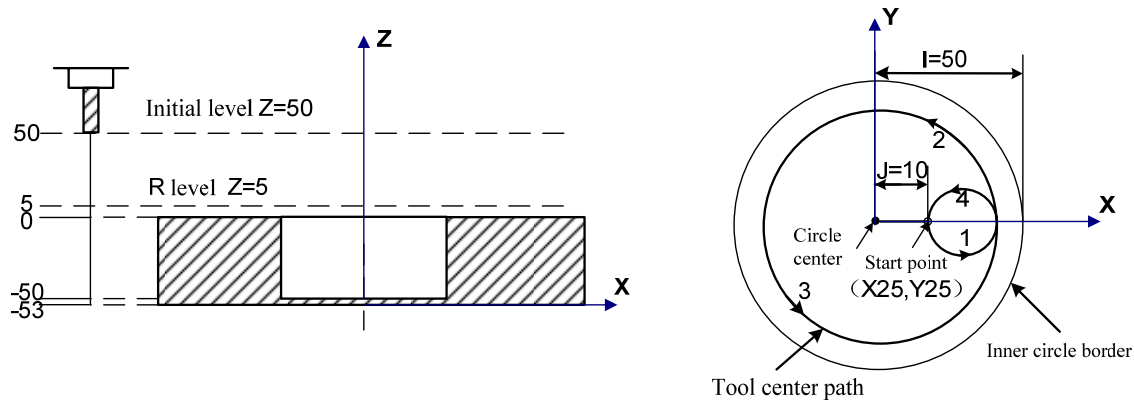
G25: CW finish milling within a circle cycle



**Fig. 4-4-2-1**

**Note:** The **NO: 12#1** should be set to 1 when the instruction is used.

**Example:** To fine mill a circular groove that has been rough milled as following by canned cycle G24 instruction:



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

```
G90 G00 X50 Y50 Z50;      (G00 rapid positioning)
G99 G24 X25 Y25 Z-50 R5 I50 J10 D1 F800;  (Canned cycle starts, and goes down to the
                                           bottom to perform the inner circle finish milling)
G80 X50 Y50 Z50;          (To cancel canned cycle and return from R level)
M30;
```

**Restriction:**

**Cancellation:** G codes of 01 group (from G00 to G03), G60 modal G code (bit parameter NO: 48#0 is set to 1) and G24/G25 cannot be specified in a same block, or G24/G25 will be cancelled.

**Tool offset:** The tool radius offset in canned cycle is ignored.

#### 4.4.3 Outer circle finish milling cycle G26/G32

**Format:**

```

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

**Function:** They are used to finish milling a circle outside a circle by the specified radius and direction and the tool returns after milling.

**Explanation:**

G26: CCW outer circle finish milling cycle

G32: CW outer circle finish milling cycle

X、Y: The start point within X, Y plane

Z: Machining depth, which is absolute position in G90 and position to R reference level in G91

R: R reference level, which is absolute position in G90 and position to start point of this block in G91

I: Finish milling circle radius, ranging within 0 mm ~9999.999mm, use the absolute value if it is a negative one.

J: Distance from the milling start point to milling circle center, ranging within



0 mm ~9999.999mm, use the absolute value if it is a negative one

D: Tool radius number, ranging within 0 ~256, D0 is defaulted for 0. The current tool radius value is obtained by the given number.

K: Repetitions.

#### Cycle process:

- (1) Rapid to a location within XY plane;
- (2) Rapid down to R level;
- (3) Feed to the hole bottom;
- (4) To position to the start point from current position at the bottom;
- (5) To interpolate by the transition arc 1 from the start point;
- (6) To make circular interpolation for the whole circle by arc 2, arc 3
- (7) To make circular interpolation by transition arc 4 and return to the start point;
- (8) Return to the initial level or R level according to G98 or G99 instruction.

#### Instruction path:

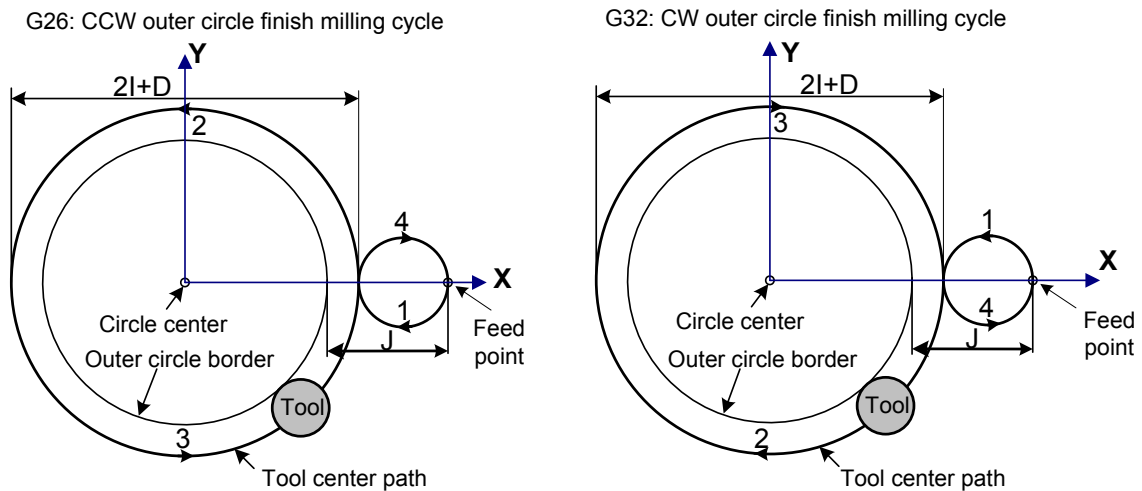


Fig. 4-4-3-1

#### Explanation:

In outer circle finish milling, the interpolation directions of transition arc and finish milling arc are different, and the interpolation direction in the instruction means the interpolation direction of the finish milling.

**Example:** To finish mill a circular groove that has been rough milled as following by canned cycle G26 instruction:

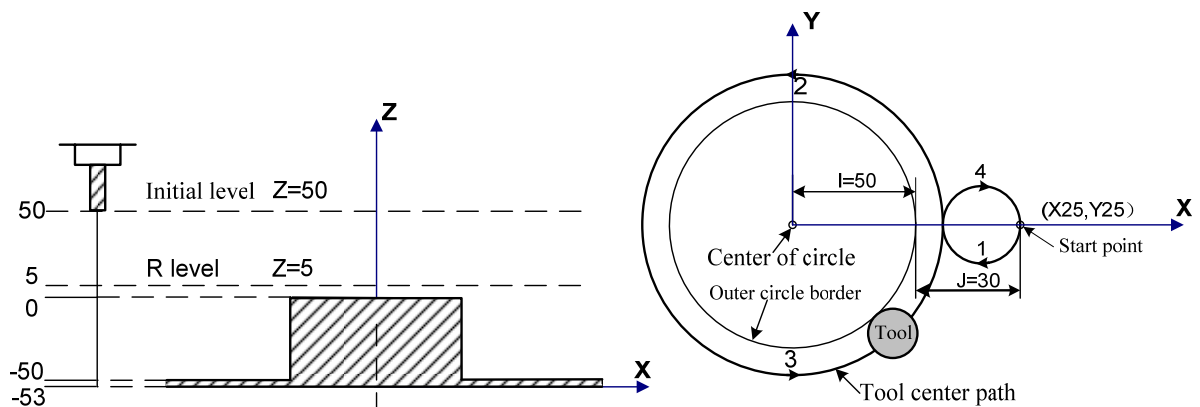


Fig. 4-4-3-2

G90 G00 X50 Y50 Z50; (G00 rapid positioning)  
 G99 G26 X25 Y25 Z-50 R5 I50 J30 D1 F800; (Canned cycle starts, and goes down to the bottom to perform the outer circle finish milling)  
 G80 X50 Y50 Z50; (To cancel canned cycle and return from R level)  
 M30;

**Restriction:**

**Cancellation :** G codes of 01 group (G00 to G03), G60 modal G code (bit parameter NO: 48#0 is set to 1) and G26/G32 cannot be specified in a same block, or G26/G32 will be cancelled.

**Tool offset :** The tool radius offset in canned cycle is ignored.

**4.4.4 Rectangular groove rough milling G33/G34****Format:**

**G33**  
**G98/G99**            **X\_ Y\_ Z\_ R\_ I\_ J\_ L\_ W\_ Q\_ V\_ U\_ D\_ F\_ K\_**  
**G34**

**Function:** These instructions are used for linear cutting cycle from the rectangle center by the parameter data specified till the rectangular groove programmed is machined.

**Explanation:**

G33: CCW rectangular groove rough milling

G34: CW rectangular groove rough milling

X、Y: The start point within X, Y plane

Z: Machining depth which is absolute position in G90 and position to R reference plane in G91

R: R reference plane which is absolute position in G90 and position to start point of this block in G91

I: Rectangular groove width in X axis, which should be over the tool radius and helical feed radius should be less than half of it .

J: Rectangular groove width in Y axis, which should be over the tool radius and helical feed radius should be less than half of it .

L: Cutting width increment within a specified plane, which should be less than the tool diameter and over 0

W: Initial cut depth in Z axis, which is a downward distance from R level and is over 0 (if the initial cut exceeds the groove bottom, it will cut at the bottom position)

Q: Cutting depth of each cutting feed

V: Distance to the end surface to be machined in rapid feed, which is over 0

U: Corner arc radius, no corner arc transition if omitted, U should be more than or equal to the tool radius.

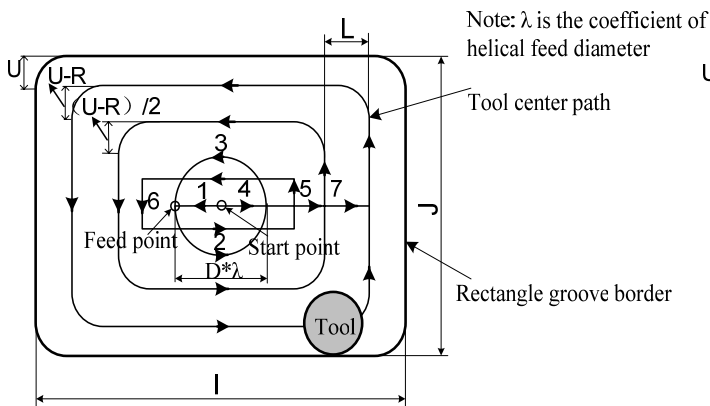
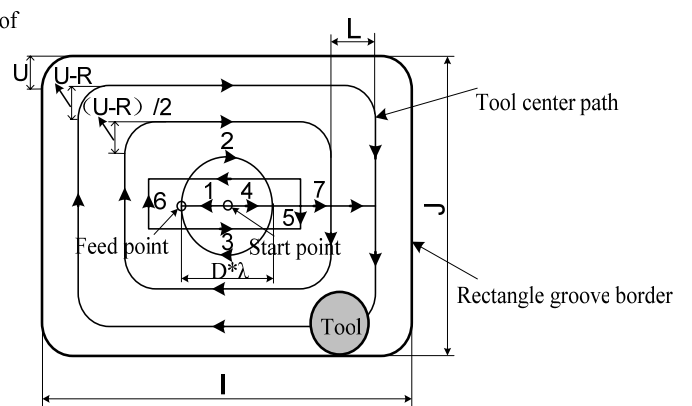
D: Tool compensation number, ranging within 0 ~ 256, D0 is defaulted for 0. The current tool diameter value is got according to the number specified.

K: Repetitions

K: Repetitions

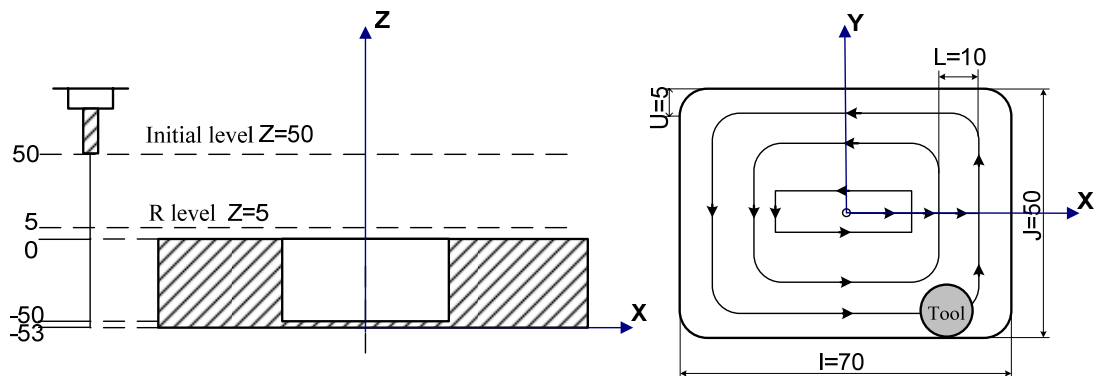
**Cycle process:**

- (1) Rapid to a start point within XY plane;
- (2) Rapid down to R level;
- (3) The diameter helical feed W width can be gotten by radius compensation value multiplying the parameter NO. 269 value
- (4) Feed to the rectangle center X0, Y0;
- (5) To mill a rectangular surface helically from center outward by L increment each time;
- (6) Z axis rapids to R level;
- (7) X, Y axes rapidly locates to the rectangle center;
- (8) Z axis rapids down to a position that has a V distance to the end surface;
- (9) Z axis cuts downward for a (Q+V) depth;
- (10) Repeat the actions of (4) ~ (8) till the rectangular surface with the total depth is machined;
- (11) Return to the initial level or R level according to G98 or G99 instruction.

**Instruction path:****G33 CCW rectangle groove rough milling****G34 CW rectangle groove rough milling****Fig. 4-4-4-1**

**Note:** The NO: 12#1 should be set to 1 when the instruction is used.

**Example:** To rough mill an inner rectangular groove as shown in the following by canned cycle G33 instruction:

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

G90 G00 X50 Y50 Z50; (G00 rapid positioning)

G99 G33 X25 Y25 Z-50 R5 I70 J50 L10 W20 Q10 V10 U5 F800;

(To run the inner rectangular groove rough milling cycle)

G80 X50 Y50 Z50; (To cancel canned cycle and return from R level)  
M30;

**Restriction:**

**Cancellation:** G codes of 01 group (G00 to G03), G60 modal G code (bit parameter NO: 48#0 is set to 1) and G33/G34 cannot be specified in a same block, or G33/G34 will be cancelled.

**Tool offset:** The tool radius offset in canned cycle is ignored.

**4.4.5 Inner rectangular groove finish milling cycle G35/G36****Format:**

**G35**  
**G98/G99**                      **X\_ Y\_ Z\_ R\_ I\_ J\_ L\_ U\_ D\_ F\_ K\_;**  
**G36**

**Function:** They are used for finish milling within a rectangle by the width and direction specified, and the tool returns after finish milling.

**Explanation:**

G35: CCW inner rectangular groove finish milling cycle

G36: CW inner rectangular groove finish milling cycle

X、Y: The start point within X, Y plane

Z: Machining depth which is absolute position in G90 and position to R reference plane in G91

R: R reference plane which is absolute position in G90 and position to start point of this block in G91

I: Rectangular width in X axis, ranging within 0~9999.999mm

J: Rectangular width in Y axis, ranging within 0~9999.999mm

L: Distance of start point to rectangular side in X axis, ranging within 0~9999.999mm;

U: Corner radius, no corner transition if omitted. Alarm is issued if U is omitted or equal to 0 and the tool radius is over 0.

D: Tool compensation number, ranging within 0 ~ 256, D0 is defaulted for 0. The current tool diameter value is got according to the number specified.

K: Repetitions

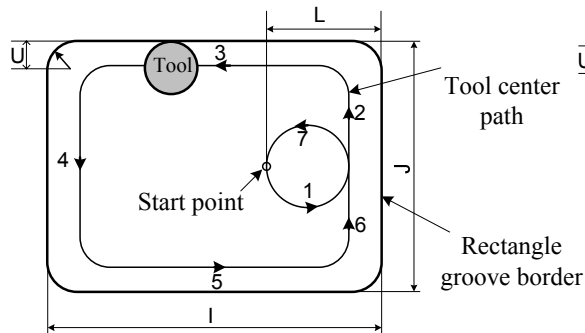
K: Repetitions.

**Cycle process:**

- (1) Rapid to a location within XY plane;
- (2) Rapid down to R level;
- (3) Feed to the hole bottom;
- (4) To position to the start point from current position at the bottom;
- (5) To make circular interpolation by the transition arc 1 from the start point;
- (6) To make linear and circular interpolation by the path 2-3-4-5-6;
- (7) To make circular interpolation by the path of transition arc 7 and return to the start point;
- (8) Return to the initial level or R level according to G98 or G99 instruction.

### Instruction path:

G35: CCW rectangle groove finish milling cycle



G36: CW rectangle groove finish milling cycle

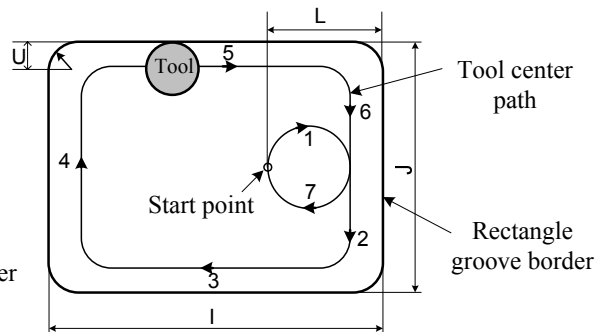


Fig. 4-4-5-1

**Note** The NO:12#1 should be set to 1 when the instruction is used.

**Example:** To finish mill a circular groove that has been rough milled as following by canned cycle G35 instruction:

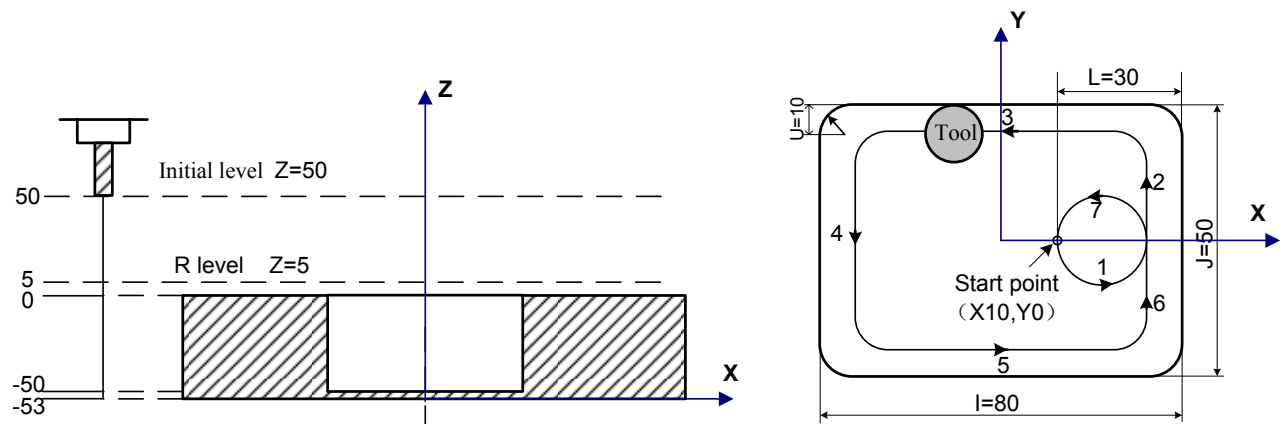


Fig. 4-4-5-2

G90 G00 X50 Y50 Z50; (G00 rapid positioning)

G99 G35 X10 Y10 Z-50 R5 I80 J50 L30 U10 F800;

(Canned cycle starts, and go down to the bottom to perform the innerrectangular groove finish milling)

G80 X50 Y50 Z50; (To cancel canned cycle and return from R level)

M30;

### Restriction:

**Cancellation:** G codes of 01 group (G00 to G03), G60 modal G code (bit parameter NO: 48#0 is set to 1) and G35/G36 cannot be specified in a same block, or G35/G36 will be cancelled.

**Tool offset:** The tool radius offset in canned cycle is ignored.

## 4.4.6 Rectangle outside finish milling cycle G35/G36

### Format:

G37

G98/G99

X\_ Y\_ Z\_ R\_ I\_ J\_ L\_ U\_ D\_ F\_ K\_

**G38**

**Function:** They are used for finish milling outside a rectangle by the width and direction specified, and the tool returns after finish milling.

**Explanation:**

G37: CCW rectangle outside finish milling cycle

G38: CW rectangle outside finish milling cycle

I: Rectangular width in X axis, ranging within 0 mm ~99999.999mm

J: Rectangular width in Y axis, ranging within 0 mm ~99999.999mm

L: Distance of start point to rectangular side in X axis, ranging within 0~9999.999mm

U: Corner radius, no corner transition if omitted

D: Tool compensation number, ranging within 0 ~ 256, D0 is defaulted for 0. The current tool diameter value is got according to the number specified.

K: Repetitions

K: Repetitions

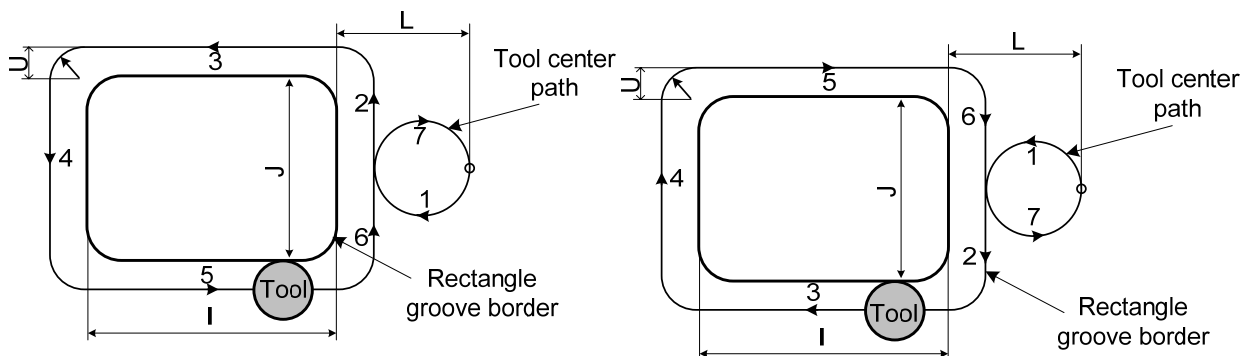
**Cycle process:**

- (1) Rapid to a location within XY plane;
- (2) Rapid down to R level;
- (3) Feed to the hole bottom;
- (4) To position to the start point from current position at the bottom;
- (5) To make circular interpolation by the transition arc 1 from the start point;
- (6) To make linear and circular interpolation by the path 2-3-4-5-6;
- (7) To make circular interpolation by the path of transition arc 7 and return to the start point;
- (8) Return to the initial level or R level according to G98 or G99 instruction.

**Instruction path:**

G37 CCW rectangle outside finish milling cycle

G38 CW rectangle outside finish milling cycle



**Fig. 4-4-6-1**

**Explanation:**

For the rectangle outside finish milling, the interpolation direction of the transition arc is not consistent with that of the finish milling arc, and the interpolation direction in explanation means that of the finish milling arc.

**Example:** Perform the rectangle outside finish milling by canned cycle G37 instruction:

G90 G00 X50 Y50 Z50; (G00 rapid positioning)

G99 G37 X25 Y25 Z-50 R5 I80 J50 L30 U10 F800;

(Canned cycle starts, and go downward to the bottom to

perform the rectangular groove finish milling)

G80 X50 Y50 Z50; (To cancel canned cycle and return from R level)

M30;

#### Restriction:

**Cancellation:** G codes of 01 group (G00 to G03), G60 modal G code (bit parameter NO: 48#0 is set to 1) and G37/G38 cannot be specified in a same block, or G37/G38 will be cancelled.

**Tool offset:** The tool radius offset during positioning in canned cycle is ignored.

### 4.4.7 High-speed peck drilling cycle G73

**Format:** G73 X\_Y\_Z\_R\_Q\_F\_K\_

**Function:** This cycle is especially defined for high-speed peck drilling, it performs intermittent cutting feed to the bottom of a hole while removing chips from the hole by rapid retraction. The operation illustration is shown as Fig. 4-4-7-1.

#### Explanation:

X\_Y\_: Hole positioning data

Z\_: In incremental programming it specifies the distance from point R level to the bottom of the hole; in absolute programming it specifies the absolute coordinate of the hole bottom.

R\_: In incremental programming it specifies the distance from the initial level to point R level; in absolute programming it specifies the absolute coordinate of point R.

Q\_: Depth of cut for each cutting feed

F\_: Cutting feedrate

K\_: Number of repeats

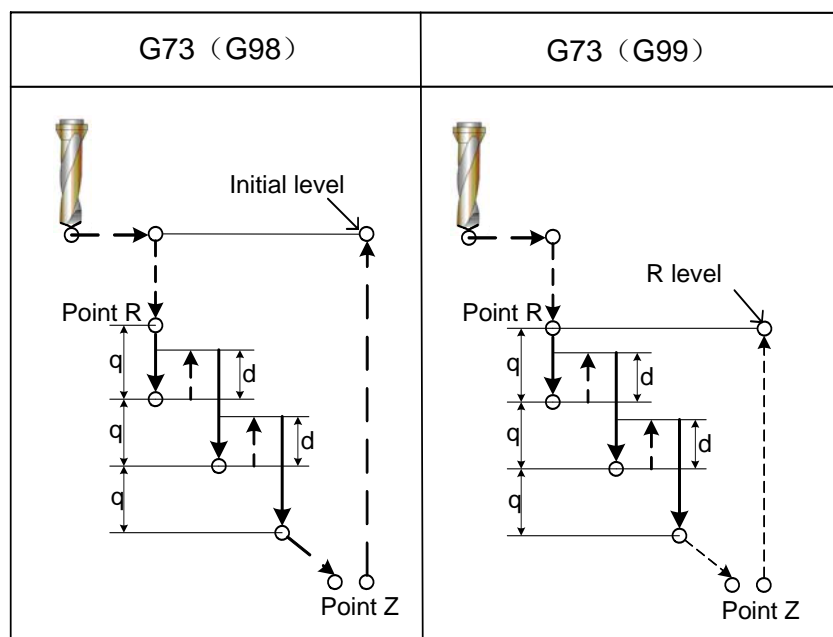


Fig. 4-4-7-1

**Z、R:** if either of the hole bottom parameter Z and R is missed while performing the 1<sup>st</sup> drilling operation, the system only change the modal state and does not perform the Z axis operation.

**Q:** If parameter Q is specified, the intermittent feed is performed as shown in above figure. And the retraction is performed by the retraction value d (Fig.4-4-7-1) set in number parameter P270. The rapid tool retraction for a distance d is performed in each intermittent feeding.

If G73 and M codes are specified in a same block, M code is executed during the 1<sup>st</sup> hole positioning operation, then the system goes on the next drilling operation.

If the repetition K is specified, M code is only executed for the first hole, not for the following holes.

**Note 1** If parameter Q is not specified, alarm "address Q not found or Q value is 0 (G73/G83)" will be issued. If Q value is specified for a negative, the intermittent feed will be performed by the absolute value of Q.

**Note 2** In canned cycle, if the tool length compensation (G43, G44 or G49) is specified, the offset value is either added or cancelled while positioning to point R level.

**Note 3** Tool length compensation: If the tool length compensation instruction G43, G44 or G49 is specified in the same block with a canned cycle instruction, the offset is added or cancelled when the tool is positioned to point R; If the tool compensation instruction G43, G44 or G49 is specified in a separate block in the canned cycle mode, the system can add or cancel the offset in real time.

#### Restriction:

**Cancellation:** G codes of 01 group (G00 to G03), G60 modal G code (bit parameter NO: 48#0 is set to 1) and G73 cannot be specified in a same block, otherwise G73 will be cancelled.

**Tool offset:** The tool radius offset during positioning in canned cycle is ignored.

#### Example 1

M3 S1500	Spindle running start
G90 G99 G73 X0 Y0 Z-15. R-10.Q5. F120.	Positioning and drill hole 1 then return to point R level
Y-50;	Positioning and drill hole 2 then return to point R level
Y-80;	Positioning and drill hole 3 then return to point R level
X10;	Positioning and drill hole 4 then return to point R level
Y10;	Positioning and drill hole 5 then return to point R level
G98 Y75;	Positioning and drill hole 6 then return to initial level
G80;	
G28 G91 X0 Y0 Z0;	Return to reference point
M5;	Spindle stop
M30;	

**Note:** The chip removal operation is still performed though Q is omitted in the machining of the holes from 2 to 6.



#### 4.4.8 Drilling cycle, spot drilling cycle G81

**Format:** G81 X\_ Y\_ Z\_ R\_ F\_ K\_

**Function:** It is used for normal drilling feed to the hole bottom, then the tool rapidly retracts from the hole bottom.

**Explanation:**

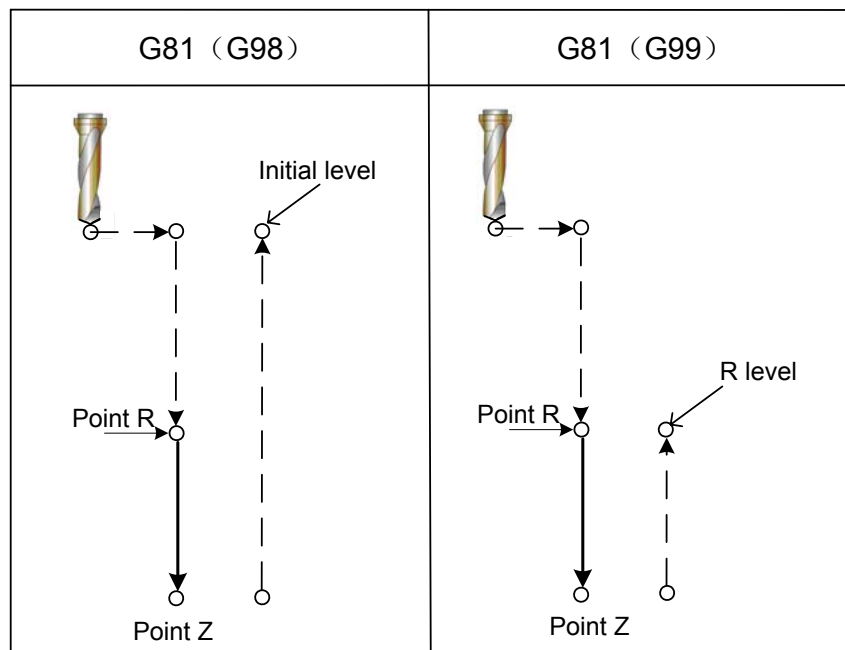
X\_ Y\_: Hole positioning data

Z\_: In incremental programming it specifies the distance from point R level to the bottom of the hole; in absolute programming it specifies the absolute coordinate of the hole bottom.

R\_: In incremental programming it specifies the distance from the initial level to point R level; in absolute programming it specifies the absolute coordinate of point R level.

F\_: Cutting feedrate

K\_: Number of repeats (if necessary)



**Fig. 4-4-8-1**

**Z、R:** if either of the hole bottom parameter Z and R is missed while performing the 1<sup>st</sup> drilling operation, the system only change the modal state and does not perform the Z axis operation.

After positioning along X and Z axes, the tool traverses to point R level to perform the drilling from point R level to point Z level, then retracts rapidly.

The spindle is rotated by miscellaneous function M code before G81 is specified.

If G81 and M code are specified in a same block, M code is executed while the 1<sup>st</sup> hole positioning operation is being performed, then the system goes on next drilling operation.

If number of repeats K is specified, M code is only executed for the 1st hole, not for the following holes.

If the tool length compensation G43, G44 or G49 is specified in canned cycle, the offset is either

added or cancelled while positioning to point R level.

Example

M3 S2000	Spindle running start
G90 G99 G81 X300. Y-250. Z-150. R-10. F120.	Positioning, drill hole 1, then return to point R level
Y-550.;	Positioning, drilling hole 2, then return to point R level
Y-750.;	Positioning, drill hole 3, then turn to point R level
X1000.;	Positioning, drill hole 4, then return to point R level
Y-550.;	Positioning, drill hole 5, then return to point R level
G98 Y-750.;	Positioning, drill hole 6, then return to initial level
G80;	
G28 G91 X0 Y0 Z0 ;	Return to reference point
M5;	Spindle stops
M30;	

**Restriction:**

**Cancellation:** G codes of 01 group (G00 to G03), G60 modal G code (bit parameter NO: 48#0 is set to 1) and G81 cannot be specified in a same block, otherwise G81 will be cancelled.

**Tool offset:** The tool radius offset during positioning in canned cycle is ignored.

#### 4.4.9 Drilling cycle, counterboring cycle G82

**Format:** G82 X\_ Y\_ Z\_ R\_ P\_ F\_ K\_;

**Function:** It is used for normal drilling to feed to the hole bottom and dwell, then retract the tool rapidly from hole bottom.

**Explanation:**

X_ Y_:	Hole positioning data
Z_:	In incremental programming it specifies the distance from point R level to the bottom of the hole; in absolute programming it specifies the absolute coordinate of the hole bottom.
R_:	In incremental programming it specifies the distance from the initial level to point R level; in absolute programming it specifies the absolute coordinate of point R.
F_:	Cutting feedrate
P_:	Dwell time
K_:	Number of repeats

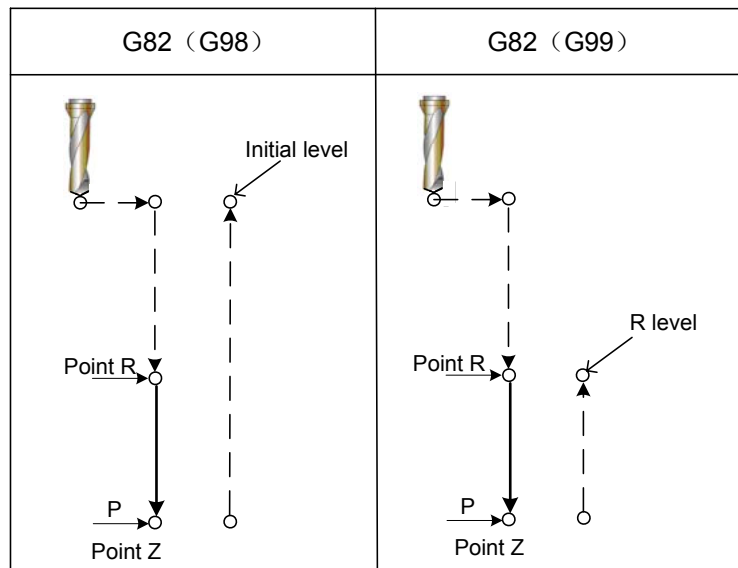


Fig. 4-4-9-1

After positioning along X and Z axes, the tool traverses to point R level to perform the drilling from point R level to point Z level, then dwells and returns rapidly after the tool reaches the hole bottom.

The spindle is rotated by miscellaneous function M code before G82 is specified.

If G82 and M code are specified in a same block, M code is executed while the 1<sup>st</sup> hole positioning operation is being performed, then the system goes on next drilling operation.

If number of repeats K is specified, M code is only executed for the 1st hole, not for the following holes.

**Note:** In the current version, M00, M01, M02, M06, M30, M98 and M99 are the M codes executed after the other instructions in a block, i.e. these M codes are executed after the execution of the current statement block.

**Tool length compensation:** If the tool length compensation instruction G43, G44 or G49 is specified in the same block with the canned cycle instruction, the offset is added or cancelled at the time of positioning to point R level; If the tool compensation instruction G43, G44 or G49 is specified in a separate block in the canned cycle mode, the system can add or cancel the offset in real time.

P is a modal instruction, and the min. value of it is set by number parameter P281, the max. value by P282. If P value is less than the setting by P281, the min. value is effective; if P value is more than the setting by P282, the max. value is effective. If P is specified in a block containing no drilling, it can't be stored as a modal datum.

### Example

M3 S2000

Spindle running start

G90 G99 G82 X300. Y-250. Z-150. R-100. P1000 F120 Positioning, drill hole 1 with 1s dwell at the hole bottom, then return to point R level

Y-550; Positioning, drill hole 2 with 1s dwell at the hole bottom, then return to point R level

Y-750; Positioning, drill hole 3 with 1s dwell at the hole bottom, then return to point R level

X1000.; Positioning, drill hole 4 with 1s dwell at the hole bottom, then return to point R level

Y-550; Positioning, drill hole 5 with 1s dwell at the hole bottom, then return to point R level

G98 Y-750; Positioning, drill hole 6 with 1s dwell at the hole bottom, then return to initial level

G80; Cancel canned cycle  
 G28 G91 X0 Y0 Z0 ; Return to reference point  
 M5; Spindle stops  
 M30;

**Restriction:**

**Cancellation:** G codes of 01 group (G00 to G03), G60 modal G code (bit parameter NO: 48#0 is set to 1) and G82 cannot be specified in a same block, otherwise G82 will be cancelled.

**Tool offset:** The tool radius offset during positioning in canned cycle is ignored.

**4.4.10 Drilling cycle with chip removal G83**

**Format:** G83 X\_ Y\_ Z\_ R\_ Q\_ F\_ K\_

**Function:** It is used for peck drilling that the tool feeds to the hole bottom by intermittent feeding with chips removed from hole during drilling.

**Explanation:**

X\_Y\_: Hole positioning data

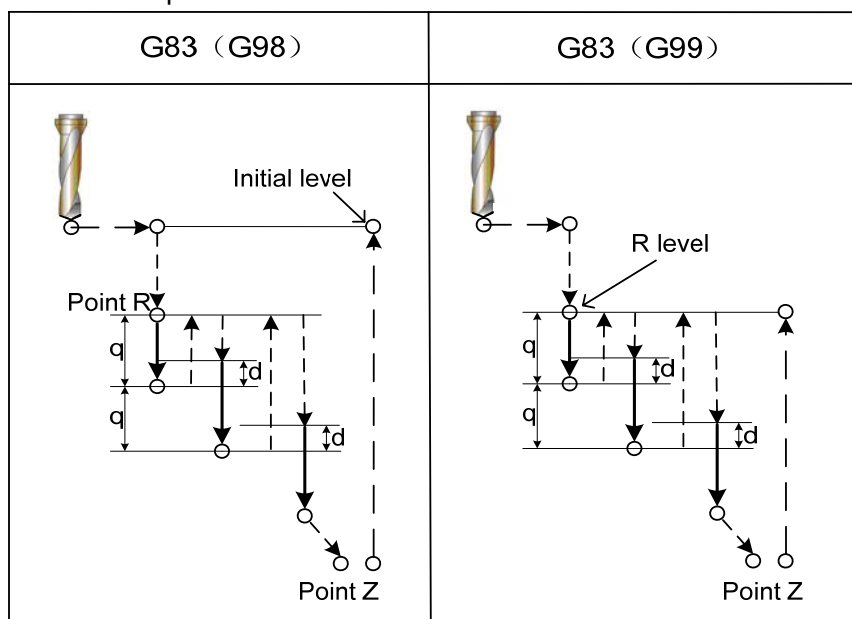
Z\_: In incremental programming it specifies the distance from point R level to the bottom of the hole; in absolute programming it specifies the absolute coordinate of the hole bottom.

R\_: In incremental programming it specifies the distance from the initial level to point R level; in absolute programming it specifies the absolute coordinate of point R.

Q\_: Depth of cut for each cutting feed

F\_: Cutting feedrate

K\_: Number of repeats



**Fig. 4-4-10-1**

**Q:** It specifies each cutting depth expressed by incremental value. In the second and the following

feeding, the tool rapidly traverse to the position which has a distance d to the end position of last drilling and still performs the feeding d that is set by parameter P270, as is shown in Fig. 4-4-10-1.

Only positive value can be specified for Q and the negative value is used as a positive one with its negative sign ignored.

Q is specified in drilling block, it can't be stored as a modal datum if it is specified in the block containing no drilling.

The spindle is rotated by miscellaneous function(M code) before G83 is specified.

If G83 and M code are specified in a same block, M code is executed while the 1<sup>st</sup> hole positioning operation is being performed, then the system goes on next drilling operation.

If number of repeats K is specified, M code is only executed for the 1st hole, not for the following holes.

**Note 1:** In the current version, M00, M01, M02, M06, M30, M98 and M99 are the M codes executed after the other instructions in a block, i.e. these M codes are executed after the execution of the current statement block.

**Note 2:** When the bit parameter NO: 43# 1=0, no alarm will be issued if there is no cut-in value specified in the peck drilling (G73, G83). At this moment, if the instruction parameter Q is not specified or it is 0, the system performs the hole positioning in XY plane, but it does not perform the drilling operation. When the bit parameter NO: 43#1=1, an alarm will be issued if no cut-in value is specified in the peck drilling (G73, G83), i.e. an alarm "0045: Address Q not found or set to 0(G73/G83)" occurs when the instruction parameter Q is not specified or it is 0. If the Q value is negative, the system uses its absolute value to perform intermittent feeding.

**Tool length compensation:** If the tool length compensation instruction G43, G44 or G49 is specified in the same block with the canned cycle instruction, the offset is added or cancelled at the time of positioning to point R level; If the tool compensation instruction G43, G44 or G49 is specified in a separate block in the canned cycle mode, the system can add or cancel the offset in real time.

### Example

M3 S2000	Spindle running start
G90 G99 G83 X300. Y-250. Z-150. R-100. Q15 F120;	Positioning, drill hole 1, then return to point R level
Y-550;	Positioning, drill hole 2, then return to point R level
Y-750;	Positioning, drill hole 3, then return to point R level
X1000;	Positioning, drill hole 4, then return to point R level
Y-550;	Positioning, drill hole 5, then return to point R level
G98 Y-750;	Positioning, drill hole 6, then return to initial level
G80;	
G28 G91 X0 Y0 Z0 ;	Return to reference point
M5;	Spindle stops
M30;	

### Restriction:

**Cancellation:** G codes of 01 group (G00 to G03), G60 modal G code (bit parameter

NO: 48#0 is set to 1) and G83 cannot be specified in a same block, otherwise G83 will be cancelled.

**Tool offset:** The tool radius offset during positioning in canned cycle is ignored.

#### 4.4.11 Right-handed tapping cycle G84

**Format:** G84 X\_ Y\_ Z\_ R\_ P\_ F\_

**Function:** It is used for tapping. In tapping, when the tool reaches the hole bottom, the spindle runs reversely.

**Explanation:**

X\_ Y\_: Hole positioning data

Z\_: In incremental programming it specifies the distance from point R level to the bottom of the hole; in absolute programming it specifies the absolute coordinate of the hole bottom.

R\_: In incremental programming it specifies the distance from the initial level to point R level; in absolute programming it specifies the absolute coordinate of point R.

P\_: Dwell time.

F\_: Cutting feedrate.

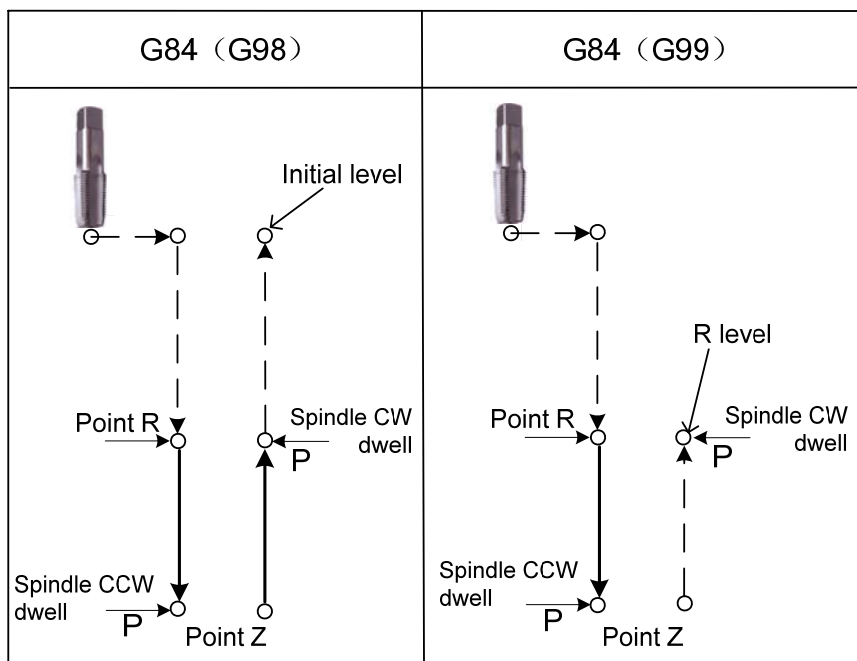


Fig. 4-4-11-1

Tapping is performed by rotating the spindle CW, when the tool reaches the hole bottom, the spindle is rotated reversely for retraction. This operation creates threads.

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

Before specifying G84, use a miscellaneous function (M code) to rotate the spindle. If the spindle CW rotation is not specified, it will be adjusted for CW rotation automatically in R level by the current spindle specification.

If G84 and M code are specified in a same block, M code is executed while the 1<sup>st</sup> hole positioning operation is being performed, then the system goes on next drilling operation.

If number of repeats K is specified, M code is only executed for the 1<sup>st</sup> hole, not for the following holes.

**Note:** In the current version, M00, M01, M02, M06, M30, M98 and M99 are the M codes executed after the other instructions in a block, i.e. these M codes are executed after the execution of the current statement block.

P is a modal instruction, and the min. value of it is set by number parameter P281, the max. value by P282. If P value is less than the setting by P281, the min. value is used; if P value is more than the setting by P282, the max. value is used. If P is specified in a block containing no drilling, it can't be stored as a modal datum.

**Tool length compensation:** If the tool length compensation instruction G43, G44 or G49 is specified in the same block with the canned cycle instruction, the offset is added or cancelled at the time of positioning to point R level; If the tool compensation instruction G43, G44 or G49 is specified in a separate block in the canned cycle mode, the system can add or cancel the offset in real time.

In feeding per minute, the relation between the thread lead and feedrate as well as spindle speed is as following:

$$\text{Feedrate } F = \text{tap pitch} \times \text{spindle speed } S$$

**For example:** for the M12×1.5 thread hole on the workpiece, the following parameter can be used:

$$S500 = 500 \text{ r/min} \quad F = 1.5 \times 500 = 750 \text{ mm/min}$$

For multi-start thread, F value can be got by multiplying the thread number.

**Example:**

M3 S100	Spindle running start
G90 G99 G84 X300. Y-250. Z-150. R-120 P300 F120	Positioning, tap hole 1, then return to point R level
Y-550.;	Positioning, tap hole 2, then return to point R level
Y-750.;	Positioning, tap hole 3, then return to point R level
X1000;	Positioning, tap hole 4, then return to point R level
Y-550.;	Positioning, tap hole 5, then return to point R level
G98 Y-750.;	Positioning, tap hole 6, then return to initial level
G80;	
G28 G91 X0 Y0 Z0 ;	Return to reference point
M5;	Spindle stops
M30;	

**Restriction:**

**Cancellation:** G codes of 01 group (G00 to G03), G60 modal G code (bit parameter NO: 48#0 is set to 1) and G84 cannot be specified in a same block, otherwise G84 will be cancelled.

**Tool offset:** The tool radius offset during positioning in canned cycle is ignored.

#### 4.4.12 Left-handed tapping cycle G74

**Format:** G74 X\_ Y\_ Z\_ R\_ P\_ F\_

**Function:** It is used for tapping cycle. In this tapping cycle, when the hole bottom is reached, the spindle rotates reversely.

**Explanation:**

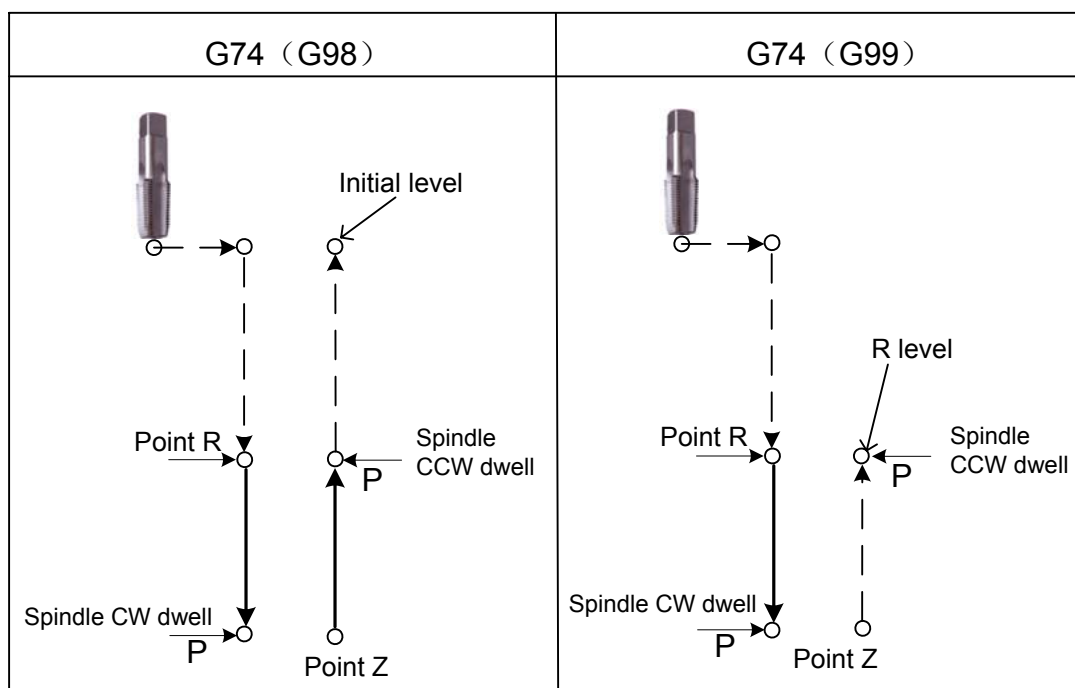
X\_Y\_: Hole positioning data

Z\_: In incremental programming it specifies the distance from point R level to the bottom of the hole; in absolute programming it specifies the absolute coordinate of the hole bottom.

R\_: In incremental programming it specifies the distance from the initial level to point R level; in absolute programming it specifies the absolute coordinate of point R.

P\_: Dwell time.

F\_: Cutting feedrate.



**Fig. 4-4-12-1**

Tapping is performed by rotating the spindle CCW, when the tool reaches the hole bottom, the spindle is rotated reversely for retraction. This operation creates threads.

Feedrate overrides are ignored during tapping. A feed hold does not stop the machine until the retraction operation is finished.

Before specifying G74, use a miscellaneous function (M code) to rotate the spindle. If the spindle CCW rotation is not specified, it will be adjusted for CCW rotation in R level automatically by the current spindle speed specified.

If G74 and M code are specified in a same block, M code is executed while the 1<sup>st</sup> hole positioning



operation is being performed, then the system goes on next drilling operation.

If number of repeats K is specified, M code is only executed for the 1<sup>st</sup> hole, not for the following holes.

**Note:** In the current version, M00, M01, M02, M06, M30, M98 and M99 are the M codes executed after the other instructions in a block, i.e. these M codes are executed after the execution of the current statement block.

P is a modal instruction, and the min. value of it is set by number parameter P281, the max. value by P282. If P value is less than the setting by P281, the min. value is used; if P value is more than the setting by P282, the max. value is used. If P is specified in a block containing no drilling, it can't be stored as a modal datum.

**Tool length compensation:** If the tool length compensation instruction G43, G44 or G49 is specified in the same block with the canned cycle instruction, the offset is added or cancelled at the time of positioning to point R level; If the tool compensation instruction G43, G44 or G49 is specified in a separate block in the canned cycle mode, the system can add or cancel the offset in real time.

If tool length compensation G43, G44 or G49 is specified in canned cycle, the offset value is either added or cancelled while positioning to point R level.

#### Example

M4 S100	Spindle running start
G90 G99 G74 X300. Y-250. Z-150. R-120 P300 F120	Positioning, tap hole 1, then return to point R level
Y-550.;	Positioning, tap hole 2, then return to point R level
Y-750.;	Positioning, tap hole 3, then return to point R level
X1000;	Positioning, tap hole 4, then return to point R level
Y-550.;	Positioning, tap hole 5, then return to point R level
G98 Y-750.;	Positioning, tap hole 6, then return to initial level
G80;	
G28 G91 X0 Y0 Z0 ;	Return to reference point
M5;	Spindle stops
M30;	

#### Restriction:

**Cancellation:** G codes of 01 group (G00 to G03), G60 modal G code (bit parameter NO: 48#0 is set to 1) and G74 cannot be specified in a same block, otherwise G74 will be cancelled.

**Tool offset:** The tool radius offset during positioning in canned cycle is ignored.

### 4.4.13 Fine boring cycle G76

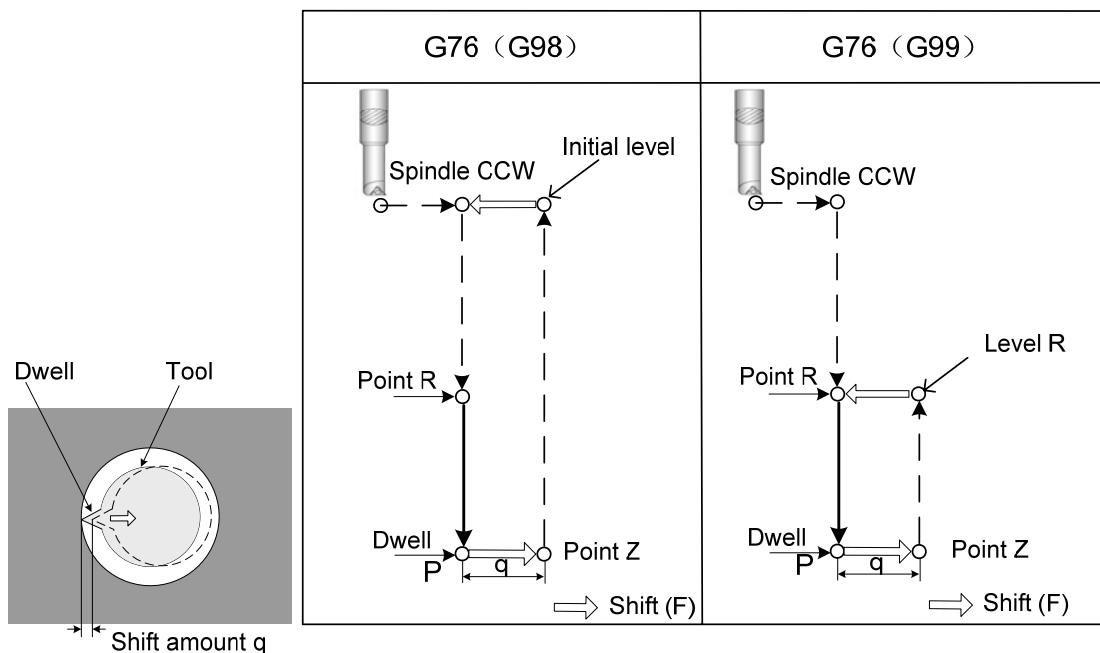
**Format:** G76 X\_Y\_Z\_Q\_R\_P\_F\_K\_

**Function:** It is used for boring a hole precisely.

When the tool reaches the hole bottom, the spindle stops and the tool departs from the machined surface of the workpiece and retracts. The retraction trail that affects machined surface finish and the tool damage should be avoided in the operation.

### Explanation:

- X\_Y\_: Hole positioning data  
 Z\_: In incremental programming it specifies the distance from point R level to the bottom of the hole; in absolute programming it specifies the absolute coordinate of the hole bottom.  
 R\_: In incremental programming it specifies the distance from the initial level to point R level; in absolute programming it specifies the absolute coordinate of point R level.  
 Q\_: Offset of the hole bottom  
 P\_: Dwell time.  
 F\_: Cutting feedrate.  
 K\_: Number of fine boring repeats



**Fig. 4-4-13-1**

When the tool reaches the hole bottom, the spindle stops at a fixed rotation position and the tool is moved in the direction opposite to the tool tip and retracted. This ensures that the machined surface is not damaged and enables precise and efficient boring. The parameter Q specifies the retraction distance and the retraction axis and direction are specified by bit parameter NO.42#4 and NO.42#5. And Q is a positive value, if Q is specified with a negative value, the sign is ignored. The hole bottom offset of Q is a modal value saved in canned cycle which should be specified carefully as it is also used for the cutting depth for G73 and G83.

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

If G76 and M code are specified in a same block, M code is executed while the 1<sup>st</sup> hole

positioning operation is being performed, then the system goes on next boring operation.

If number of repeats K is specified, M code is only executed for the 1st hole, not for the following holes.

**Note:** In the current version, M00, M01, M02, M06, M30, M98 and M99 are the M codes executed after the other instructions in a block, i.e. these M codes are executed after the execution of the current statement block.

**Tool length compensation:** If the tool length compensation instruction G43, G44 or G49 is specified in the same block with the canned cycle instruction, the offset is added or cancelled at the time of positioning to point R level; If the tool compensation instruction G43, G44 or G49 is specified in a separate block in the canned cycle mode, the system can add or cancel the offset in real time.

Axis switching: before the boring axis is changed, the canned cycle must be cancelled.

Boring: In a block that does not contain X , Y , Z, R or any additional axes, boring is not performed.

#### Example

M3 S500	Spindle running start
G90 G99 G76 X300.Y-250.	Positioning, bore hole 1, then return to point R level
Z-150. R-100.Q5.	Orient at the hole bottom, then shift by 5mm
P1000 F120.;	Stop at the hole bottom for 1s
Y-550.;	Positioning, bore hole 2, then return to point R level
Y-750.;	Positioning, bore hole 3, then return to point R level
X1000.;	Positioning, bore hole 4, then return to point R level
Y-550.;	Positioning, bore hole 5, then return to point R level
G98 Y-750.;	Positioning, bore hole 6, then return to initial level
G80 G28 G91 X0 Y0 Z0 ;	Return to reference point
M5;	Spindle stops

#### Restriction:

**Cancellation:** G codes of 01 group (G00 to G03), G60 modal G code (bit parameter NO: 48#0 is set to 1) and G76 cannot be specified in a same block, otherwise G76 will be cancelled.

**Tool offset:** The tool radius offset during positioning in canned cycle is ignored.

#### 4.4.14 Boring cycle G85

**Format:** G85 X\_ Y\_ Z\_ R\_ F\_ K\_

**Function:** It is used to bore a hole.

#### Explanation:

X\_ Y\_: Hole positioning data

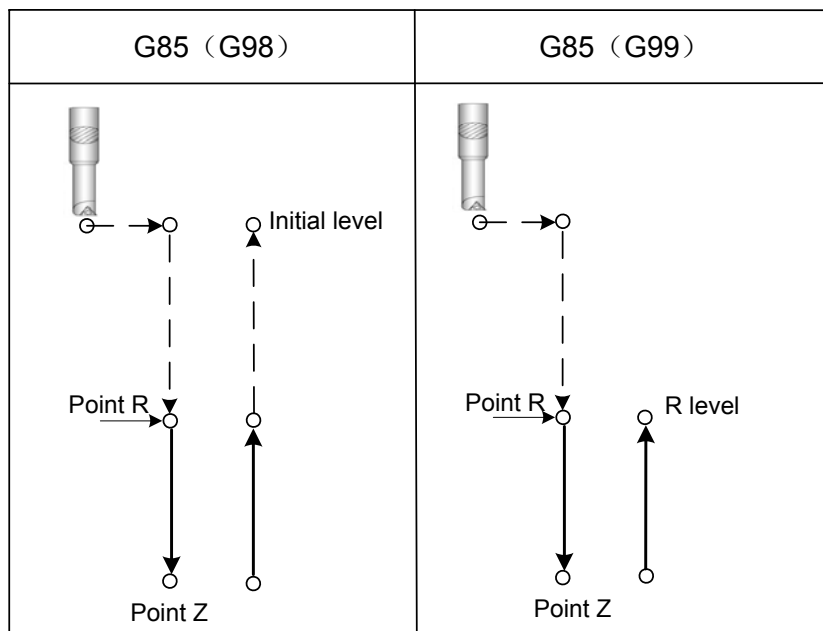
Z\_: In incremental programming it specifies the distance from point R level to the bottom of the hole; in absolute programming it specifies the absolute coordinate of the hole bottom.

R\_: In incremental programming it specifies the distance from the initial level to point R

level; in absolute programming it specifies the absolute coordinate of point R.

F\_: Cutting feedrate.

K\_: Number of repeats



**Fig. 4-4-14-1**

After positioning along X and Y axis, traverse is performed to point R level, and boring is performed from point R level to point Z level. As the tool reaches the hole bottom, cutting feed is performed then return to point R level.

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

If G85 and M code are specified in a same block, M code is executed while the 1<sup>st</sup> hole positioning operation is being performed, then the system goes on next boring operation.

If number of repeats K is specified, M code is only executed for the 1<sup>st</sup> hole, not for the following holes.

**Note:** In the current version, M00, M01, M02, M06, M30, M98 and M99 are the M codes executed after the other instructions in a block, i.e. these M codes are executed after the execution of the current statement block.

**Tool length compensation:** If the tool length compensation instruction G43, G44 or G49 is specified in the same block with the canned cycle instruction, the offset is added or cancelled at the time of positioning to point R level; If the tool compensation instruction G43, G44 or G49 is specified in a separate block in the canned cycle mode, the system can add or cancel the offset in real time.

Axis switching: Before the boring axis is changed, the canned cycle must be cancelled.

Boring: In a block that does not contain X , Y , Z , R or any additional axes, boring is not performed.

#### Example

M3 S100

Spindle running start

G90 G99 G85 X300. Y-250. Z-150. R-120. F120. Positioning, bore hole 1, then return to

point R level

Y-550.;	Positioning, bore hole 2, then return to point R level
Y-750.;	Positioning, bore hole 3, then return to point R level
X1000.;	Positioning, bore hole 4, then return to point R level
Y-550.;	Positioning, bore hole 5, then return to point R level
G98 Y-750.;	Positioning, bore hole 6, then return to initial level
G80;	
G28 G91 X0 Y0 Z0 ;	Return to reference point
M5;	Spindle stops
M30;	

**Restriction:**

**Cancellation:** G codes of 01 group (G00 to G03), G60 modal G code (bit parameter NO: 48#0 is set to 1) and G85 cannot be specified in a same block, otherwise G85 will be cancelled.

**Tool offset:** The tool radius offset during positioning in canned cycle is ignored.

#### 4.4.15 Boring cycle G86

**Format:** G86 X\_ Y\_ Z\_ R\_ F\_ K\_;

**Function:** It is used to perform a boring cycle.

**Explanation:**

X_Y_:	Hole positioning data
Z_:	In incremental programming it specifies the distance from point R level to the bottom of the hole; in absolute programming it specifies the absolute coordinate of the hole bottom.
R_:	In incremental programming it specifies the distance from the initial level to point R level; in absolute programming it specifies the absolute coordinate of point R.
F_:	Cutting feedrate
K_:	Number of repeats

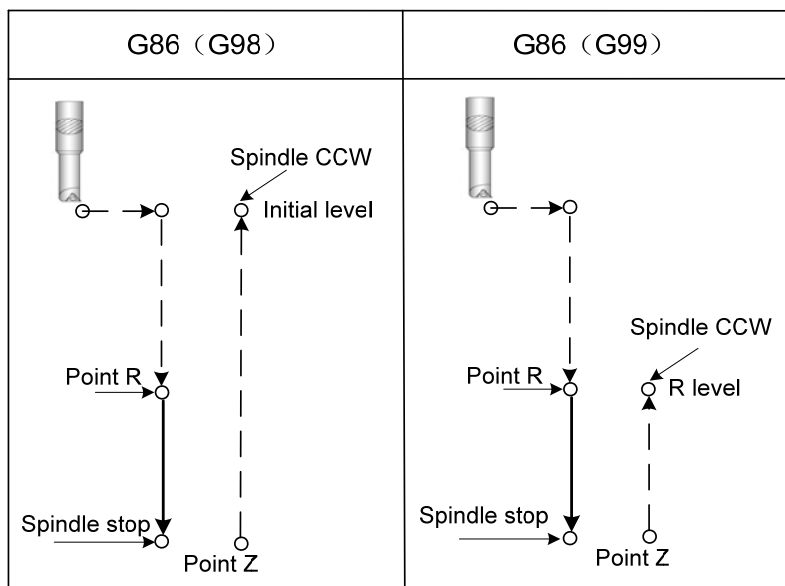


Fig. 4-4-15-1

After positioning along X and Y axis, the tool rapidly traverses to point R level. And boring is performed from point R level to point Z level. When the tool reaches the hole bottom, it is retracted in traverse.

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

If G86 and M code are specified in a same block, M code is executed while the 1<sup>st</sup> hole positioning operation is being performed, then the system goes on next boring operation.

If number of repeats K is specified, M code is only executed for the 1<sup>st</sup> hole, not for the following holes.

**Note:** In the current version, M00, M01, M02, M06, M30, M98 and M99 are the M codes executed after the other instructions in a block, i.e. these M codes are executed after the execution of the current statement block.

**Tool length compensation:** If the tool length compensation instruction G43, G44 or G49 is specified in the same block with the canned cycle instruction, the offset is added or cancelled at the time of positioning to point R level; If the tool compensation instruction G43, G44 or G49 is specified in a separate block in the canned cycle mode, the system can add or cancel the offset in real time.

Axis switching: Before the boring axis is changed, the canned cycle must be cancelled.

Boring: In a block that does not contain X, Y, Z, R or any additional axes, boring is not performed.

#### Example

```

M3 S2000           Spindle running start
G90 G99 G86 X300. Y-250. Z-150. R-100. F120.  Positioning, bore hole 1, then return to
                                                    point R level
Y-550.;           Positioning, bore hole 2, then return to point R level
Y-750.;           Positioning, bore hole 3, then return to point R level
X1000.;           Positioning, bore hole 4, then return to point R level
Y-550.;           Positioning, bore hole 5, then return to point R level
G98 Y-750;        Positioning, bore hole 6, then return to initial level
G80;
G28 G91 X0 Y0 Z0 ; Return to reference point
  
```

M5; Spindle stops

M30;

**Restriction:**

**Cancellation:** G codes of 01 group (G00 to G03), G60 modal G code (bit parameter NO: 48#0 is set to 1) and G86 cannot be specified in a same block, otherwise G86 will be cancelled.

**Tool offset:** The tool radius offset during positioning in canned cycle is ignored.

#### 4.4.16 Boring cycle, back boring cycle G87

**Format:** G87 X\_Y\_Z\_R\_Q\_F\_

**Function:** It is used for accurate boring.

**Explanation:**

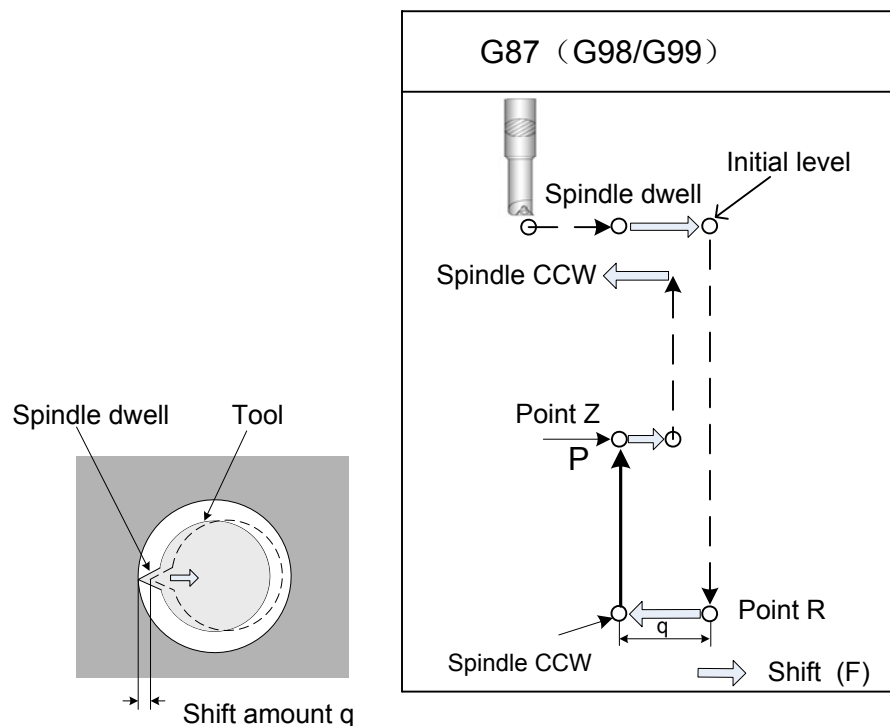
X\_Y\_: Hole positioning data

Z\_: In incremental programming it specifies the distance from point R level to point Z level; in absolute programming it specifies the absolute coordinate of the point Z level.

R\_: In incremental programming it specifies the distance from the initial level to point R level; in absolute programming it specifies the absolute coordinate of point R level . (hole bottom)

Q\_: Offset of the hole bottom

F\_: Cutting feedrate



**Fig. 4-4-16-1**

After positioning along X and Y axis, the tool is stopped after spindle orientation. And the tool is

moved in the direction opposite to the tool tip, positioning is performed at the hole bottom point R level. Then the tool is moved in the tool tip direction and the spindle is rotated clockwise. Boring is performed in the positive direction along Z axis until point Z is reached. At point Z, the spindle is stopped at the fixed rotation position after it is oriented again. And the tool is retracted to the initial level in the opposite direction of the tool tip and then is shifted in the direction of the tool tip. And the spindle is rotated clockwise to proceed to the next block operation.

The parameter Q specifies the retraction distance and the retraction direction is set by system parameter NO.42#4 and NO.42#5. Q must be a positive value, if Q is specified with a negative value, the sign is ignored. The hole bottom offset of Q is a modal value saved in canned cycle which should be specified carefully as it is also used for the cutting depth for G73 and G83.

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

If G87 and M code are specified in a same block, M code is executed while the 1<sup>st</sup> hole positioning operation is being performed, then the system goes on next boring operation.

If number of repeats K is specified, M code is only executed for the 1<sup>st</sup> hole, not for the following holes.

**Note:** In the current version, M00, M01, M02, M06, M30, M98 and M99 are the M codes executed after the other instructions in a block, i.e. these M codes are executed after the execution of the current statement block.

**Tool length compensation:** If the tool length compensation instruction G43, G44 or G49 is specified in the same block with the canned cycle instruction, the offset is added or cancelled at the time of positioning to point R level; If the tool compensation instruction G43, G44 or G49 is specified in a separate block in the canned cycle mode, the system can add or cancel the offset in real time.

The canned cycle can only be executed in G17 plane.

**Boring:** In a block that does not contain X, Y, Z, R or any additional axes, boring is not performed.

**Annotation:** The value of Z and R must be specified in the back boring cycle programming. Alarm occurs if point Z is below point R.

#### Example

```
M3 S500           Spindle running start
G90 G99 G87 X300. Y-250. Z-120. R-150. Q5. P1000 F120.
```

Positioning, bore hole 1, orient at the initial level then shift by 5mm and dwell at point Z for 1s

```
Y-550.;           Positioning, bore hole 2, then return to point R level
Y-750.;           Positioning, bore hole 3, then return to point R level
X1000.;           Positioning, bore hole 4, then return to point R level
Y-550.;           Positioning, bore hole 5, then return to point R level
G98 Y-750.;       Positioning, bore hole 6, then return to initial level
G80 G28 G91 X0 Y0 Z0 ; Return to reference point
M5;              Spindle stops
```

#### Restriction:

**Cancellation:** G codes of 01 group (G00 to G03), G60 modal G code (bit parameter NO: 48#0 is set to 1) and G87 cannot be specified in a same block,



otherwise G87 will be cancelled.

**Tool offset:** The tool radius offset during positioning in canned cycle is ignored.

#### 4.4.17 Boring cycle G88

**Format:** G88 X\_Y\_Z\_R\_P\_F\_

**Function:** It is used to bore a hole.

**Explanation:**

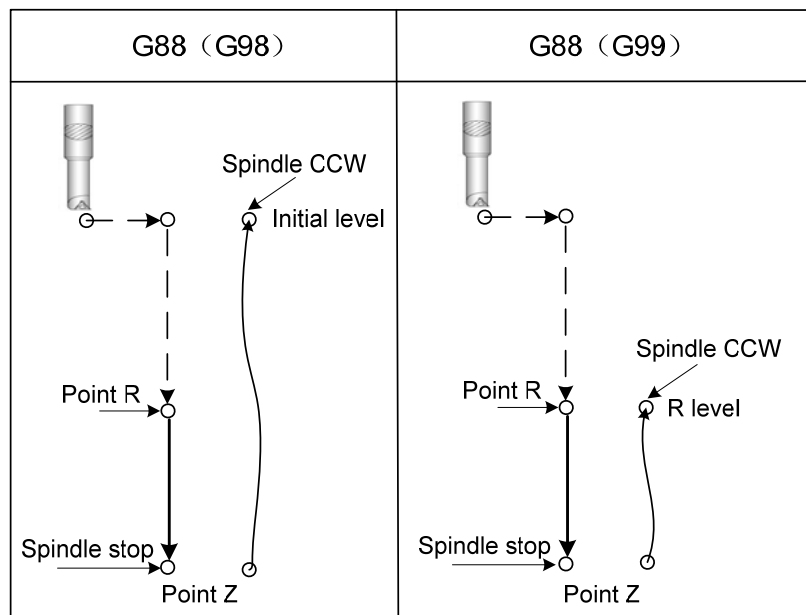
X\_Y\_: Hole positioning data

Z\_: In incremental programming it specifies the distance from point R level to the bottom of the hole; in absolute programming it specifies the absolute coordinate of the hole bottom.

R\_: In incremental programming it specifies the distance from the initial level to point R level; in absolute programming it specifies the absolute coordinate of point R.

P\_: Dwell time.

F\_: Cutting feedrate



**Fig. 4-4-17-1**

After positioning along X and Y axis, the tool rapidly traverses to point R level. Boring is performed from point R level to point Z. When boring is completed, a dwell is performed then the spindle is stopped. The tool is manually retracted from the hole bottom point Z to point R level(in G99) or the initial level(in G98) and the spindle is rotated CCW.

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

If G88 and M code are specified in a same block, M code is executed while the 1<sup>st</sup> hole positioning operation is being performed, then the system goes on next boring operation.

If number of repeats K is specified, M code is only executed for the 1<sup>st</sup> hole, not for the following

holes.

**Note:** In the current version, M00, M01, M02, M06, M30, M98 and M99 are the M codes executed after the other instructions in a block, i.e. these M codes are executed after the execution of the current statement block.

P is a modal instruction, with its min. value set by data parameter P281 and max. value by P282. If P value is less than the value set by P281, the min. value takes effect; if P value is more than the value set by P282, the max. value takes effect. P cannot be stored as modal data if it is specified in a block that does not perform drilling.

Tool length compensation: If the tool length compensation instruction G43, G44 or G49 is specified in the same block with the canned cycle instruction, the offset is added or cancelled at the time of positioning to point R level; If the tool compensation instruction G43, G44 or G49 is specified in a separate block in the canned cycle mode, the system can add or cancel the offset in real time.

Axis switching: Before the boring axis is changed, the canned cycle must be cancelled.

Boring: In a block that does not contain X, Y, Z, R or any additional axes, boring is not performed.

### Example

M3 S2000	Spindle running start
G90 G99 G88 X300. Y-250. Z-150. R-100. P1000 F120.	Positioning, bore hole 1, then return to point R level
Y-550.;	Positioning, bore hole 2, then return to point R level
Y-750.;	Positioning, bore hole 3, then return to point R level
X1000.;	Positioning, bore hole 4, then return to point R level
Y-550.;	Positioning, bore hole 5, then return to point R level
G98 Y-750.;	Positioning, bore hole 6, then return to initial level
G80 G28 G91 X0 Y0 Z0 ;	Return to reference point
M5;	Spindle stops

### Restriction:

**Cancellation:** G codes of 01 group (G00 to G03), G60 modal G code (bit parameter NO: 48#0 is set to 1) and G88 cannot be specified in a same block, otherwise G88 will be cancelled.

**Tool offset:** The tool radius offset during positioning in canned cycle is ignored.

### 4.4.18 Boring cycle G89

**Format:** G89 X\_ Y\_ Z\_ R\_ P\_ F\_ K\_

**Function:** It is used to bore a hole.

### Explanation:

X\_ Y\_: Hole positioning data

Z\_: In incremental programming it specifies the distance from point R level to the bottom of

the hole; in absolute programming it specifies the absolute coordinate of the hole bottom.

R\_: In incremental programming it specifies the distance from the initial level to point R level; in absolute programming it specifies the absolute coordinate of point R.

P\_: Dwell time

F\_: Cutting feedrate.

K\_: Number of repeats

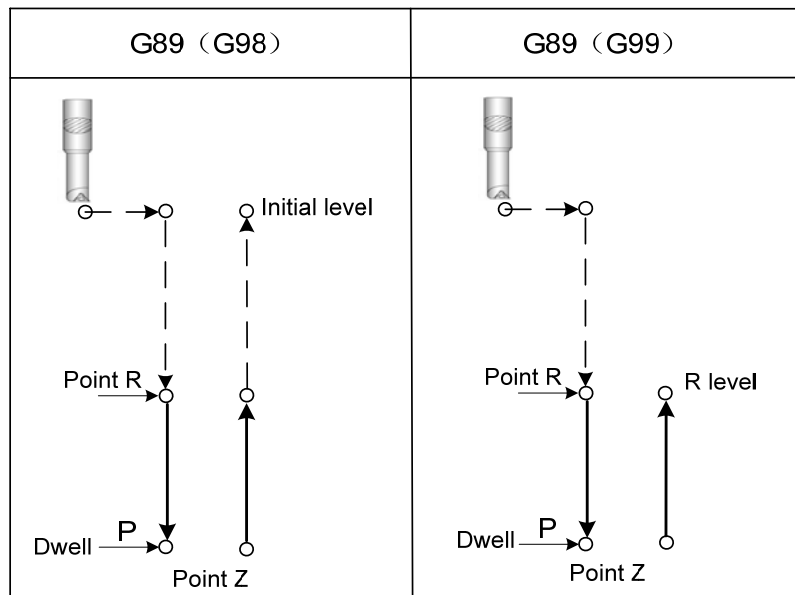


Fig. 4-4-18-1

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

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

If G89 and M code are specified in a same block, M code is executed while the 1<sup>st</sup> hole positioning operation is being performed, then the system goes on next drilling operation.

If number of repeats K is specified, M code is only executed for the 1<sup>st</sup> hole, not for the following holes.

**Note:** In the current version, M00, M01, M02, M06, M30, M98 and M99 are the M codes executed after the other instructions in a block, i.e. these M codes are executed after the execution of the current statement block.

P is a modal instruction, and the min. value of it is set by number parameter P281, the max. value by P282. If P value is less than the setting by P281, the min. value is used; if P value is more than the setting by P282, the max. value is used. If P is specified in a block containing no drilling, it can't be stored as a modal datum.

**Tool length compensation:** If the tool length compensation instruction G43, G44 or G49 is specified in the same block with the canned cycle instruction, the offset is added or cancelled at the time of positioning to point R level; If the tool compensation instruction G43, G44 or G49 is specified in a separate block in the canned cycle mode, the system can add or cancel the offset in real time.

**Axis switching:** Before the boring axis is changed, the canned cycle must be cancelled.

Boring: In a block that does not contain X, Y, Z, R or any additional axes, boring is not performed.

### Example

```

M3 S100           Spindle running start
G90 G99 G89 X300. Y-250. Z-150. R-120. P1000 F120.
Positioning, bore hole 1 with 1s dwell at the hole bottom, then return to point R level
Y-550.;           Positioning, bore hole 2, then return to point R level
Y-750.;           Positioning, bore hole 3, then return to point R level
X1000.;           Positioning, bore hole 4, then return to point R level
Y-550.;           Positioning, bore hole 5, then return to point R level
G98 Y-750.;       Positioning, bore hole 6, then return to initial level
G80;
G28 G91 X0 Y0 Z0 ; Return to Reference point
M5;               Spindle stops
M30;

```

### Restriction:

**Cancellation:** G codes of 01 group (G00 to G03), G60 modal G code (bit parameter NO: 48#0 is set to 1) and G89 cannot be specified in a same block, otherwise G89 will be cancelled.

**Tool offset:** The tool radius offset during posiin canned cycle is ignored.

### 4.4.19 Left-handed rigid tapping G74

**Format:** G74 X\_Y\_Z\_R\_P\_F\_K\_

**Function:** In rigid tapping the spindle is controlled by a servo motor. This instruction can be used for left-hand high-speed and high-precision tapping.

### Explanation:

X\_Y\_: Hole positioning data

Z\_: In incremental programming it specifies the distance from point R level to the bottom of the hole; in absolute programming it specifies the absolute coordinate of the hole bottom.

R\_: In incremental programming it specifies the distance from the initial level to point R level; in absolute programming it specifies the absolute coordinate of point R.

P\_: Dwell time at hole bottom or at point R in retraction

F\_: Cutting feedrate.

K\_: Number of repeats

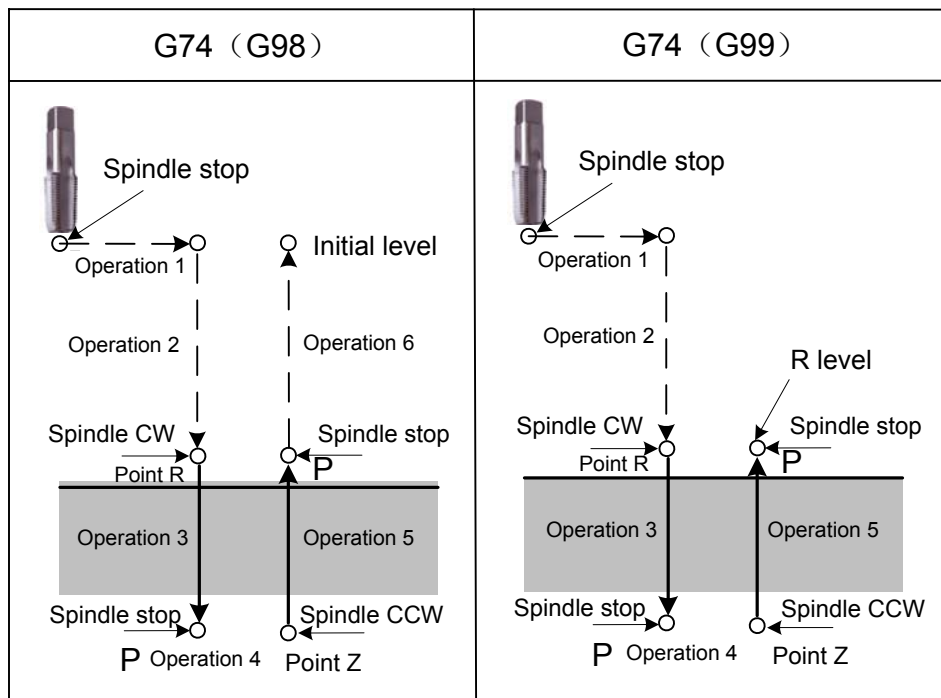


Fig. 4-4-19-1

After positioning along X and Y axis, traverse is performed by Z axis to point R level. The spindle is rotated CW for tapping from point R level to Z level by G74 instruction. When tapping is finished, the spindle is stopped and a dwell is performed. The spindle is then rotated in the reverse direction to retract to point R level and stops. And traverse to initial level is then performed. When the tapping is being performed, the feedrate override and the spindle override are assumed to be 100%.

#### Rigid mode:

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

- (1) Specify M29 S\*\*\*\*\* before a tapping instruction
- (2) Specify M29 S\*\*\*\*\* in a block that contains a tapping instruction

If G74 and M code are specified in a same block, M code is executed while the 1<sup>st</sup> hole positioning operation is being performed, then the system goes on next tapping operation.

If number of repeats K is specified, M code is only executed for the 1<sup>st</sup> hole, not for the following holes.

**Note:** In the current version, M00, M01, M02, M06, M30, M98 and M99 are the M codes executed after the other instructions in a block, i.e. these M codes are executed after the execution of the current statement block.

P is a modal instruction, and the min. value of it is set by number parameter P281, the max. value by P282. If P value is less than the setting by P281, the min. value is used; if P value is more than the setting by P282, the max. value is used. If P is specified in a block containing no drilling, it can't be stored as a modal datum.

**Tool length compensation:** If the tool length compensation instruction G43, G44 or G49 is specified in the same block with the canned cycle instruction, the offset is added or cancelled at the time of positioning to point R level; If the tool compensation instruction G43, G44 or G49 is specified in a separate block in the canned cycle mode, the system can add or cancel the offset in real time.

**Axis switching:** Before the tapping axis is changed, the canned cycle must be cancelled. Alarm occurs if the tapping axis is changed in rigid mode.

If S and axis movement instructions are specified between M29 and G74, alarm is issued. If M29 is specified in a tapping cycle, alarm is also issued.

The thread lead is obtained from the expression: feedrate/spindle speed.

Feedrate of Z axis=spindle speed × thread lead

**Example:**

Spindle speed 1000r/min

Thread lead 1.0mm

Then Feedrate of Z axis=1000×1=1000mm/min

G00 X120 Y100;                      Positioning

M29 S1000                              Rigid mode specified

G74 Z-100 R-20 F1000;              Rigid tapping

**Restriction**

**F:** Alarm is issued if the F value specified exceeds the upper limit of the cutting feedrate.

**S:** Alarm is issued if the rotation speed exceeds the max. speed of the gear used which is set by number parameter P294~296.

**Cancellation:** G codes in 01 group (G00 to G03), G60 modal G code (bit parameter NO: 48#0 is set to 1) and G74 cannot be specified in a same block, otherwise G74 will be cancelled.

**Tool offset:** The tool radius offset during positioning in canned cycle is ignored.

**Program restart:** It is ineffective during the rigid tapping.

#### 4.4.20 Right-handed rigid tapping G84

**Format:** G84 X\_Y\_Z\_R\_P\_F\_K\_

**Function:** In rigid tapping, the spindle is controlled by a servo motor that can perform the high-speed and high-precision tapping and it can ensure the tapping initial level without changing point R level. I.e. If a tapping instruction is repeated for many times at the same position, the thread shape will not be damaged.

**Explanation:**

X\_Y\_: Hole positioning data

Z\_: In incremental programming it specifies the distance from point R level to the bottom of the hole; in absolute programming it specifies the absolute coordinate of the hole bottom.

R\_: In incremental programming it specifies the distance from the initial level to point R level; in absolute programming it specifies the absolute coordinate of point R.

P\_: Dwell time at hole bottom or at point R in retraction

F\_: Cutting feedrate

K\_: Number of repeats

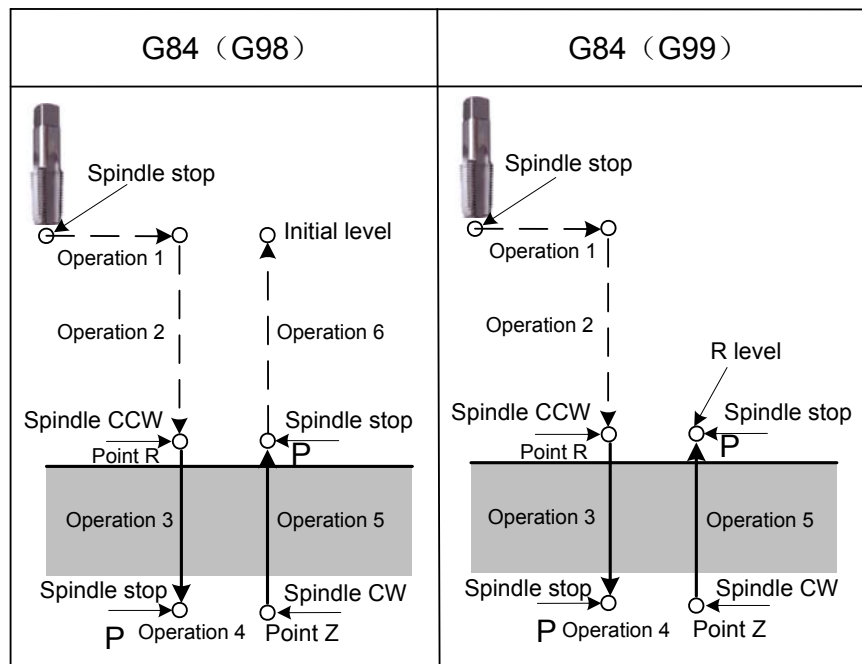


Fig. 4-4-20-1

After positioning along X and Y axis, the Z axis rapidly traverses to point R level. The spindle is rotated CCW for tapping from point R level to Z level by G84 instruction. When tapping is finished, the spindle is stopped and a dwell is performed. The spindle is then rotated in the reverse direction, the tool is retracted to point R level, then the spindle is stopped. And traverse to initial level is then performed.

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

#### Rigid mode:

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

- (1) Specify M29 S\*\*\*\*\* before a tapping instruction
- (2) Specify M29 S\*\*\*\*\* in a block that contains a tapping instruction

If G84 and M code are specified in a same block, M code is executed while the 1<sup>st</sup> hole positioning operation is being performed, then the system goes on next tapping operation.

If number of repeats K is specified, M code is only executed for the 1<sup>st</sup> hole, not for the following holes.

**Note:** In the current version, M00, M01, M02, M06, M30, M98 and M99 are the M codes executed after the other instructions in a block, i.e. these M codes are executed after the execution of the current statement block.

P is a modal instruction, and the min. value of it is set by number parameter P281, the max. value by P282. If P value is less than the setting by P281, the min. value is used; if P value is more than the setting by P282, the max. value is used. If P is specified in a block containing no drilling, it can't be stored as a modal datum.

**Tool length compensation:** If the tool length compensation instruction G43, G44 or G49 is specified in the same block with the canned cycle instruction, the offset is added or cancelled at the time of positioning to point R level; If the tool compensation instruction G43, G44 or G49 is specified in a separate block in the canned cycle mode, the system can add or cancel the offset in real time.

**Axis switching:** Before the tapping axis is changed, the canned cycle must be cancelled. Alarm occurs if the tapping axis is changed in rigid mode.

In feed-per-minute mode, the thread lead is obtained from the expression:

Thread lead = feedrate/spindle speed.  
 Feedrate of Z axis = spindle speed × thread lead  
 In feed-per-revolution mode,  
 Thread lead = federate.  
 Federate of Z axis = thread lead

**Example:**

Spindle speed 1000r/min  
 Thread lead 1.0mm  
 then Feedrate of Z axis =  $1000 \times 1 = 1000$  mm/min  
 G00 X120 Y100;                      Positioning  
 M29 S1000                              Rigid mode specified  
 G84 Z-100 R-20 F1000;              Rigid tapping

**Restriction**

**F:** Alarm is issued if the F value specified exceeds the upper limit of the cutting feedrate.

**S:** Alarm is issued if the rotation speed exceeds the max. speed of the gear specified which is set by number parameter P294~296.

**Cancellation:** G codes in 01 group (G00 to G03), G60 modal G code (bit parameter

NO: 48#0 is set to 1) and G84 cannot be specified in a same block, otherwise G84 will be cancelled.

**Tool offset:** The tool radius offset during positioning in canned cycle is ignored.

**Program restart:** It is ineffective during the rigid tapping.

**4.4.21 Peck drilling rigid tapping (chip removal) cycle**

**Format:** G84 (or G74) X\_Y\_Z\_R\_P\_Q\_F\_K\_

**Function:** In deep hole rigid tapping, it is executed many times feed till reaches the bottom of the hole.

**Explanation:**

X\_Y\_: Hole positioning data  
 Z\_: In incremental programming it specifies the distance from point R level to the bottom of the hole; in absolute programming it specifies the absolute coordinate of the hole bottom.  
 R\_: In incremental programming it specifies the distance from the initial level to point R level; in absolute programming it specifies the absolute coordinate of point R.  
 Q: Cutting depth for each cutting feed.  
 P\_: Dwell time at hole bottom or at point R in retraction  
 F\_: Cutting feedrate.  
 K\_: Number of repeats



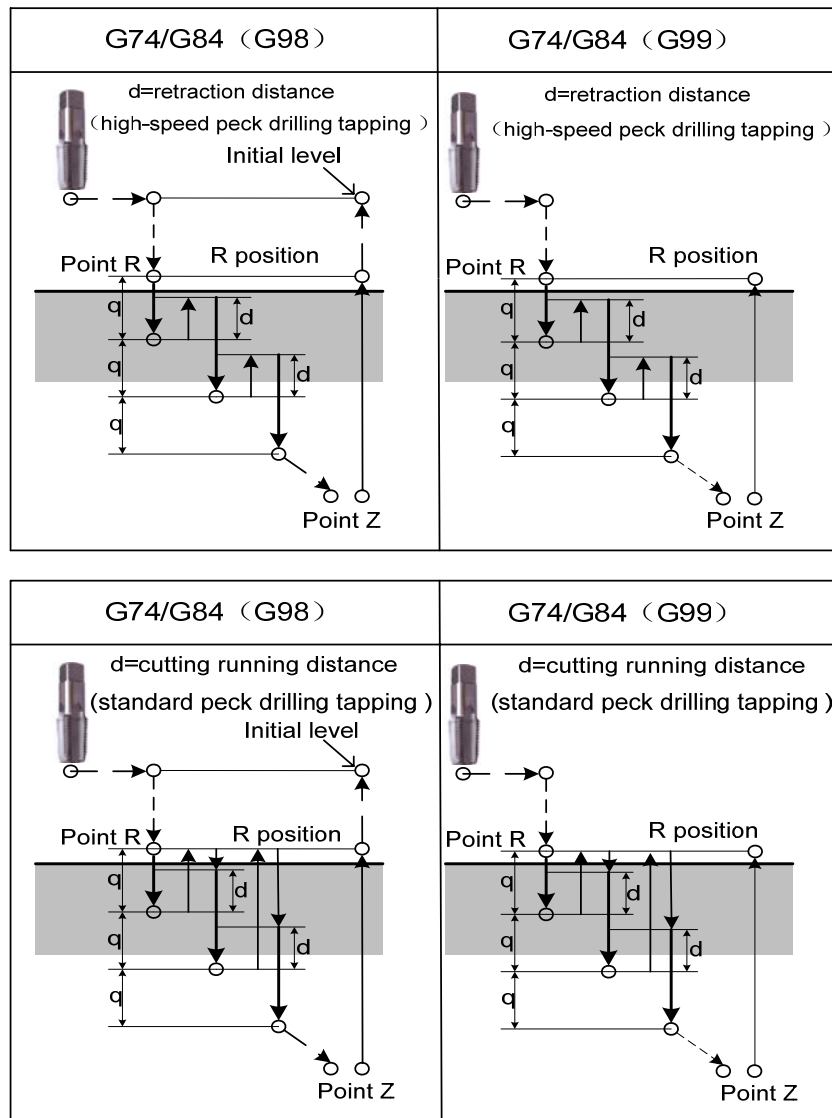


Fig. -4-21-1

There are two types for peck drilling rigid tapping cycle: high-speed peck drilling tapping cycle and standard peck drilling tapping cycle. And they are specified by bit parameter NO: 44#5.

When bit parameter **NO:44#5=1**, the mode is high-speed peck drilling tapping cycle: After positioning along X and Y axis, traverse is performed to point R level. The cutting is performed with feed depth Q (cutting depth for each feed) from point R. And then the tool retracted for a distance d (set by data parameter P270), the override in rigid tapping retraction is set by bit parameter NO:44#4, and the retract override is set by bit parameter NO:45#3. A same time constant for rigid tapping feed and retract is set by bit parameter NO:45#2, when it reaches the point Z, the spindle stops, and then rotates reversely for retraction.

When bit parameter **NO:44#5=0**, the mode is standard peck drilling tapping cycle: After positioning along X and Y axis, traverse is performed to point R level. The cutting is performed with feed depth Q (cutting depth for each feed) from point R. and then return to point R, the override in rigid tapping retract is set by bit parameter NO:44#4, and the retract override is specified by bit parameter NO:45#3, reperform the cutting with the cutting feedrate F value from point R to a point with a distance to the final point of the previous cutting. A same time constant in rigid tapping feed and retract is set by bit parameter NO:45#2, when it reaches the Z point, the spindle stops, and then rotates reversely for retraction.

**Restriction:**

F: Alarm is issued if the F value specified exceeds the upper limit of the cutting feedrate.

S: Alarm is issued if the rotation speed exceeds the max. speed of the gear used which is set by number parameter P294~296.

**Cancellation:** G codes of 01 group (G00 to G03), and G84 (or G74) cannot be specified in a same block, otherwise G84 (or G74) will be cancelled.

**Tool offset:** The tool radius offset during positioning in canned cycle is ignored.

**Program restart:** It is ineffective during the rigid tapping.

**4.4.22 Canned cycle cancel G80**

**Format:** G80

**Function:** It is used to cancel the canned cycle.

**Explanation:**

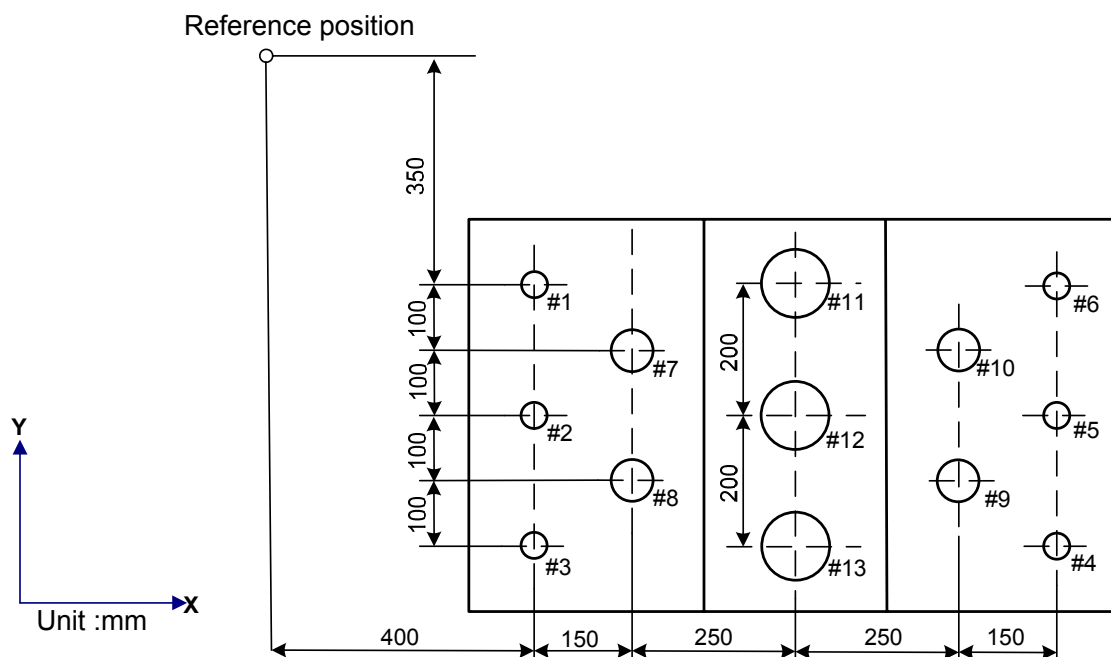
All canned cycles are cancelled for normal operation. Point R and point Z are cancelled too. Other drilling and boring datas are also cancelled.

**Example:**

M3 S100	Spindle running start
G90 G99 G88 X300. Y-250. Z-150. R-120. F120.	
	Positioning, bore hole 1, then return to point R
Y-550.;	Positioning, bore hole 2, then return to point R
Y-750.;	Positioning, bore hole 3, then return to point R
X1000.;	Positioning, bore hole 4, then return to point R
Y-550.;	Positioning, bore hole 5, then return to point R
G98 Y-750.;	Positioning, bore hole 6, then return to initial level
G80;	
G28 G91 X0 Y0 Z0 ;	Return to Reference point and cancel canned cycle
M5;	Spindle stops

**Example:**

Usage of canned cycle using tool length compensation



# 1 ~ 6...drilling  $\Phi 10$  hole

# 7 ~ 10...drilling  $\Phi 20$  hole

#11 ~ 13..boring  $\Phi 95$  hole

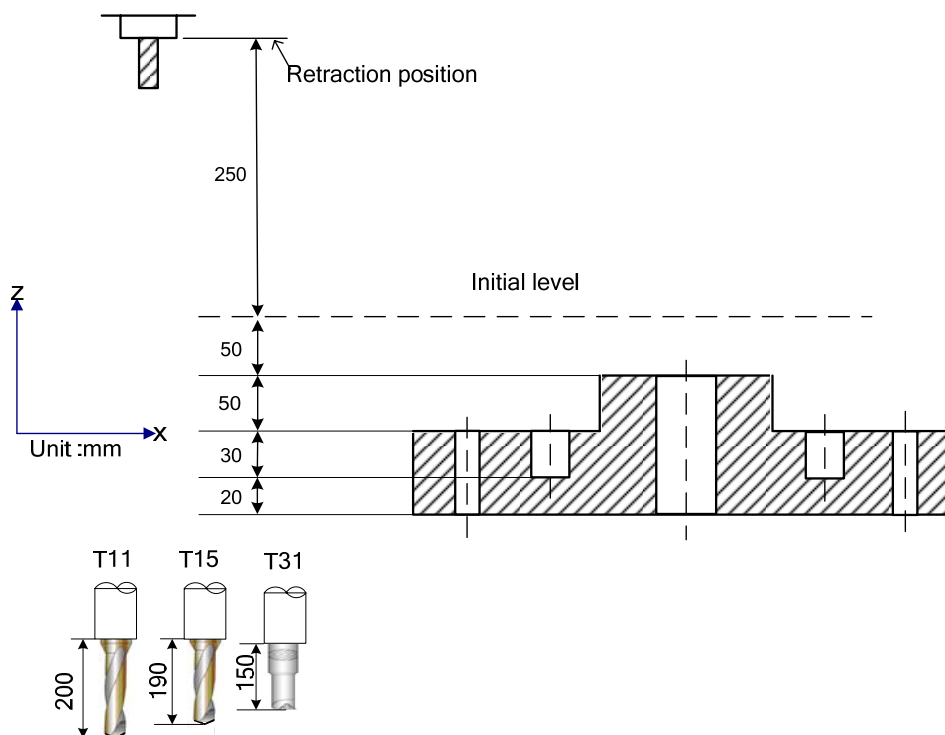


Fig. 4-4-22-1

Value 200 is set in offset No.11, 190 is set in offset No.15, 150 is set in offset No.31. The program is as following:

N001 G92 X0 Y0 Z0 ;	Coordinate setting at reference point
N002 G90 G00 Z250 T11 M6 ;	Tool change
N003 G43 Z0 H11 ;	Tool length compensation at initial level

N004 S300 M3 ;	Spindle start
N005 G99 G81 X400 Y-350 ; Z-153 R-97 F120 ;	Positioning, then #1 drilling
N006 Y-550 ;	Positioning, then #2 drilling and point R level return
N007 G98 Y-750 ;	Positioning, then #3 drilling and initial level return
N008 G99 X1200 ;	Positioning, then #4 drilling and point R level return
N009 Y-550 ;	Positioning, then #5 drilling and point R level return
N010 G98 Y-350 ;	Positioning, then #6 drilling and initial level return
N011 G00 X0 Y0 M5 ;	Reference point return, spindle stop
N012 G49 Z250 T15 M6 ;	Tool length compensation cancel, tool change
N013 G43 Z0 H15 ;	Initial level, Tool length compensation
N014 S200 M3 ;	Spindle start
N015 G99 G82 X550 Y-450 ; Z-130 R-97 P30 F70 ;	Positioning, then #7 drilling and point R level return
N016 G98 Y-650 ;	Positioning, then #8 drilling and initial level return
N017 G99 X1050 ;	Positioning, then #9 drilling and point R level return
N018 G98 Y-450 ;	Positioning, then #10 drilling and initial level return
N019 G00 X0 Y0 M5 ;	Reference point return, spindle stop
N020 G49 Z250 T31 M6 ;	Tool length compensation cancel, tool change
N021 G43 Z0 H31 ;	Initial level, Tool length compensation
N022 S100 M3 ;	Spindle start
N023 G85 G99 X800 Y-350 ; Z-153 R47 F50 ;	Positioning, then #11 drilling and point R level return
N024 G91 Y-200 ; Y-200 ;	Positioning, then #12, 13 drilling and point R level return
N025 G00 G90 X0 Y0 M5 ;	Reference point return, spindle stop
N026 G49 Z0 ;	Tool length compensation cancel
N027 M30 ;	Program stop

## 4.5 Tool compensation G code

### 4.5.1 Tool length compensation G43, G44, G49

#### Function:

G43 specifies the positive compensation for tool length.

G44 specifies the negative compensation for tool length.

G49 is used to cancel tool length compensation.

#### Format:

There are 2 modes A/B for tool length offset which are set by bit parameter No. 39.0 in this system.

Mode A:

G43     }  
G44     } Z\_ H\_ ;  
Mode B:

G17 G43 Z\_H;

G17 G44 Z\_H;

G18 G43 Y\_H;

G18 G44 Y\_H;

G19 G43 X\_H;

G19 G44 X\_H;

Tool length offset mode cancel: G49; or H0;

#### Explanation:

The instruction above is used to shift an offset value for the end point of specified axis. Due to the difference of the tool length value assumed (usually the 1<sup>st</sup> tool) and the actual tool length in machining saved in the offset memory, the tool of different lengths can be used for machining only by changing the tool length offset value, but not changing the program.

G43, G44 specify the different offset direction and H code specifies the offset number. For the tool length compensation the effectiveness of the offset value by H code respecified or in next block is set by bit parameter **NO:39#6**.

#### 1 Offset direction

G43: Positive offset (frequently -used)

G44: Negative offset

Either for absolute instruction or incremental instruction, when G43 is specified, the offset value (stored in offset memory) specified with the H code is added to the coordinate of the specified axis moving end point in the program. When G44 is specified, the offset value specified by H code is subtracted from the coordinate of the end position, and the resulting value obtained is taken as the final coordinate of the end position.

G43, G44 are modal G code, which are effective till another G code belonging to the same group is used.

#### 2 Specification of offset value

The length offset number is specified by H code, and the new moving instruction value of Z axis is obtained by plusing or subtracting the value of the offset number from the moving instruction value of Z axis. The offset number can be specified by H00~H256 as required.

The value of the offset number can be stored into the offset memory in advance by LCD/MDI panel.

The range of the offset value is as follows:

	mm input
Offset value H	-999.999mm~+999.999mm

The offset value corresponding to offset No.00 (H00) is 0. It can't be set in the system.

**Note** While the offset value is changed due to the offset number changing, the old offset value is replaced by the new one, not the adding of the new offset value and the old one.

**For example:**

H01..... offset value 20

H02..... offset value 30

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

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

### 3 Sequence of the offset value

Once the length offset mode is set up, the current offset number takes effect at once; if the offset number is changed, the old offset value will be immediately replaced by the new one. For **Example:**

Oxxxxx;

H01;

G43 Z10; (1) Offset number H01 takes effect

G44 Z20 H02; (2) Offset number H02 takes effect

H03; (3) Offset number H03 takes effect

G49; (4) Offset cancel, H00 takes effect

M30;

### 4 Tool length compensation cancel

Specify G49 or H00 to cancel tool length compensation. And the tool length compensation is cancelled immediately after they are specified.

**Note** 1. After B mode of tool length offset is executed along two or more axes, all the axes offset can be cancelled by G49, while only the axis offset perpendicular to a specified plane can be cancelled by H0.

2. when cancel the length compensation without Z axis move instruction, the machine tool will shift corresponding length compensation up or down, therefore, ensure that Z axis is in the safe height when using G49 to protect against tool collision or workpiece damage.

### 5 G53, G28 or G30 in tool length offset mode

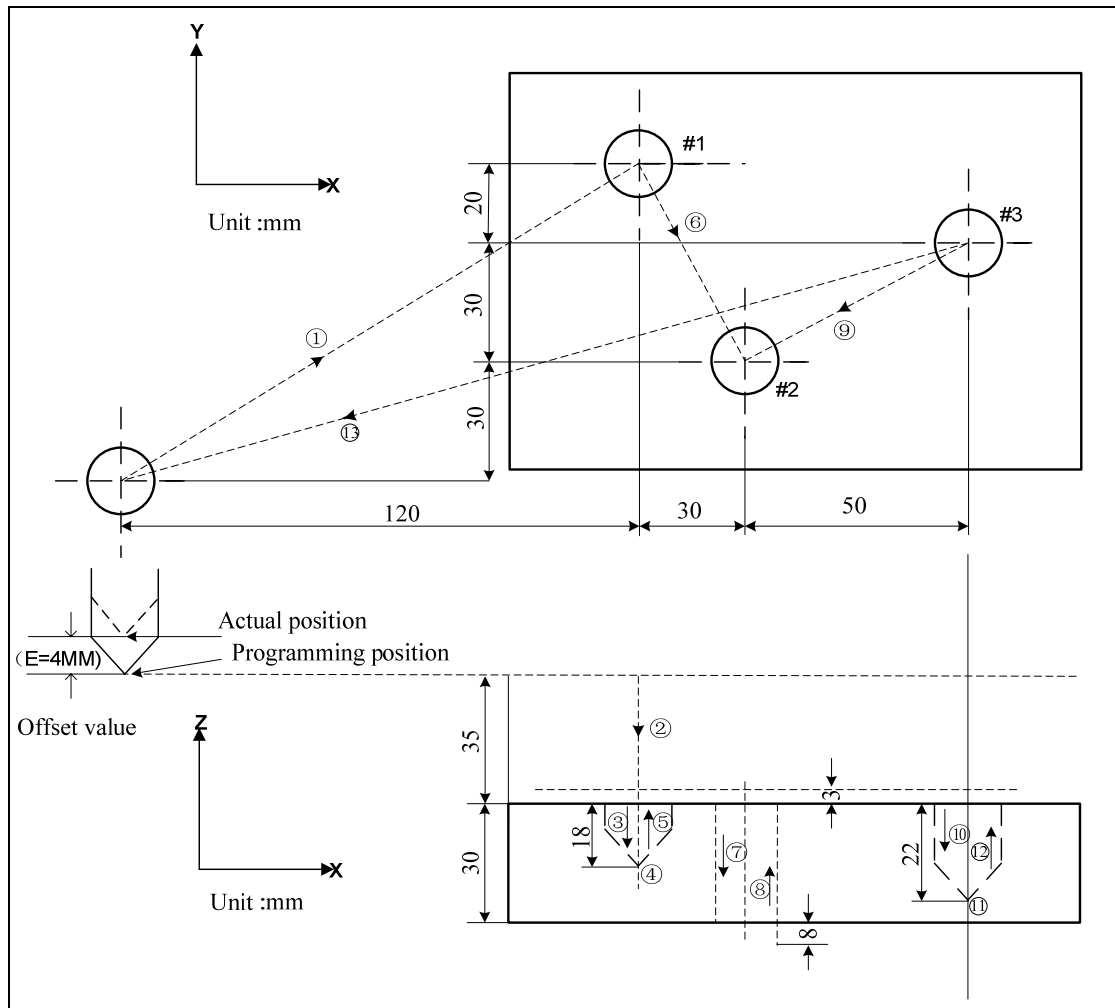
While G53, G28 or G30 is specified in the tool length offset mode, the offset vector of the tool length offset axis is cancelled after it moves to a specified position (G53 cancelled at the specified position; G28, G30 cancelled at the intermediate point), but the modal code is not switched to G49 and the axes except the tool length offset axis are not cancelled. If G53 and G49 are in the same

block, all the axis length offsets are cancelled after the axis moves to the specified position; if G28 or G30 is in the same block with G49, all the axes cancel the length offset after they move to the intermediate point. In tool length offset, the offset vector cancelled by G53, G28 or G30 will be restored in the next block in the buffer.

## 6 Example for tool length compensation

(A) Tool length compensation ( in boring hole # 1, #2, #3)

(B) H01= offset value - 4



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

N1 G91 G00 X120 Y80 ; ..... (1)  
N2 G43 Z-32 H01 ; ..... (2)  
N3 G01 Z-21 F200 ; ..... (3)  
N4 G04 P2000 ; ..... (4)  
N5 G00 Z21 ; ..... (5)  
N6 X30 Y-50 ; ..... (6)  
N7 G01 Z-41 F200 ; ..... (7)  
N8 G00 Z41 ; ..... (8)  
N9 X50 Y30 ; ..... (9)  
N10 G01 Z-25 F100 ; ..... (10)  
N11 G04 P2000 ; ..... (11)  
N12 G00 Z57 H00 ; ..... (12)

```

N13 X-200 Y-60 ; .....(13)
N14 M30 ;
N14 M30 ;

```

#### 4.5.2 Tool radius compensation G40/G41/G42

##### Format:

```

{ G41 D_X_Y_
  G42 D_X_Y_
  G40 X_Y_

```

##### Function:

G41 specifies the left offset of the tool moving.

G42 specifies the right offset of the tool moving.

G40 specifies the tool radius compensation cancel.

##### Explanation:

##### 1 Tool radius compensation

As following figure, to cut workpiece A using the tool with the radius R, the tool center path is shown as B, the distance from B to A is R, the distance that the tool deviates from the workpiece A is called compensation.

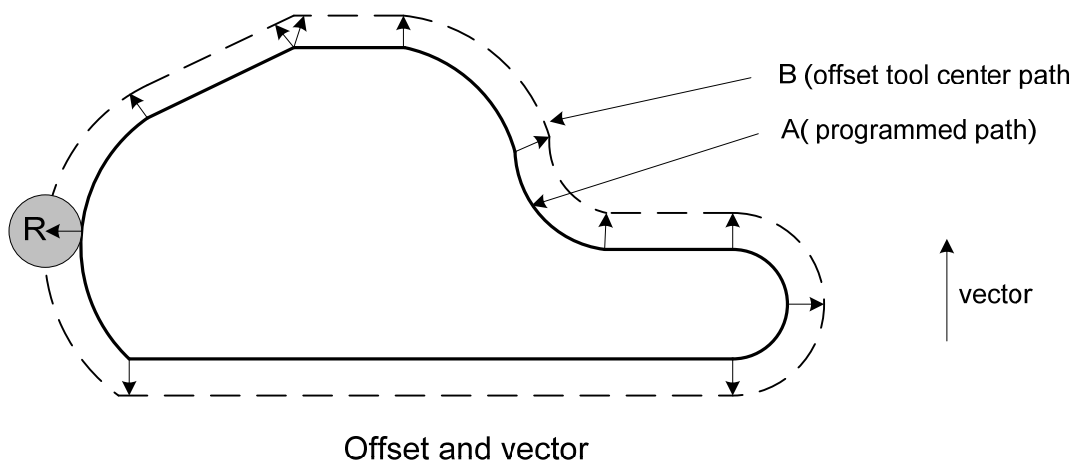


Fig. 4-5-2-1

The tool radius compensation is programmed for machining program by programmer. During the machining, the tool diameter is measured and input into the CNC memory. And the tool path turns into a offset path B.

##### 2 Offset value (D value)

The radius offset number is specified by D code, and the new moving instruction value is obtained by the value of the offset number plusing or subtracting the moving value of the program. The offset number can be specified by D00~D256 as required. The diameter or radius value of it can be set by bit parameter **N0: 40#7**.

The offset value of the offset number can be saved into the offset memory in advance by



LCD/MDI panel. For the tool radius compensation the effectiveness of the offset value by D code respecified or in next block is set by bit parameter No.39.4.

The range of the offset value is as follows:

**Table 4-5-2-1**

	mm input
Offset value D (input in mm)	-999.9999mm~999.9999mm
Offset value D (input in inch)	-99.99998 inch~99.99998 inch

**Note:** The default offset value of D00 is 0 that can't be set or modified by user.

### 3 Plane selection and vector

Compensation calculation is carried out in the plane determined by G17, G18, G19. This plane is called the compensation plane. For example, if XY plane is selected, the compensation and vector calculation are carried out by (X, Y) in program. The coordinates of the axis not in compensation plane are not affected by compensation.

In simultaneous 3 axes control, only the tool path projected on the compensation plane is compensated.

The change of the compensation plane can only be performed after the compensation is cancelled.

**Table 4-5-2-2**

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

### 4 G40, G41 and G42

The cancellation and execution of the tool radius compensation vector are specified by G40, G41, G42. They are used to define a mode to determine the value and the direction of the offset vector by combining with G00, G01, G02, G03.

**Table 4-5-2-3**

G code	Function
G40	Tool radius compensation cancel
G41	Tool radius offset left
G42	Tool radius compensation right

### 5 G53, G28, G29 or G30 in the tool radius compensation mode

When G53, G28, G29 or G30 are specified in tool radius compensation mode, the offset vector of tool radius compensation axis is cancelled after it moves to a specified position ( G53 is cancelled at the specified position, G28,G29 and G30 are cancelled at the intermediate point),

but the modal code is not switched to G40, and the axes except tool radius compensation axis are not cancelled. If G53 and G40 are in the same block, all the axes are cancelled after they move to the specified position; if G28, G29 or G30 are in a same block with G40, all the axes cancel the length compensation after they move to the intermediate point. In tool radius compensation, the radius compensation offset vector cancelled by G53, G28, G29 or G30 will be restored in the next block in the buffer. ( Note: in offset mode, if the compensation will be temporarily cancelled when specifying G28、G30 is defined by parameter **NO: 40#2** )

Note: in offset mode, whether the compensation is temporarily cancelled when G28 or G30 moves to the intermittent point is decided by bit parameter No: 40#2.

### Tool radius compensation cancel (G40)

Use G40 X\_\_ Y\_\_ instruction to cancel the tool radius compensation in G00, G01 mode.

It perform the linear motion from the old vector of the start point to the end point. In G00 mode, the axes rapidly traverse to the end point. By using this instruction, the system enters into tool radius compensation cancel mode from tool radius compensation mode

If G40 is specified without X\_\_ Y\_\_, no operation is performed by the tool.

### Tool radius compensation left (G41)

#### 1) In G00, G01 mode

G41 X\_\_ Y\_\_ D\_\_ ; It specifies a new vector being vertical to the direction of(X, Y) at the block end point. The tool is moved from the tip of the old vector to the tip of the new one at the start point.

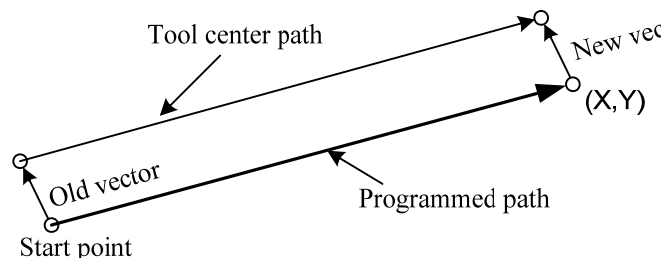


Fig. 4-5-2-2

When the old vector is zero, by this instruction the tool is switched to tool radius compensation mode from tool offset cancel mode. And the offset value is specified by D code.

#### 2) In G02, G03 mode

**G41.....;**

.....

.....

**G02 /G03 X\_\_ Y\_\_ R\_\_ ;**

By program above, the new vector that is located on the line between the circle center and the end point can be made out. From the arc advancing direction, it points to the left (or right). The tool center moves along an arc from the old vector tip to the new vector tip with the precondition that the old vector is has been made out.

The offset vector points to or is apart from the circle center from the start point or the end point.

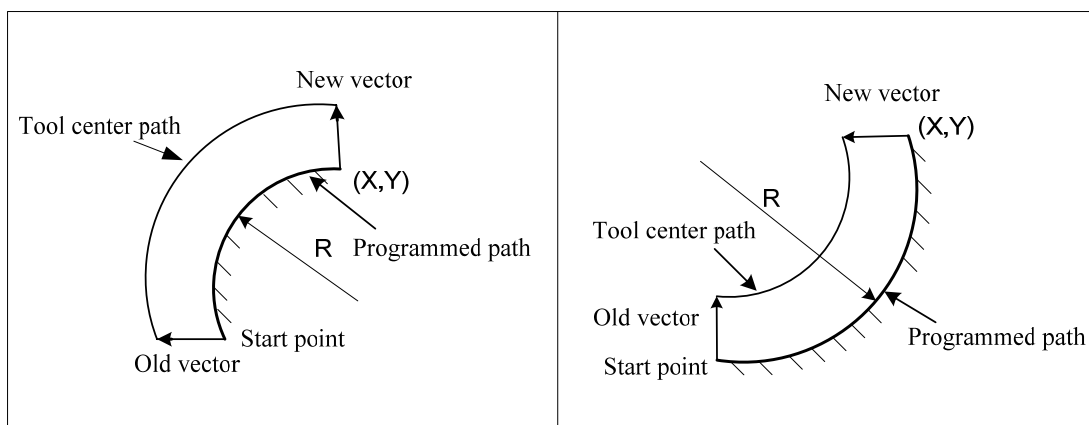


Fig. 4-5-2-3

### Tool radius compensation right (G42)

By contrast to G41, G42 specifies the tool to deviate at the right side of the workpiece along the tool advancing direction. I.e. the vector direction got in G42 is reverse to the vector direction got in G41. Besides the direction, the deviation of G42 is identical with that of G41.

#### 1) In G00, G01 mode

G42 X\_\_ Y\_\_ D\_\_ ;

G42 X\_\_ Y\_\_ ;

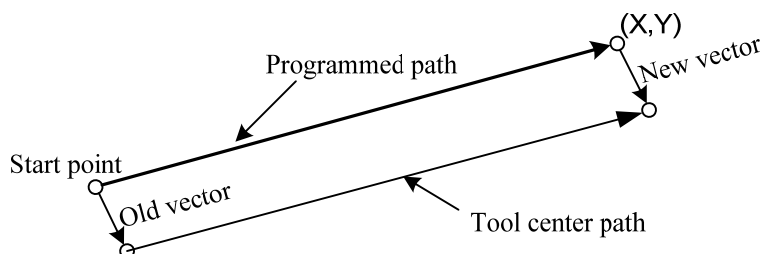


Fig. 4-5-2-4

#### 2) In G02, G03 mode

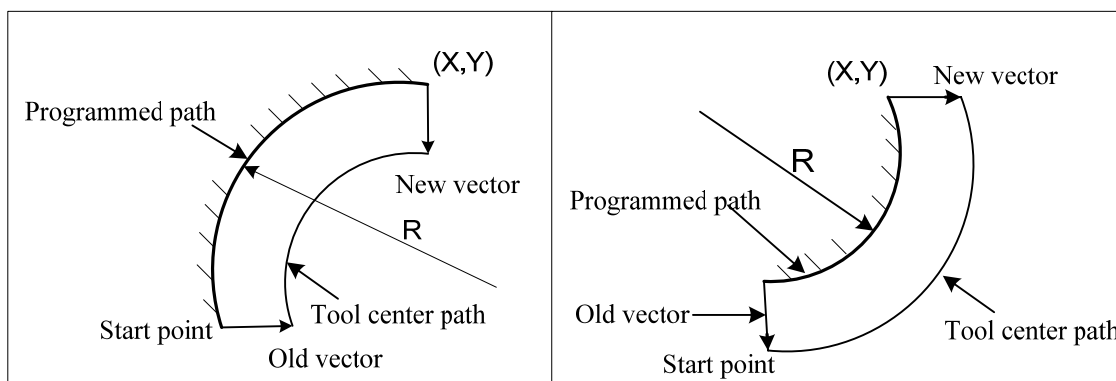


Fig. 4-5-2-5

## 6 Precautions on offset

### (A) Specification of offset number

G41, G42 and G40 are modal instructions. The offset number is specified by D code, they can be specified at any place from the offset cancel mode to tool radius compensation mode. Alarm is issued if G41, G42 instructions are not followed by moving instructions.

### (B) From the offset cancel mode to tool radius compensation mode

The moving instruction must be positioning (G00) or linear interpolation (G01) when the mode is switched from the offset cancel to tool radius compensation. And the circular interpolation (G02, G03) is impermitted.

### (C) Switching of tool radius compensation

The offset direction is usually changed from the left to the right or vice versus via offset cancel mode. But the positioning (G00) or linear interpolation (G01) can be changed directly not via offset cancel mode, and the tool path is as follows:

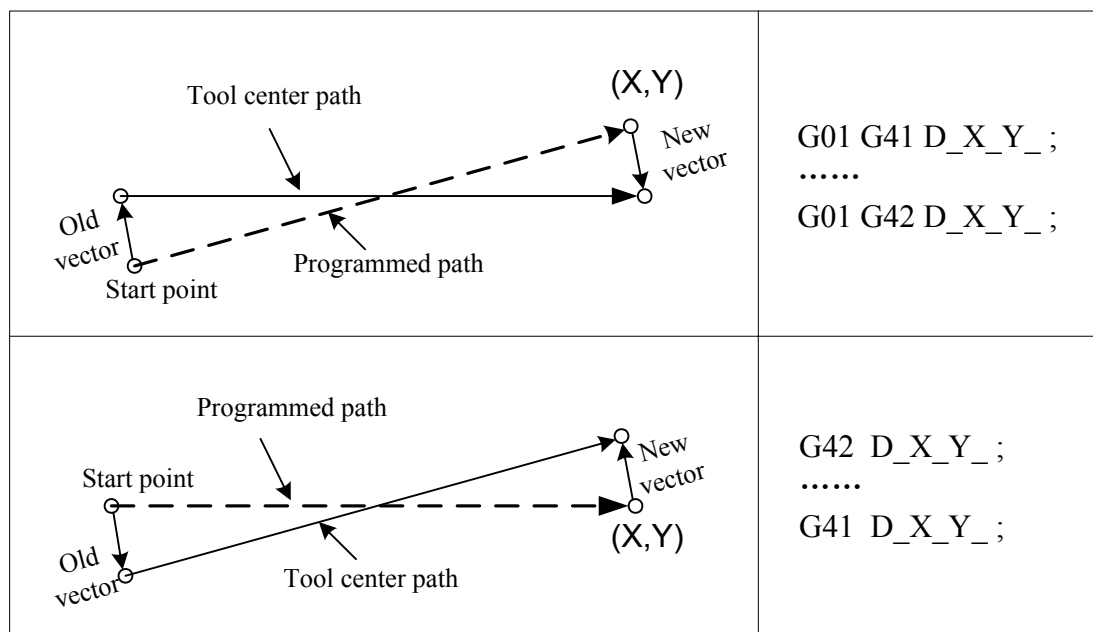


Fig. 4-5-2-6

### (D) The change of offset value

The change of offset value is usually performed at the tool change in offset cancel mode, but for the positioning (G00) or linear interpolation (G01) it can also be performed in offset mode. It is shown as follows:

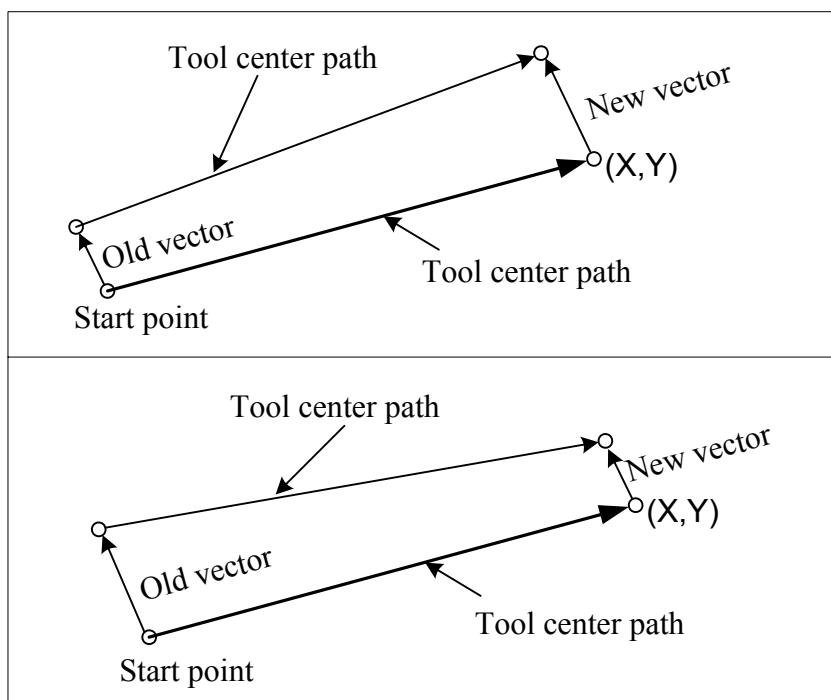


Fig. 4-5-2-7( the change of offset value)

**(E) The positive and negative offset value and the tool center path**

If the offset value is set for negative, it is equivalent to change the G41 and G42 in program that the outer cutting for workpiece turns into inner cutting, and inner cutting turns into outer cutting.

In the following programming figure, the offset value is assumed for positive:

When a tool path is programmed as(A), and the offset value is set for negative, the tool center moves as in (B); if a tool path is programmed as(B), and the offset value is set for negative, the tool center moves as in (A).

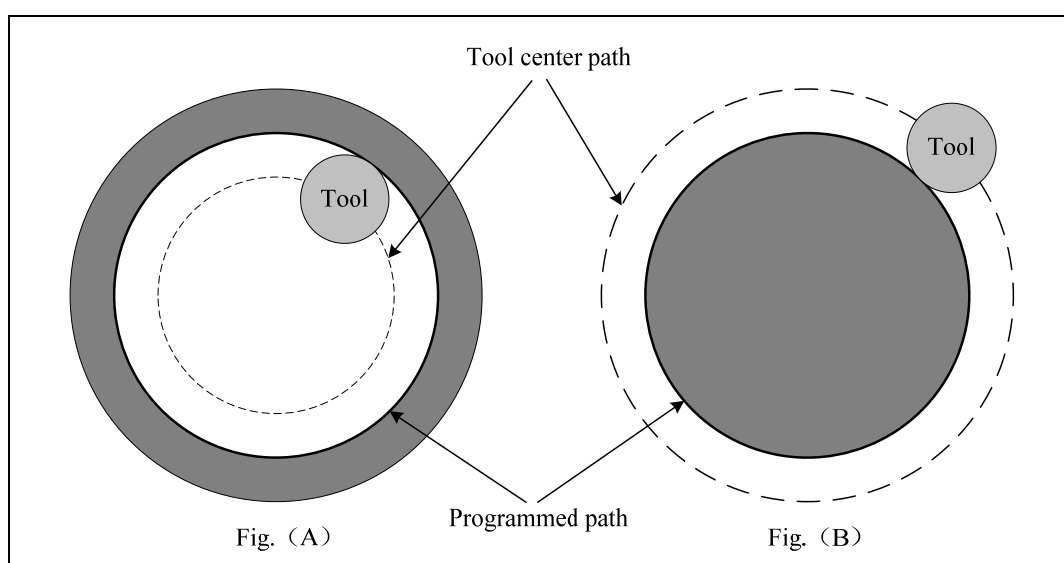


Fig. 4-5-2-8

The figure with acute angles is often used (with sharp-angle arc interpolation figure). If the offset value is set for negative, the inner side of the workpiece can't be cut. When cutting the inner sharp

angle in a point, interpolate an arc with a proper radius at the point for smooth cutting transition.

The compensation for left or right is judged by the compensation direction (workpiece unmoved) to the direction of the tool movement relative to the workpiece. By G41 or G42, the system enters compensation mode, and by G40 the compensation mode is cancelled.

The example for compensation program is as following:

The block 1, in which the compensation cancel mode is changed for compensation mode by G41 instruction, is called start. At the block end, the tool center is compensated by the tool radius that is vertical to the next block (from P1 to P2). The offset value is specified by D07, i.e. the offset number is set for 7. and G41 specifies the tool path compensation left.

During the offset, the workpiece figure is programmed as P1→P2.....P9→P10→P11, and the tool path compensation is performed automatically.

Program example for the tool path compensation:

G92 X0 Y0 Z0;

(1) N1 G90 G17 G0 G41 D7 X250 Y550 ; (The offset value must be preset by the offset number.)

(2) N2 G1 Y900 F150 ;

(3) N3 X450 ;

(4) N4 G3 X500 Y1150 R650 ;

(5) N5 G2 X900 R-250 ;

(6) N6 G3 X950 Y900 R650 ;

(7) N7 G1 X1150 ;

(8) N8 Y550 ;

(9) N9 X700 Y650 ;

(10) N10 X250 Y550 ;

(11) N11 G0 G40 X0 Y0 ;

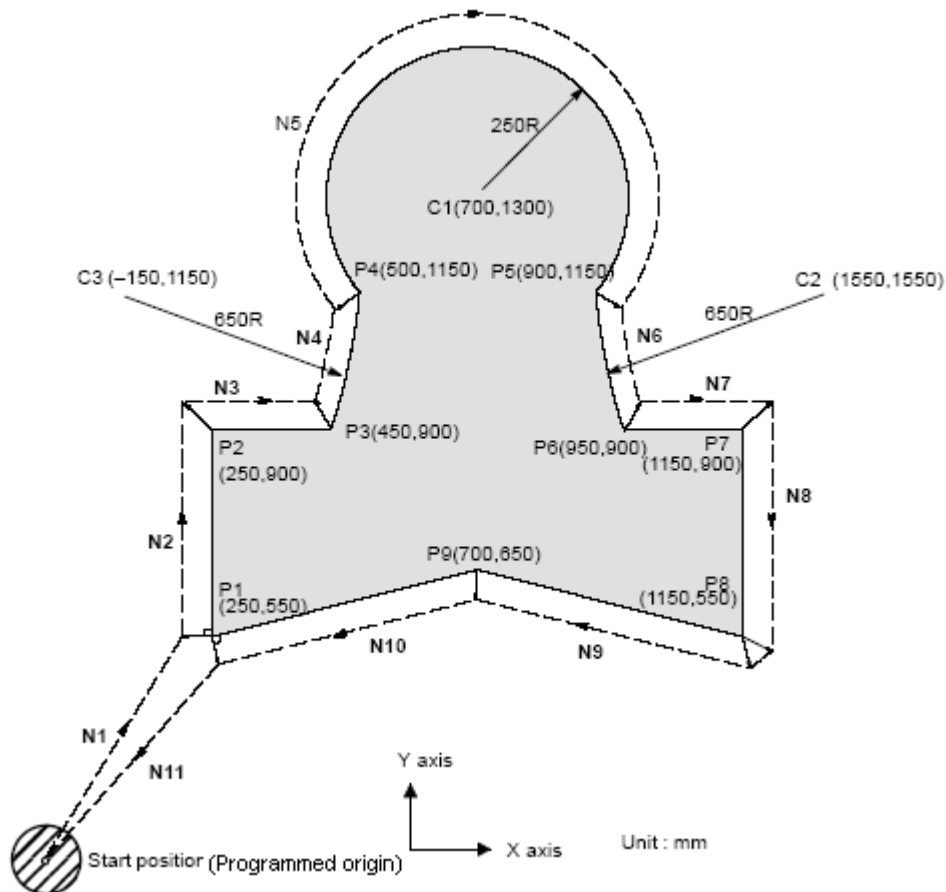


Fig. 4-5-2-9

### 4.5.3 Explanation of tool radius compensation

#### Conception:

Inner side and outer side: when an angle of intersection created by tool paths specified with move instructions for two blocks is over  $180^\circ$ , it is called inner side, when the angle is between  $0^\circ$  and  $180^\circ$ , it is called outer side.

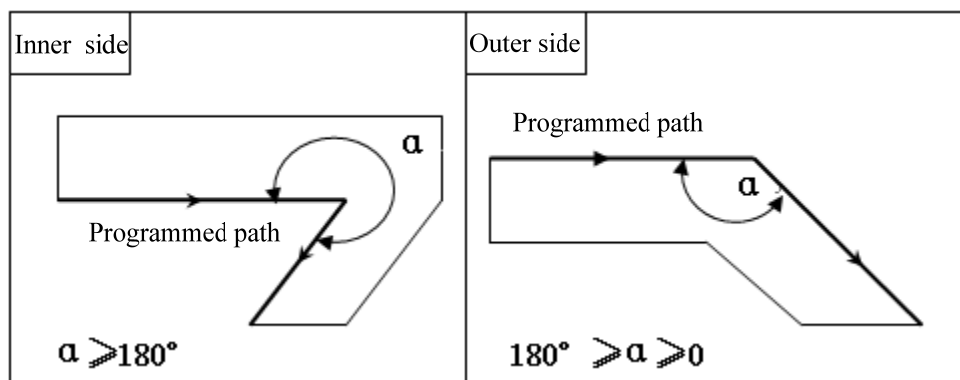


Fig. 4-5-3-1

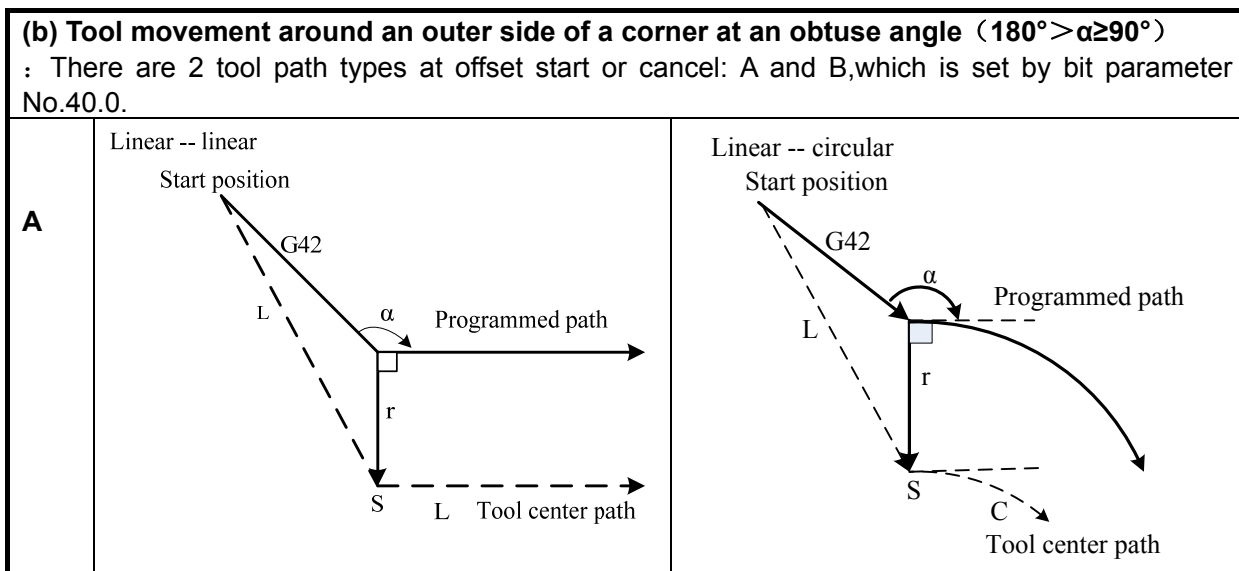
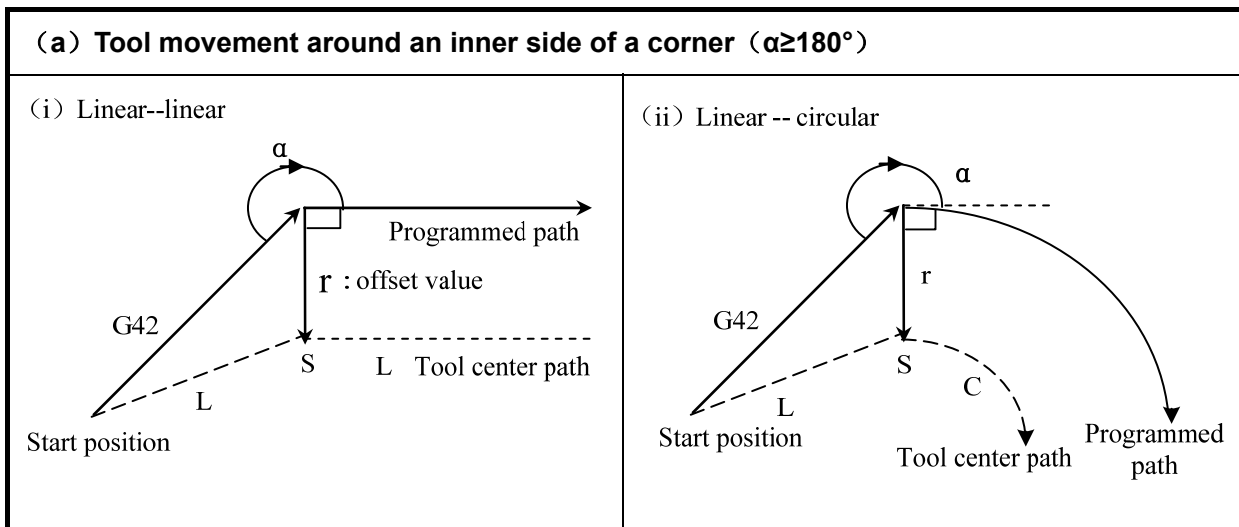
Meanings of symbols:

The following symbols are used in following figures:

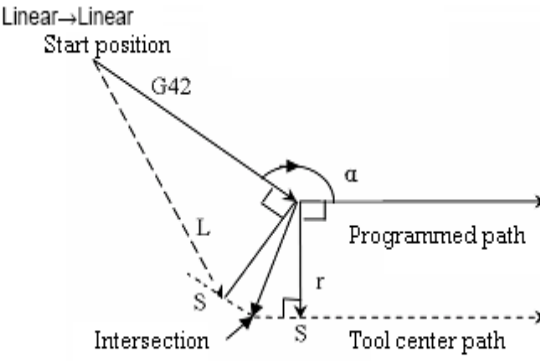
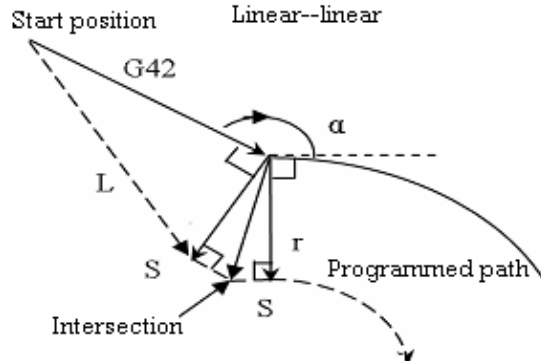
——S indicates a position at which a single block is executed once.

- SS indicates a position at which a single block is executed twice.
- SSS indicates a position at which a single block is executed three times
- L indicates that the tool moves along a straight line.
- C indicates that the tool moves along an arc.
- r indicates the tool radius compensation value.
- An intersection is a position at which the programmed paths of two blocks intersect with each other after they are shifted by r
- O indicates the center of the tool

1. Tool movement in start-up When the offset cancel mode is changed to offset mode, the tool moves as illustrated below(start-up):

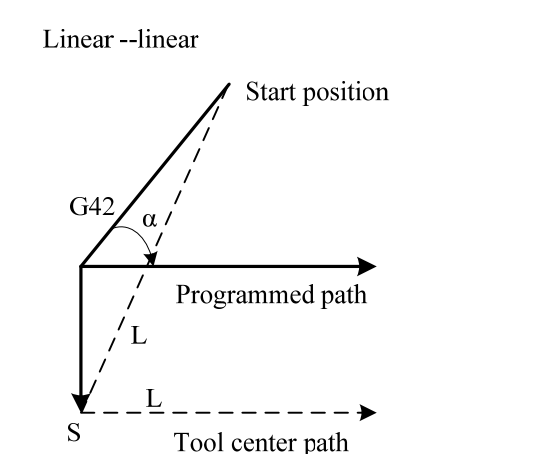
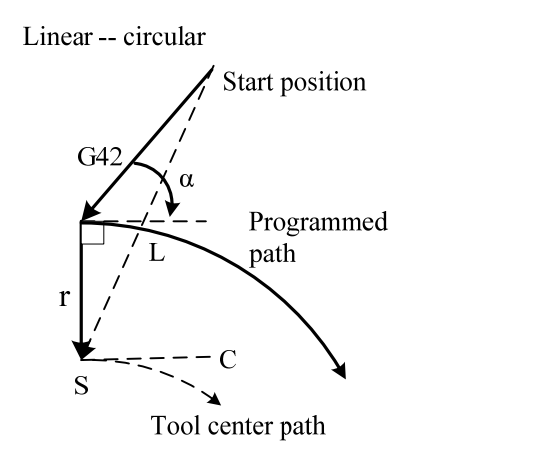
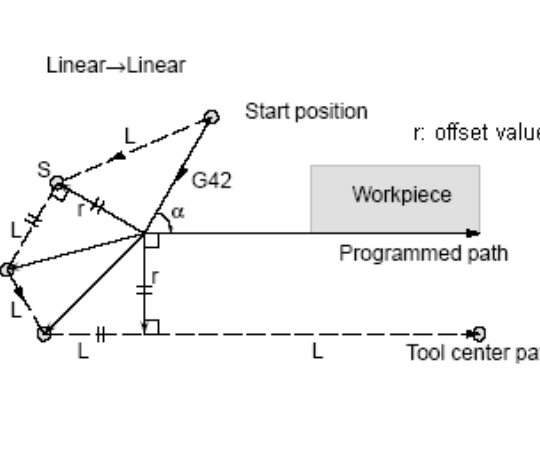
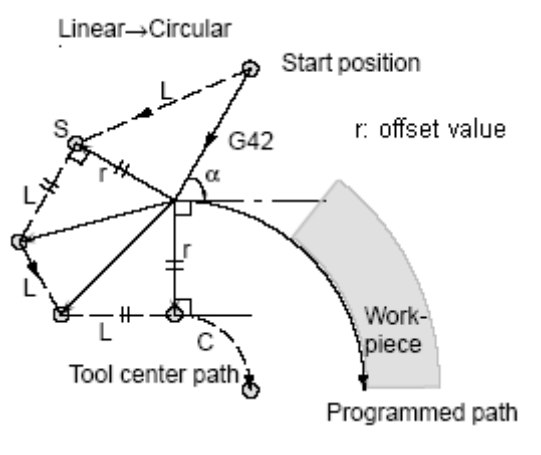




<p><b>B</b></p>	<p>Linear→Linear Start position</p>  <p>Note: intersection is the position where offset path of two successive blocks intersects</p>	<p>Linear--linear Start position</p> 
-----------------	---	---

**(C) Tool movement around an outer side of a corner at an acute angle ( $\alpha < 90^\circ$ )**

There are 2 tool path types at offset start or cancel: A and B, which is set by bit parameter No.40.0.

<p><b>A</b></p>	<p>Linear --linear</p> 	<p>Linear -- circular</p> 
<p><b>B</b></p>	<p>Linear→Linear</p> 	<p>Linear→Circular</p> 

**(d) Tool movement around an outer side of a corner at an acute angle less than  $1^\circ$  ( $\alpha < 1^\circ$ )**  
linear→linear

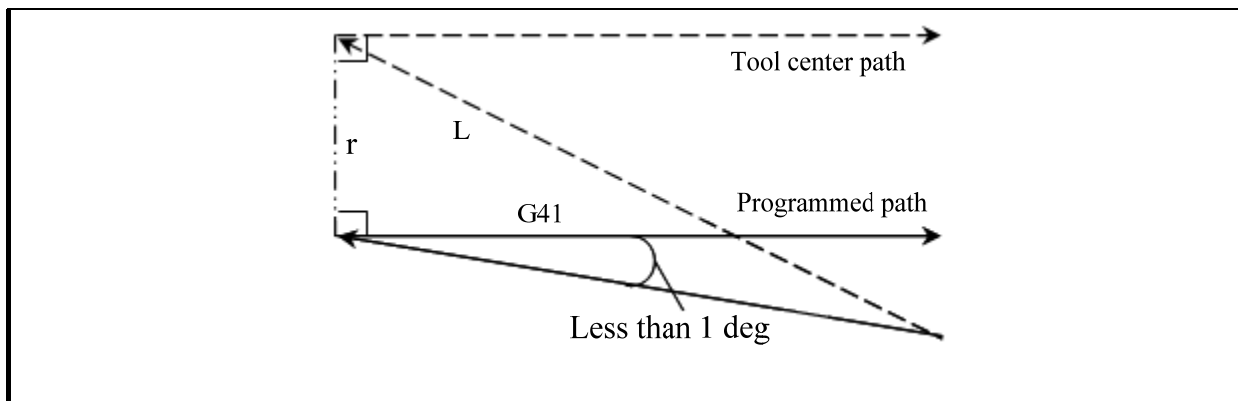


Fig. 4-5-3-2

## 2. Tool movement in offset mode

Alarm occurs and tool stops if the offset plane is changed during the offset. The tool movement in offset mode is as following figures:

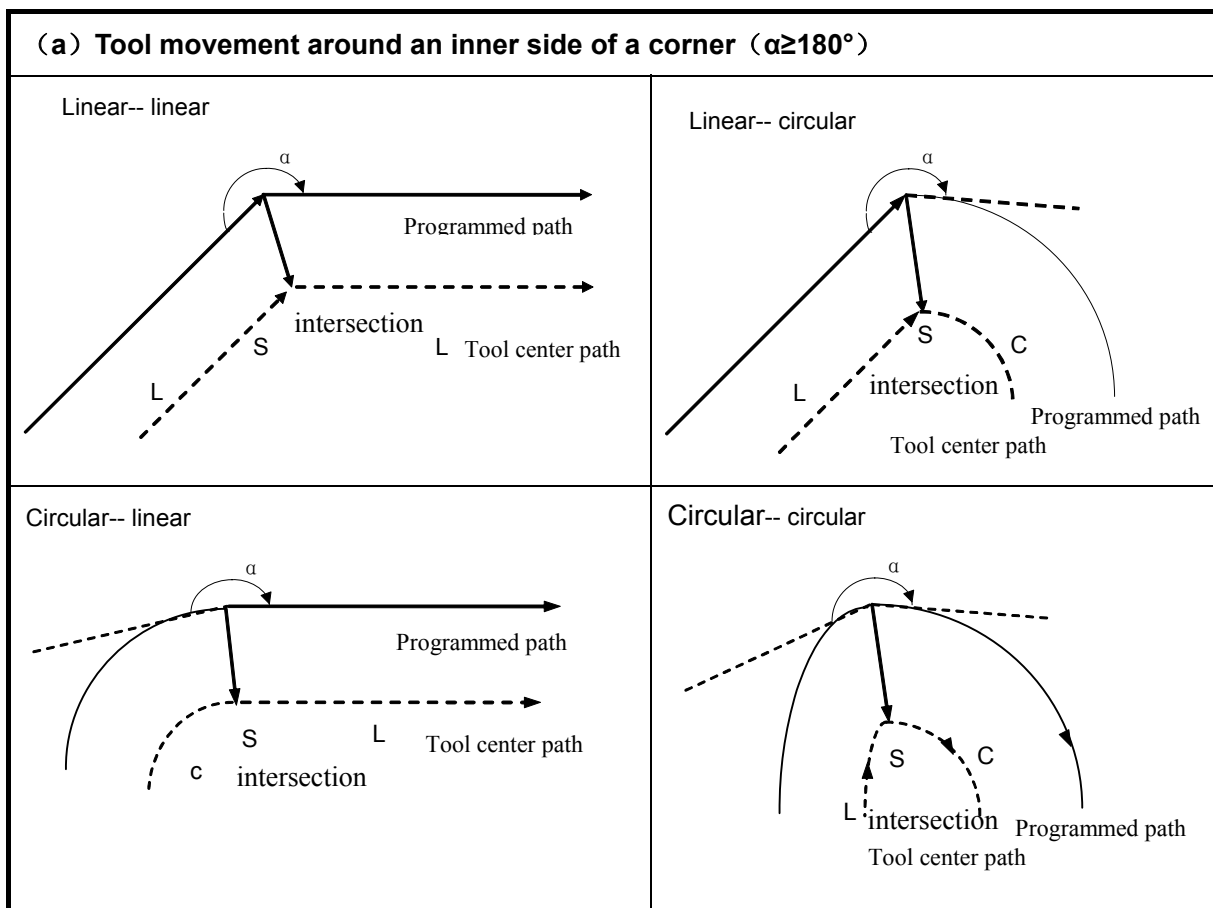


Fig. 4-5-3-3

## 3. Special condition:

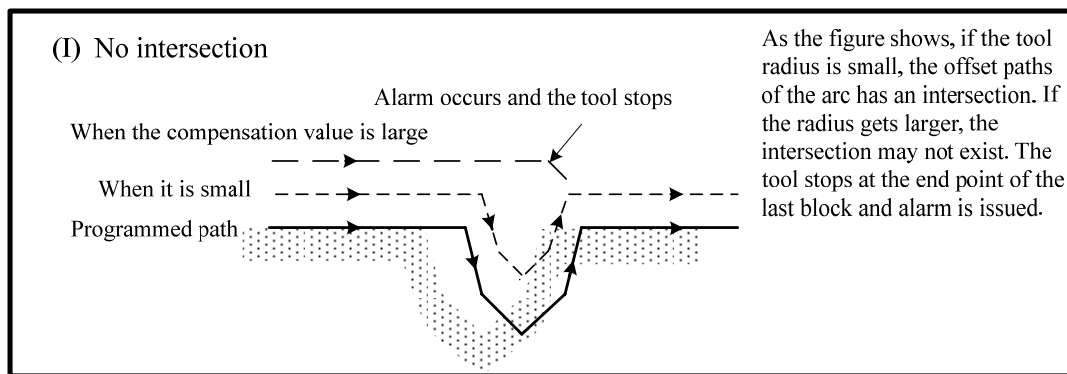


Fig. 4-5-3-4

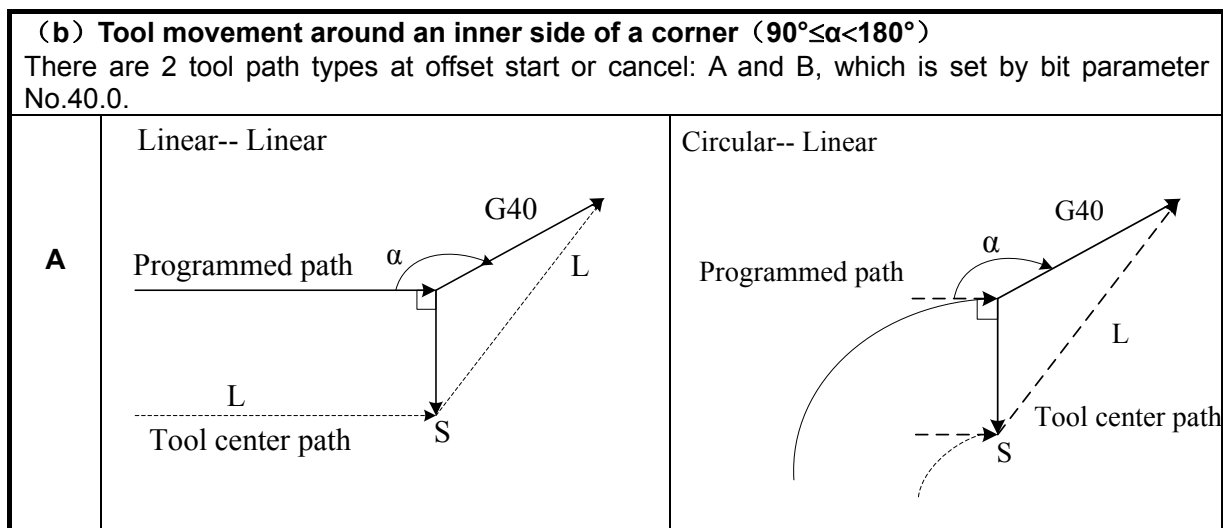
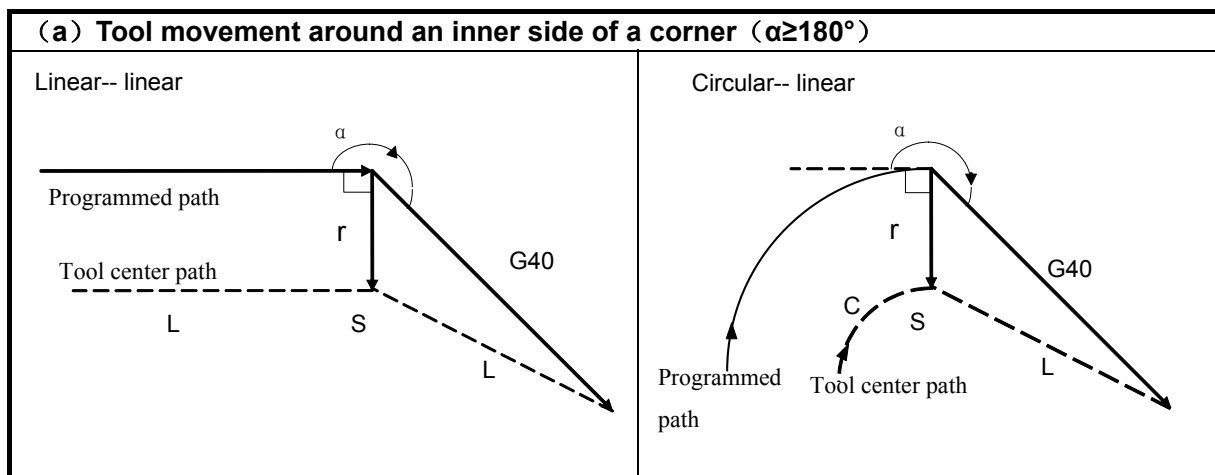
#### 4. Tool movement in offset cancel mode

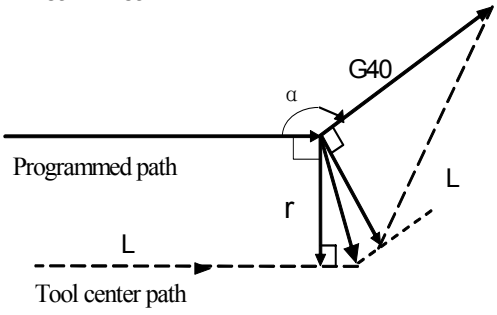
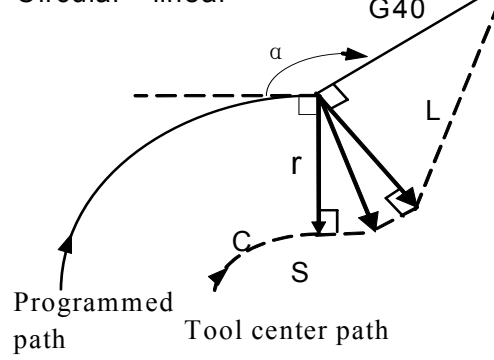
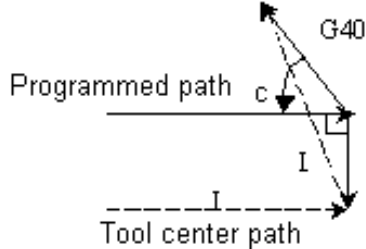
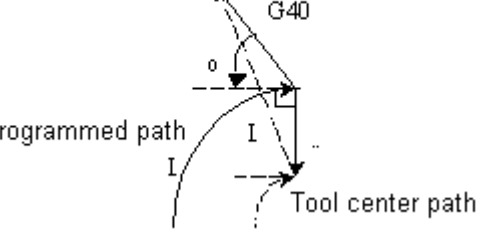
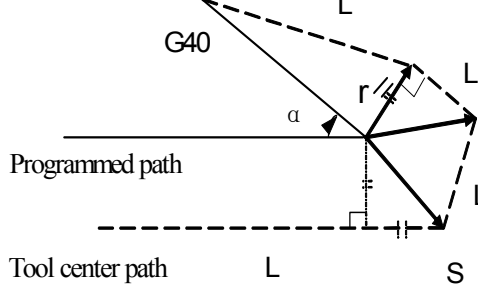
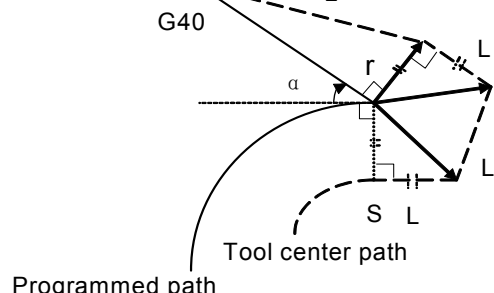
In offset mode, when the block complies to any of the following condition is executed, the system enters offset cancel mode. The operation of this block is called offset cancel.

a ) Instruction G40

b ) When the tool radius compensation number is 0:

Arc instruction (G03 or G02) is unallowed in offset cancel mode. Alarm is issued and tool stops if arc is specified



<p><b>B</b></p>	<p>Linear-- linear</p>  <p>Programmed path</p> <p>Tool center path</p>	<p>Circular-- linear</p>  <p>Programmed path</p> <p>Tool center path</p>
<p><b>(c) Tool movement around an outer side of a corner at an acute angle (<math>\alpha &lt; 90^\circ</math>)</b>  There are 2 tool path types at offset start or cancel: A and B, which is set by bit parameter NO: 40#0.</p>		
<p><b>A</b></p>	<p>Linear--linear</p>  <p>Programmed path</p> <p>Tool center path</p>	<p>Circular--linear</p>  <p>Programmed path</p> <p>Tool center path</p>
<p><b>B</b></p>	<p>Linear-- linear</p>  <p>Programmed path</p> <p>Tool center path</p>	<p>Circular-- linear</p>  <p>Programmed path</p> <p>Tool center path</p>

**(d) Tool movement around an outer side of a corner at an acute angle less than  $1^\circ$**   
**( $\alpha < 1^\circ$ )**  
**Linear→linear**

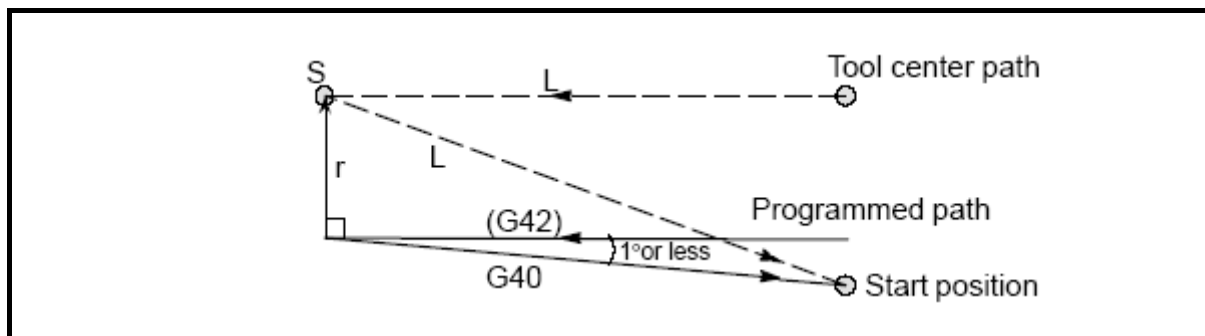


Fig. 4-5-3-5

### 5. Offset direction change in offset mode

The offset direction is defined by tool radius compensation G code. The sign of the offset value is as following:

Table 4-5-3-1

Sign of offset value G code	+	-
G41	Left offset	Right offset
G42	Right offset	Left offset

In a special situation, the offset direction can be changed in offset mode, however the direction change is unallowed in the start-up block and the block following it. There is no inner and outer side when the offset direction is changed. The following offset value is assumed to be positive.

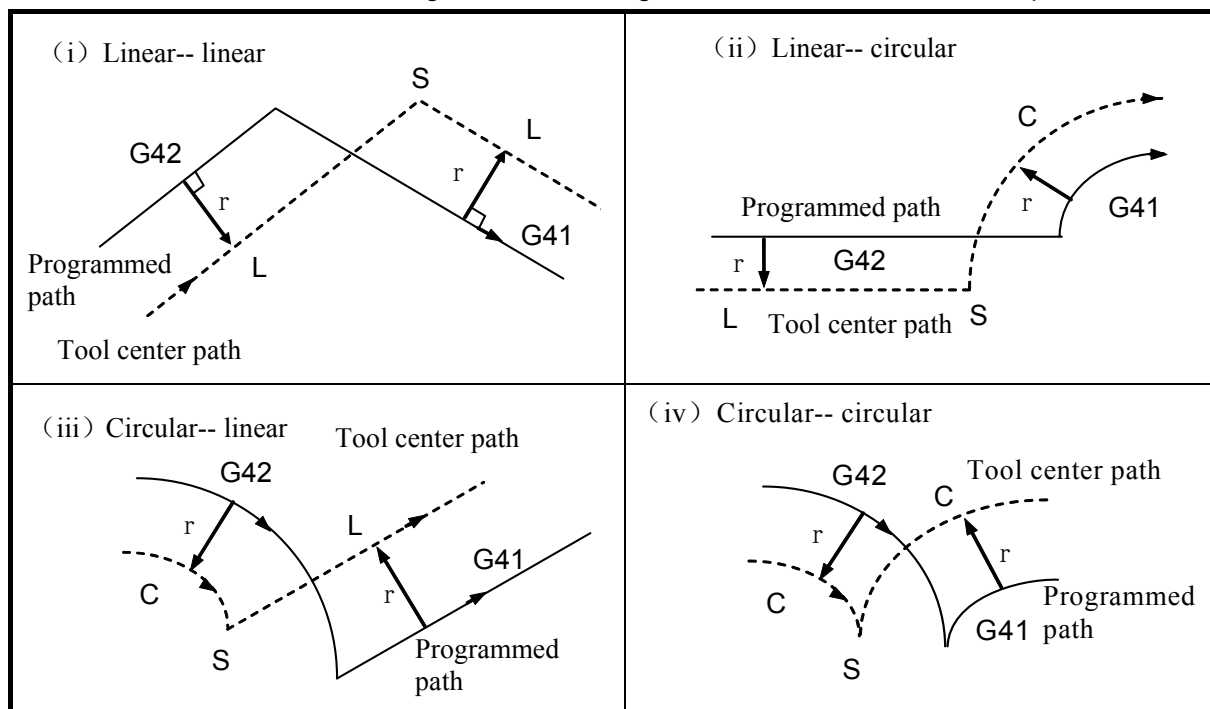


Fig. 4-5-3-6

(v) For the offset without an intersection

When changing the offset direction from block A to block B using G41 and G42, if intersection of the offset path is not required, the vector normal to block B is created at the start point.

(1) Linear---- linear

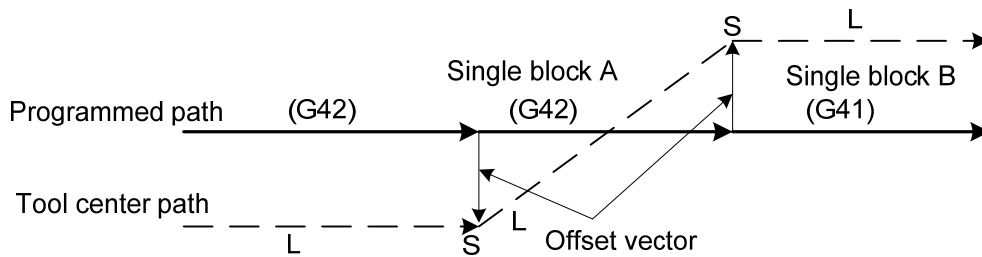


Fig. 4-5-3-7

(2) Linear----- circular

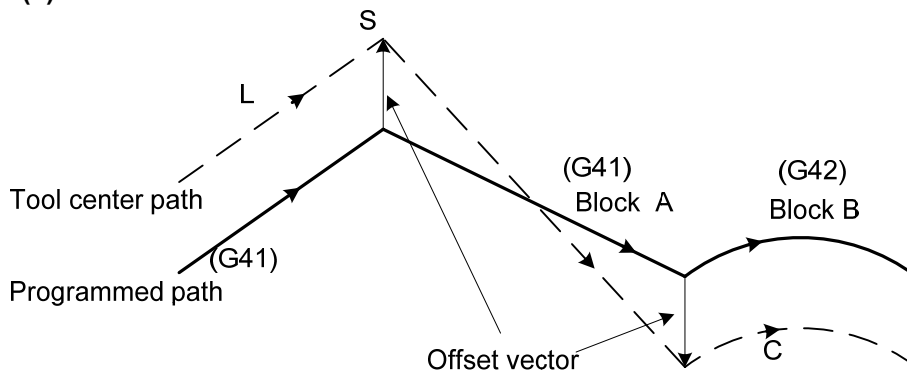


Fig. 4-5-3-8

(3) Circular----- circular

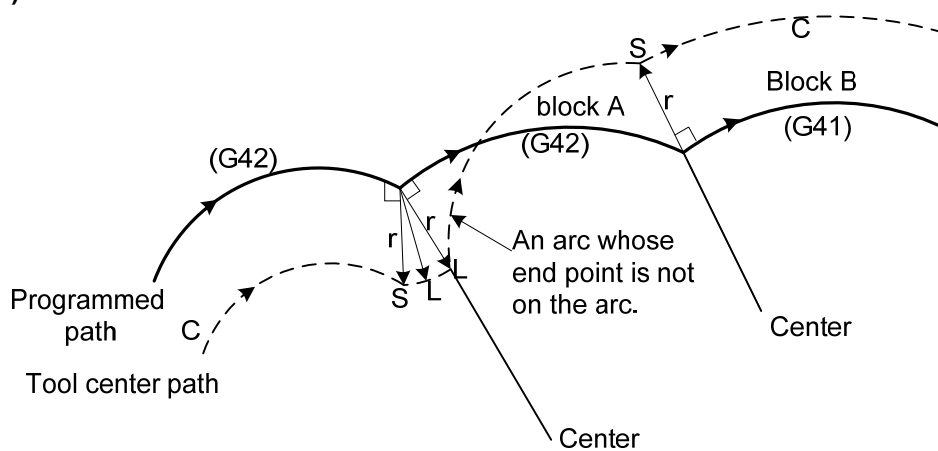


Fig. 4-5-3-9

(iv) Normally there is almost no possibility of generating the situation that the length of the tool center path is larger than the circumference of a circle. When G41 and G42 are changed, the following situation may occur:

Circular ----- circular (linear-----circular) Alarm occurs if the tool offset direction is changed and alarm that the tool offset can't be cancelled by arc instruction is issued when the tool number is D0.

Linear----- linear The tool offset direction can be changed.

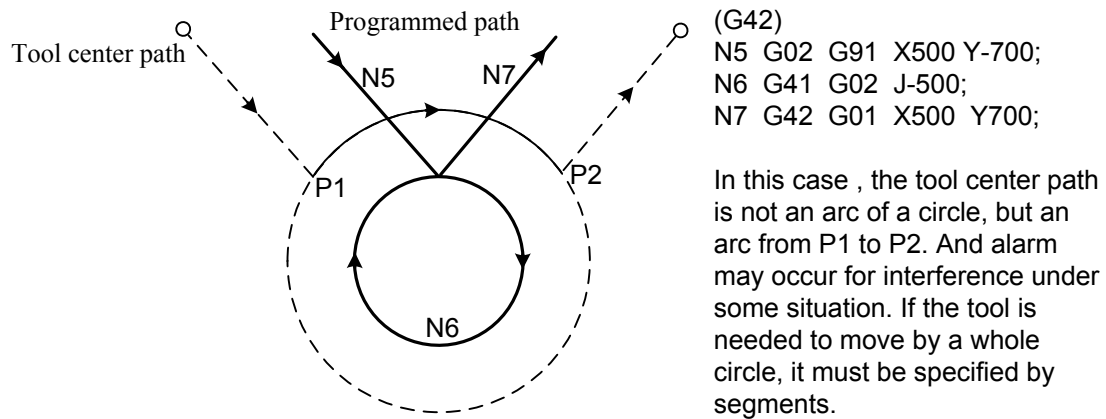


Fig. 4-5-3-10

## 6. Temporary offset cancel

in offset mode, if the compensation will be temporarily cancelled at the intermediate position when specifying G28、G30 is defined by parameter **NO: 40#2**

Refer to offset cancel and offset start for the details of this operation.

### a) G28 automatic reference point return

If G28 is specified in offset mode, the offset is cancelled at the intermediate position and automatically restored after reference point return.

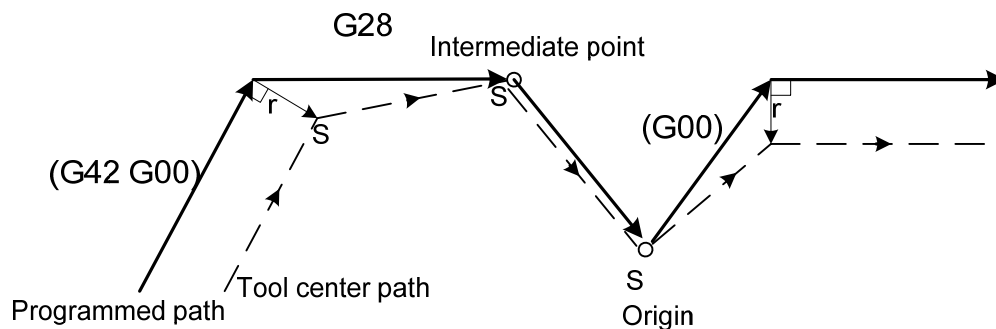


Fig. 4-5-3-11

### b) G29 automatic return from reference point

If G29 is specified in offset mode, the offset is cancelled at the intermediate position and automatically restored at the next block.

If it is specified immediately after G28:

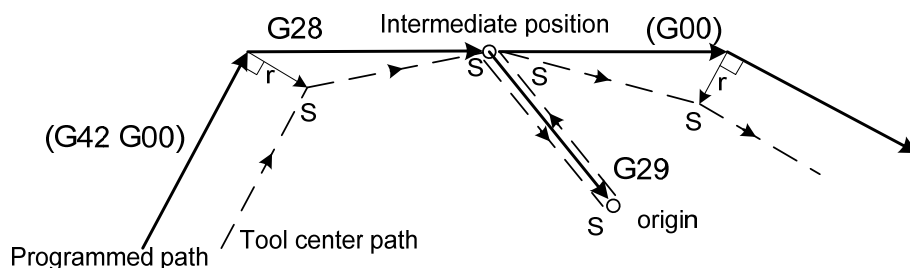


Fig. 4-5-3-12

If it is not specified immediately after G28:

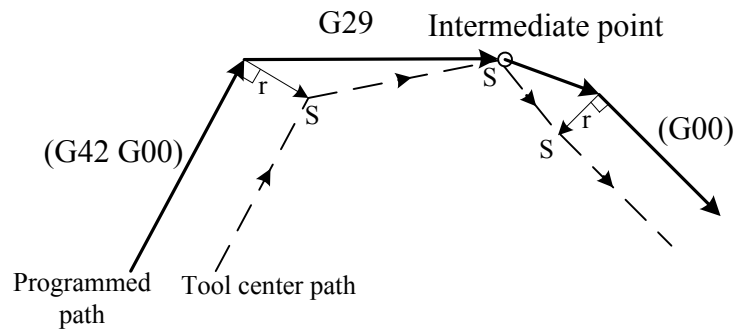


Fig. 4-5-3-13

### 7. Tool radius compensation G code in offset mode

In offset mode, if the tool radius compensation G code (G41, G42) is specified, relative to the move direction, a vector perpendicular to the previous block will be created, which is irrelative to the machining inner or outer side. If this G code is specified in circular instructions, the arc will not be correctly generated.

If this code is specified in a circular instruction, correct motion will not be obtained. Refer to ( 5 ) for offset direction change by tool radius compensation G ( G41, G42 )

#### Linear----- linear

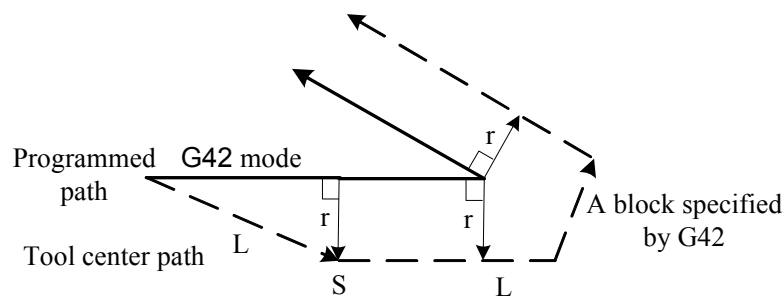


Fig. 4-5-3-14

#### Circular----- linear

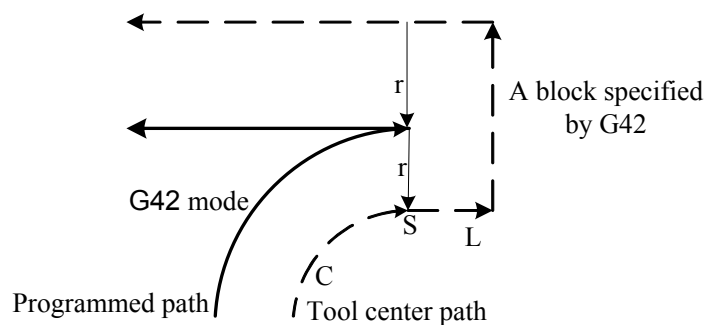


Fig. 4-5-3-15



### 8. Instruction for cancelling the offset vector temporarily

In offset mode, if G92 (absolute programming) is specified, the offset vector is temporarily cancelled and then the offset vector is restored automatically.

In this case, different from the offset cancel mode, the tool moves directly from the intersection to the specified point where the offset vector is cancelled. Also when offset mode is restored, the tool moves directly to the intersection.

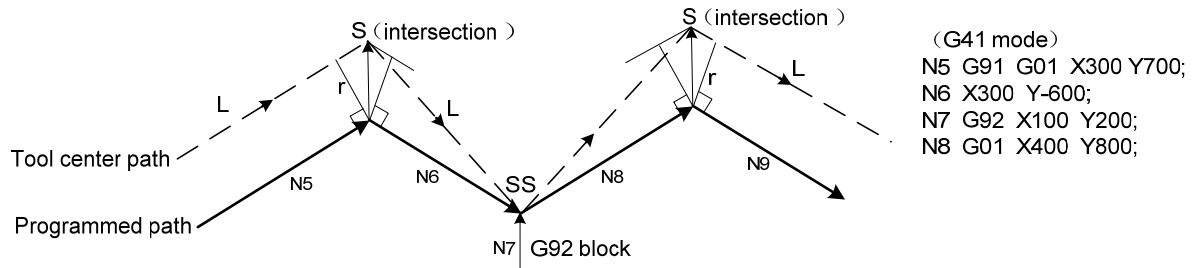


Fig. 4-5-3-16

### 9. A block without tool movement

The following blocks have no tool movement. In these blocks, the tool will not move even if tool radius compensation is effective.

- (1) M05 ; ..... M code output
- (2) S21 ; ..... S code output
- (3) G04 X10000; ..... Dwell
- (4) (G17) Z100 ; ..... Move instruction not included in offset plane
- (5) G90 ; ..... G code only
- (6) G01 G91 X0; ..... Move distance is zero.

#### a) Specified at offset start

If the tool movement is not made by the start-up block, it will be done by the next moving instruction block by the system.

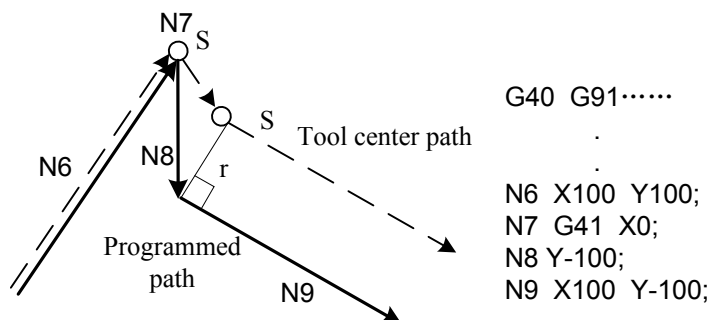


Fig. 4-5-3-17

#### b) Specified at offset mode

If a block with no tool movement is exclusively specified in offset mode, the vector and the tool center path are identical with that the block is not specified. (Refer to item (3) Offset mode). And this block is executed at the single block stop position.

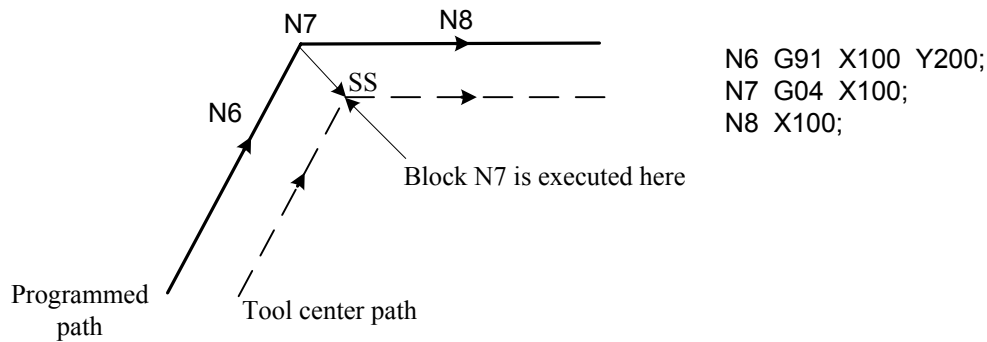


Fig. 4-5-3-18

However, when the block moving amount is 0, the tool movement is identical with that of the two or more blocks containing no moving instruction even only one block is specified.

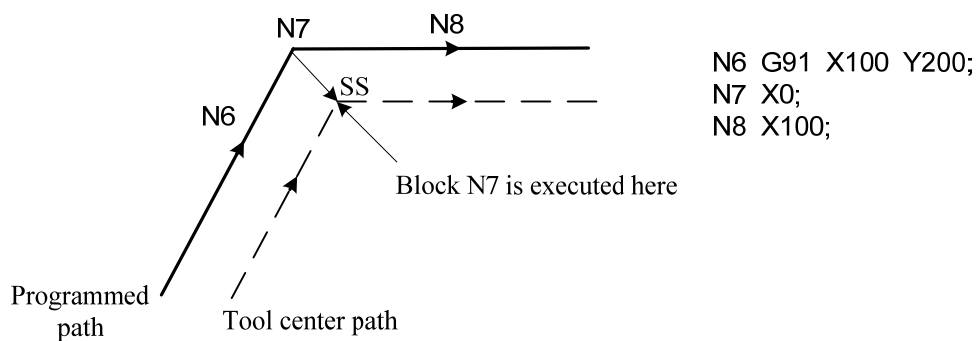


Fig. 4-5-3-19

**Note:** The blocks above are executed in G1, G41 mode and the path in G0 doesn't conform to the figure.

c) Specified with the offset cancel

A vector with a length offset and the direction perpendicular to the movement direction of the previous block is formed when the block specified together with offset cancel contains no tool movement, and it will be cancelled in next moving instruction.

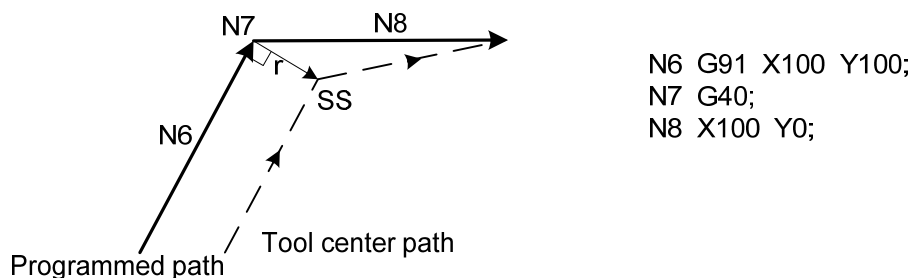


Fig. 4-5-3-20

## 10. Corner movement

If two or more vector are formed at the end of the block, the tool traverses straightly to another vector from one vector, the movement is called corner movement.

If  $\Delta V_x \leq \Delta V$  limit and  $\Delta V_y \leq \Delta V$  limit, the hind vectors are ignored.

If these vectors are not consistent, a movement around the corner is generated, which belongs to the hind block.

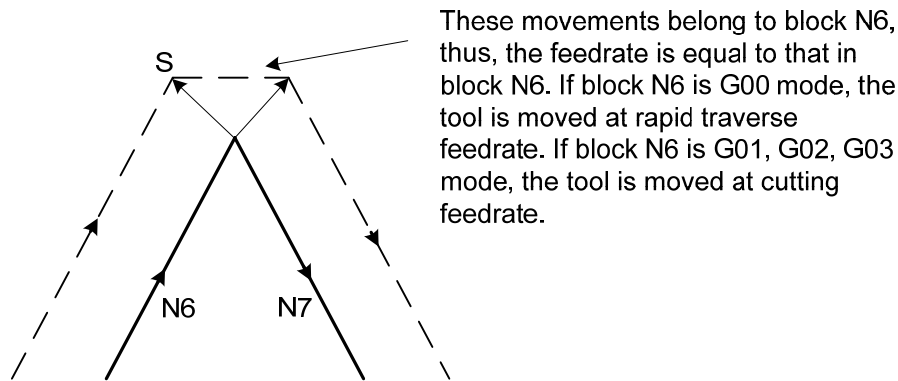


Fig. 4-5-3-21

But if the path of the next block overpasses the semicircle, the function above is not performed. The reason is that:

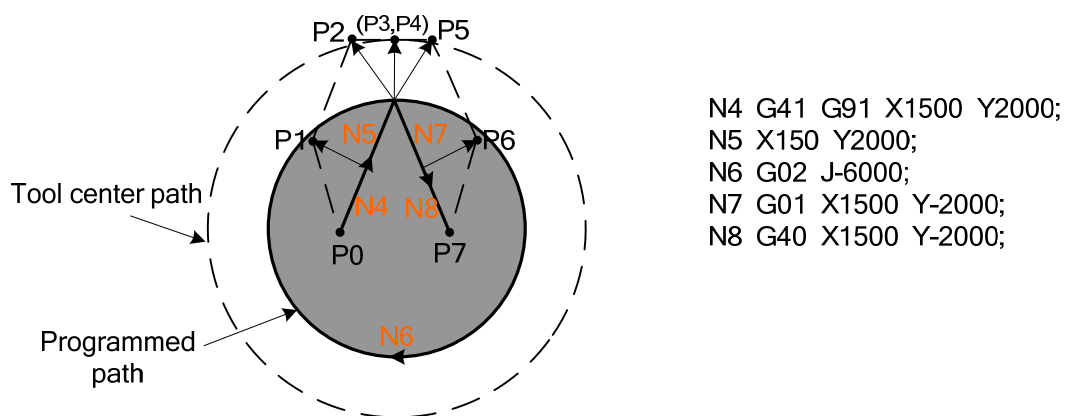


Fig. 4-5-3-22

If the vector is not ignored, the tool path is as follows:

P0 → P1 → P2 → P3 (arc) → P4 → P5 → P6 → P7

If the distance between P2 and P3 is ignored, P3 is ignored, the tool path is as follows:

P0 → P1 → P2 → P4 → P5 → P6 → P7 The arc cutting of the block N6 is ignored.

## 11. Interference check

The tool overcutting is called "interference". The Interference check function checks the tool overcutting in advance. If the interference is detected by grammar check function after the program is loaded, alarm is issued. The interference check in tool radius compensation is set by bit parameter **NO: 41#6**.

Primary conditions of interference:

- (1) The tool path is different from the program path. (The included angle between paths is from 90° to 270°).
- (2) Except above conditions, in arc machining, the included angle between the start point and the end point of the tool center path is much different from that of the program path (above 180°).

### Example

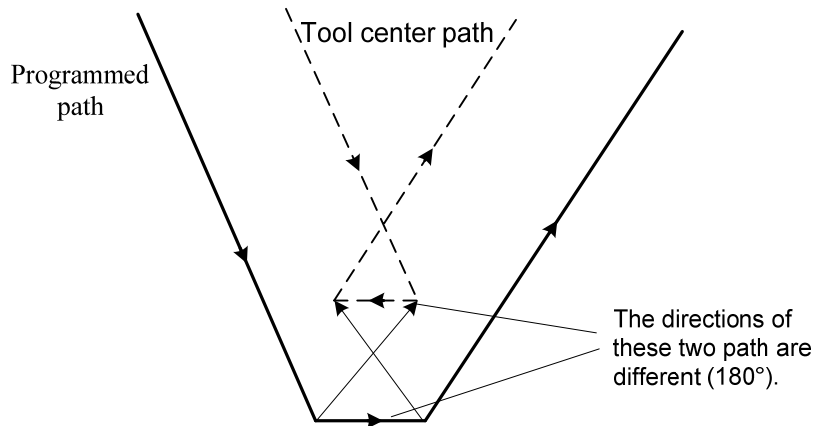


Fig. 4-5-3-23

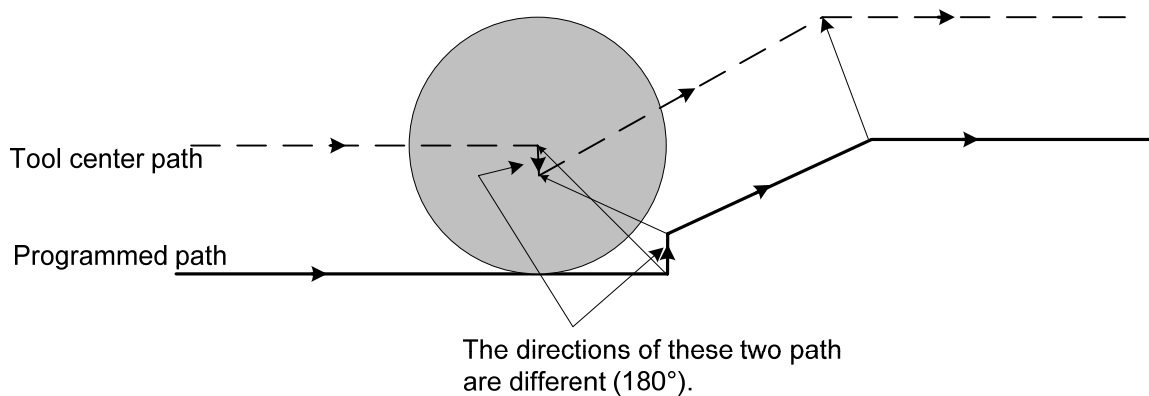


Fig. 4-5-3-24

## 12. Manual operation

See the manual operation in Operation section for the manual tool radius compensation. If the tool length compensation is performed in tool radius compensation, the offset value of the tool radius is regarded to be changed.

## 13. Precautions for offset

### (a) To specify offset value

The offset value number is specified by D code. Once specified, D code is effective till another one is specified or offset is cancelled. Besides the offset value for the tool radius compensation, it is also used for tool offset value.

### (b) To change the offset value

Usually during tool change, the offset value must be changed in offset cancel mode. If it is changed in offset mode, the new offset value is obtained at the block end.

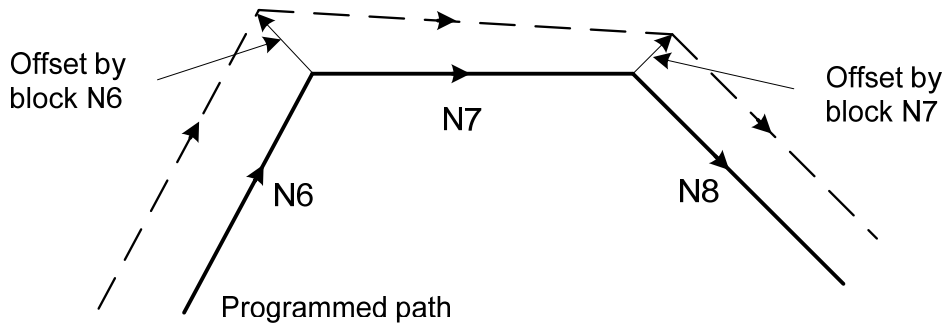


Fig. 4-5-3-25

## (c) Positive and negative tool offset value and tool center path

If the offset value is negative (  $-$  ), G41 and G42 is exchanged in program. If the tool center is moving around the outer side of the workpiece, it will pass around the inner side, and vice versa.

The figure below shows the example. Generally, the offset value is programmed to be positive (  $+$  ). When a tool path is programmed as in figure (a), if the offset value is made for negative (  $-$  ), the tool center moves as in (b), and vice versa. So the same program permits cutting for male or female shape, and the gap between them can be adjusted by the selection of the offset value.

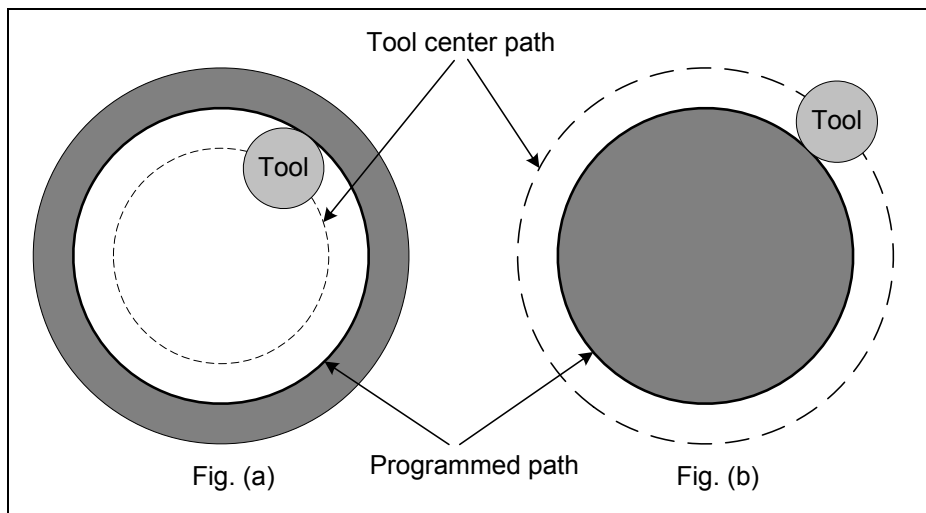
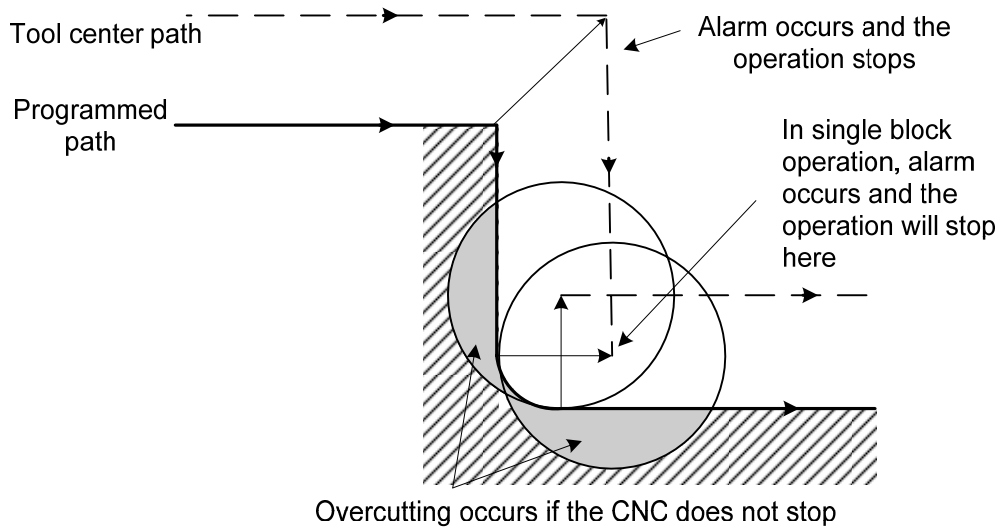


Fig. 4-5-3-26

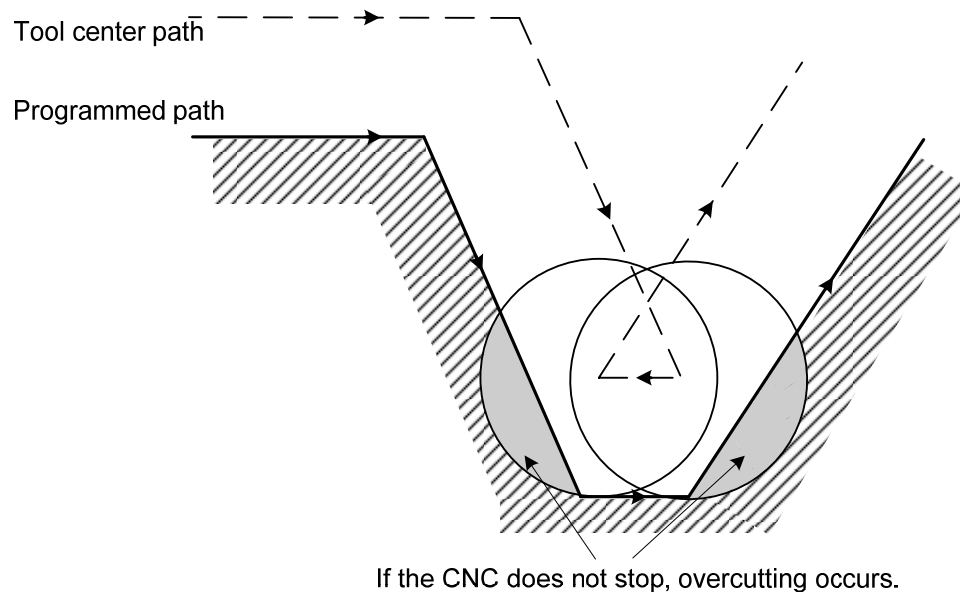
## (d) Overcutting by tool radius compensation

## (1) Machining an inner side of the corner at a radius smaller than the tool radius

When the radius of a corner is smaller than the tool radius, because the inner offsetting of the tool will result in overcuttings, an alarm will be issued and this is because overcutting is generated when the single block execution is stopped.

**Fig. 4-5-3-27**

(2) When machining a groove smaller than the tool radius, since the tool radius offset forces the path of the tool center to move in the reverse of the programmed direction, overcutting will result.

**Fig. 4-5-3-28**

### (3) Machining a step smaller than the tool radius

When machining a slot smaller than the tool radius specified by circular machining in the case of a program containing this step, the tool center path with the common offset becomes reverse to the programmed direction. In this case, the first vector is ignored, and the tool moves linearly to the second vector position. This single block operation is stopped at this point. If the machining is not in the single block mode, the auto run is continued. If the step is linear, no alarm will be issued and the tool cuts correctly. But uncut part will remain.

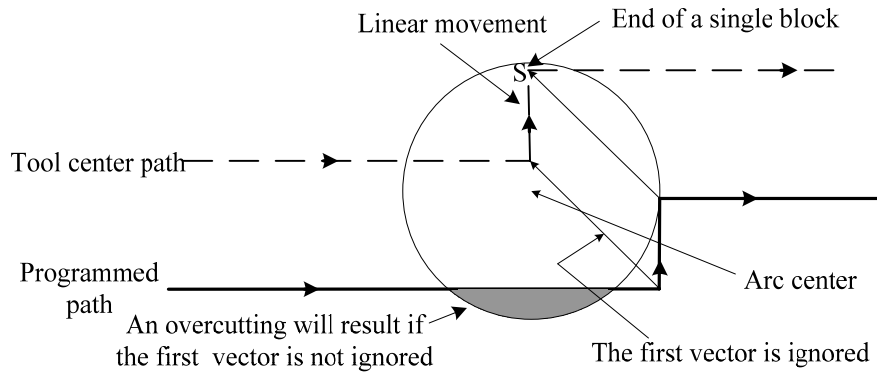


Fig. 4-5-3-29

Starting tool radius compensation and cutting along Z axis

It is usually used such a method that the tool is moved along the Z axis after the tool radius compensation is effected at some distance from the workpiece at the start of the machining. In the case above, if it is desired to divide the motion along the Z axis into rapid feed and cutting feed, follow the procedure below:

For block N3 (Z axis moving instruction), it is divided as following:

```
N1 G91 G00 X500 Y500 H01;
```

```
N3 Z-250;
```

```
N5 G01 Z-50 F1;
```

```
N6 Y100 F2;
```

```
N1 G91 G0 X500 Y500 H01;
```

```
N3 G01 Z-300 F1;
```

```
N6 Y100 F2;
```

N6 enters into the buffer when N3 is being executed. The relation between them is the offset as shown in figure.

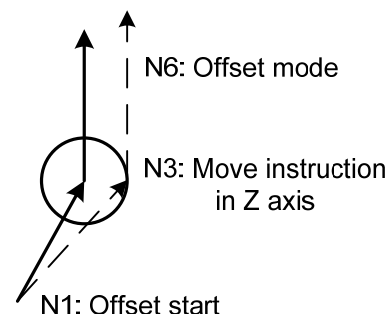


Fig. 4-5-3-30

#### 4.5.4 Corner offset circular interpolation (G39)

**Format:** G39 or  
 I\_ J\_  
 G39 I\_ K\_  
 J\_ K\_

**Function:** By specifying G39 in offset mode during tool radius compensation, corner offset circular interpolation can be specified. The radius of the corner offset equals the offset value. And the effectiveness of the corner arc in radius compensation is set by bit parameter **NO: 41#5**.

**Explanation:**

1. When G39 is specified, corner circular interpolation in which the radius equals offset value can be performed.

2. G41 or G42 preceding this instruction determines whether the arc is CW or CCW. G39 is a non-modal G code.
3. When G39 (without I, J, K) is programmed, the arc at the corner is formed so that the vector at the end point of the arc is perpendicular to the start point of the next block. It is shown as follows:

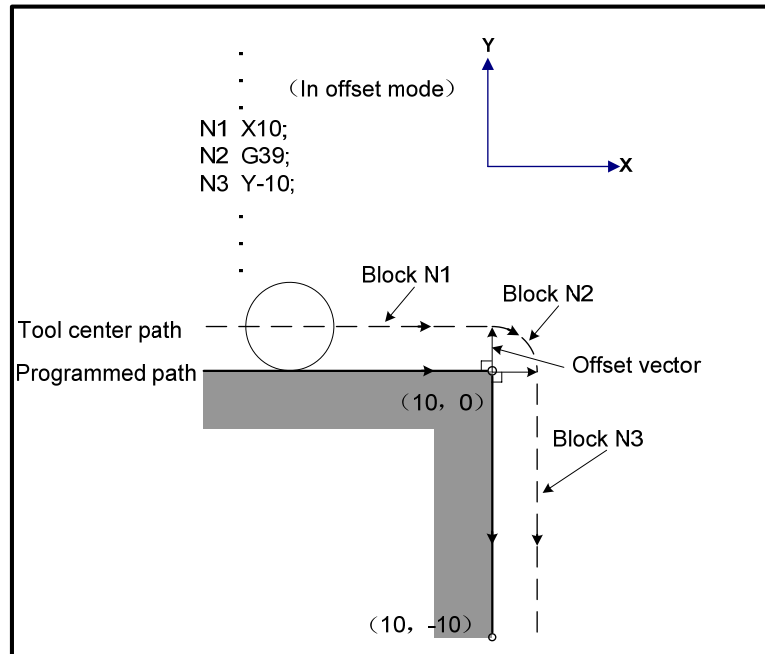


Fig. 4-5-4-1 G39

#### 4.5.5 Tool offset value and number input by program (G10)

##### Format:

G10 L10 P\_ R\_ ; Geometric offset value of H code

G10 L12 P\_ R\_ ; Geometric offset value of D code

G10 L11 P\_ R\_ ; Wear offset value of H code

G10 L13 P\_ R\_ ; Wear offset value of D code

P : Tool offset number

R : Tool offset value in absolute mode (G90)

For the tool offset value in incremental mode (G91), it is added by the value of the offset number specified (the result is the tool offset value.)

**Explanation:** The range of tool offset value:

Geometric offset: metric input  $\pm 999.999\text{mm}$ ; inch input  $\pm 99.9999\text{ inch}$

Wear offset: metric input  $\pm 99.999\text{mm}$ ; inch input  $\pm 9.9999\text{ inch}$

- Note**
- 1 For inch and metric conversion, the tool offset value automatic change is set by bit parameter No.41#0.
  - 2 The max. value of the wear offset is restrained by data P267.



## 4.6 Feed G code

### 4.6.1 Feed mode G64/G61/G63

**Format:**

Dwell (exact stop) mode G61

Tapping mode G63

Cutting mode G64

**Function:**

Dwell mode G61: Once specified, this function is effective till G62, G63 or G64 is specified. The tool is decelerated for an in-position check at the end point of a block, then next block is executed.

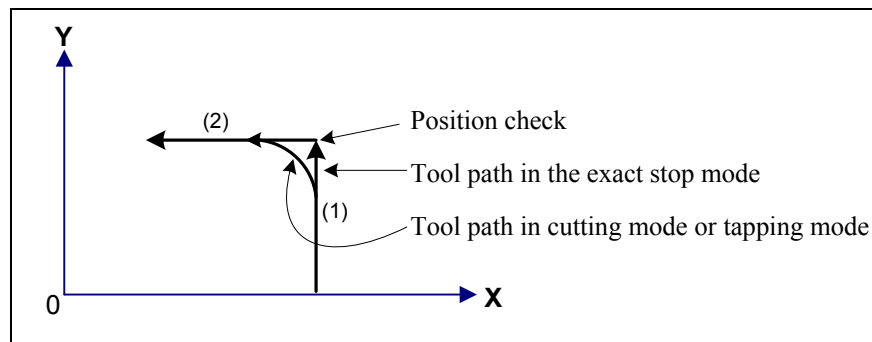
Tapping mode G63: Once specified, this function is effective till G62, G61 or G64 is specified. The tool is not decelerated at the end point of a block, but the next block is executed. When G63 is specified, feedrate override and feed hold are both ineffective.

Cutting mode G64: Once specified, this function is effective till G62, G61 or G63 is specified. The tool is not decelerated at the end point of a block, and the next block is executed.

**Explanation:**

1. No parameter format.
2. G64 is the system default feed mode, no deceleration is performed at the end point of a block and next block is executed directly.
3. The purpose of in-position check in dwell mode is to check whether the servo motor has reached within a specified range.
4. In exact stop mode, the tool movement paths in cutting mode and tapping mode are different.

See following **Fig. 4-6-1-1**:



**Fig. 4-6-1-1 Tool path from block 1 to block 2**

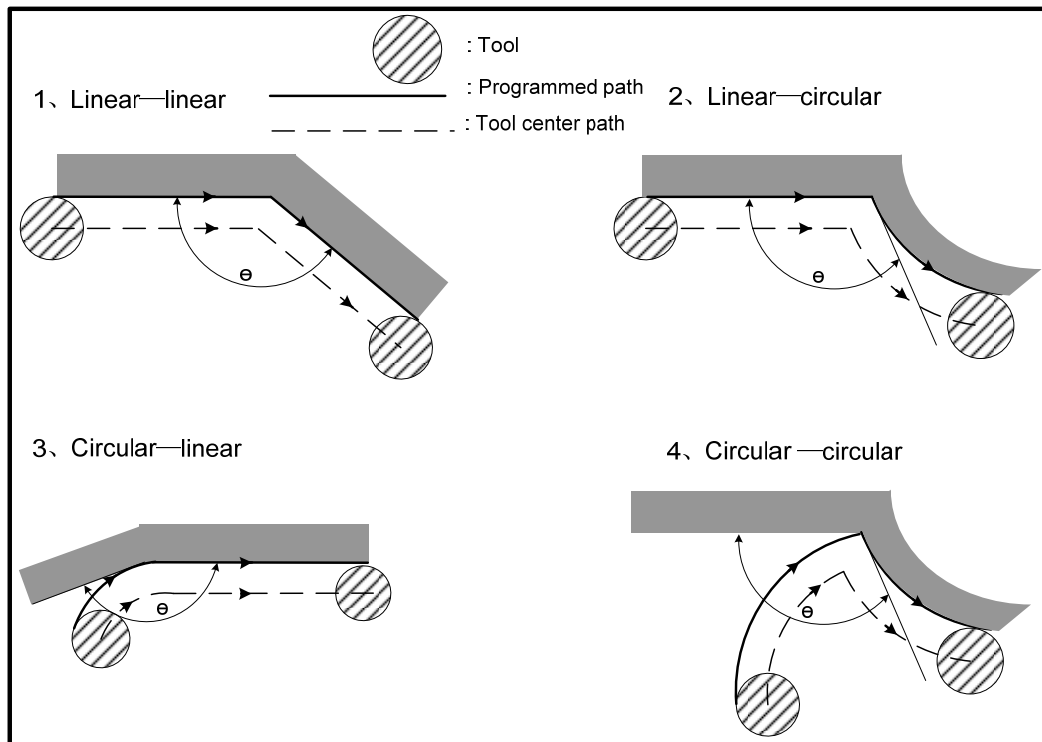
### 4.6.2 Automatic override for inner corners (G62)

**Format:** G62

**Function:** Once specified, this function is effective till G63, G61 or G64 is specified. When the tool moves along an inner corner during tool radius compensation, override is applied to the cutting feedrate to suppress the amount of cutting per unit of time to get a good surface finish.

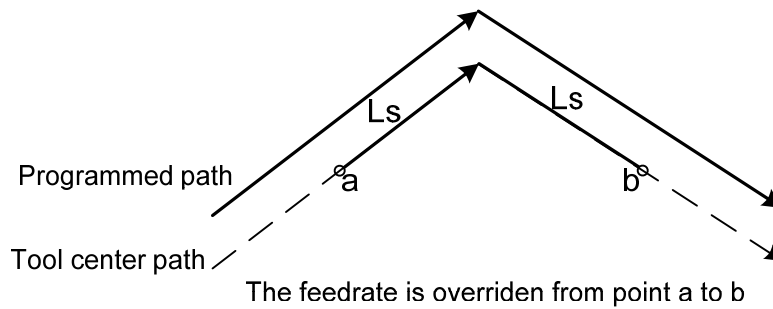
**Explanation:**

1. When the tool moves along an inner corner and inner arc area during tool radius compensation, it decelerates automatically to reduce the load of the tool to get a smooth surface.
2. Whether automatic corner override function is valid or not is set by bit parameter **NO: 16#7**; Automatic corner deceleration function is controlled by bit parameter NO: 15#2(0: angle control, 1: speed difference control).
3. When G62 is specified, and the tool path with tool radius compensation forms an inner corner, the feedrate is automatically overridden at both ends of the corner. There are four types of inner corners as shown in Fig. 4-6-2-1. In figure:  $2^\circ \leq \theta \leq \theta_p \leq 178^\circ$ ;  $\theta_p$  is set by number parameter P144.



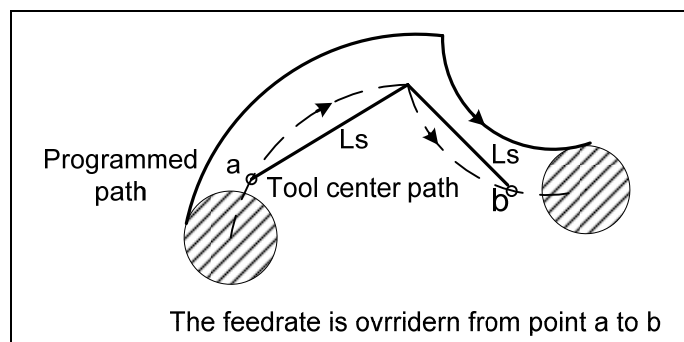
**Fig. 4-6-2-1**

4. When a corner is determined to be an inner corner, the feedrate is overridden before and after the inner corner. The  $L_s$  and  $L_e$ , where the feedrate is overridden, are distances from points on the tool center path to the corner (Fig. 4-6-2-2), where  $L_s + L_e \leq 2\text{mm}$ .



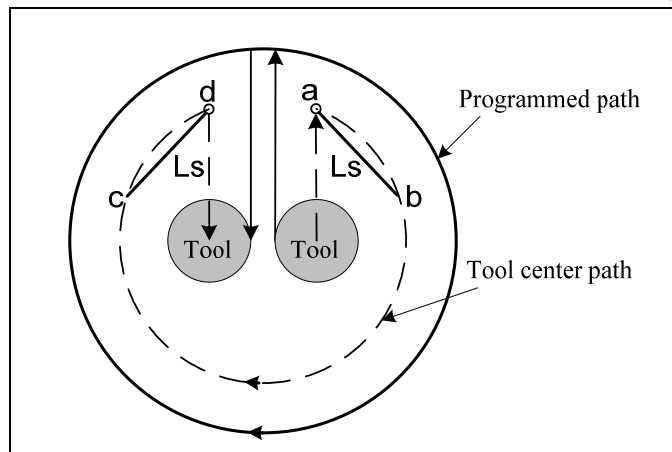
**Fig. 4-6-2-2 Straight line to straight line**

- 5 When a programmed path consists of two arcs, the feedrate is overridden if the start and end points are in the same quadrant or in adjacent quadrants. The min.deceleration feed speed of auto corner deceleration is controlled by data parameter P145 (Fig. 4-6-2-3).



**Fig. 4-6-2-3 Arc to arc**

- 6 Regarding a program from straight line to arc or from arc to straight line, the feedrate is overridden from point a to point b and from point c to point d. (Fig. 4-6-2-4)



**Fig. 4-6-2-4 Straight line to arc, arc to straight line**

### Restriction

- 1 Override for inner corners is disabled during acceleration/deceleration before interpolation.
- 2 Override for inner corners is disabled if the corner is preceded by a start-up block or followed by a block including G41 or G42.
- 3 Override for inner corner is not performed if the offset is zero.

## 4.7 Macro G code

### 4.7.1 Custom macro

The function by a group of instructions can be saved into memory like a subprogram in advance, and the functions are represented by an instruction. If the instruction is written out in the program, these functions can be used. This group of instructions is called custom macro body and the instruction represented is called “custom macro instruction”. The custom macro body is also abbreviated for macro. The custom macro instruction is also called macro calling instruction.

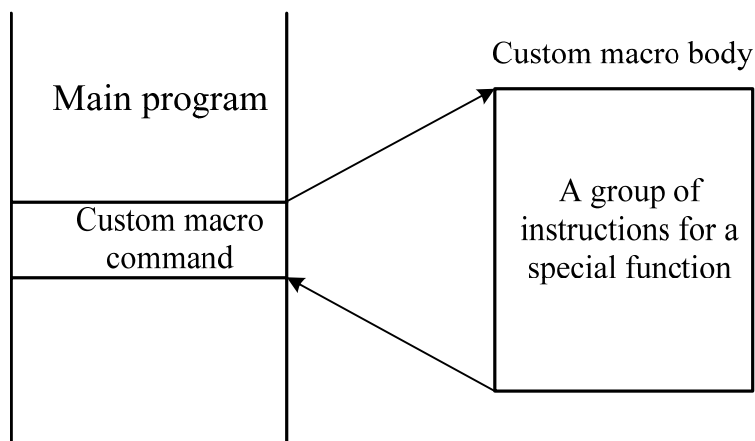


Fig. 4-7-1-1

Variables can be used in custom macro body, and they can be operated and assigned by macro instructions.

### 4.7.2 Macro variables

Both the common CNC instructions and the variables, operation as well as the transfer instructions can be used in the custom macro body. It begins with program number and ends with M99.

O0066;	Program number
G65 H01 .....;	Operation command
G90 G00 X#101 .....;	CNC instructions using variables
.....	
.....	
.....	
G65 H82 .....;	Transfer command
.....	
.....	
M99;	End of custom macro body

Fig. 4-7-2-1 the composition of custom macro body

#### 1. Variables usage

With a variable, the parameter value in custom macro body can be specified. The variable value can be assigned by the main program or set by LCD/MDI; or be assigned by a computation during custom macro body execution.

Multiple variables can be used in custom macro and they are differentiated by their variable

numbers.

(1) Variable representation

The variable is expressed by a sign # followed by a variable number, Format:

#i (i = 1, 2, 3, 4 .....)

(e.g.) #5, #109, #1005

(2) Variable citation

The variable can be used to replace the value of parameter.

(e.g)        F#103        When #103 = 15, it is the same as F15.  
               G#130        When #103 =3, it is the same as G3.

**Note 1** Variable can't be cited by parameter word O and N (program number and sequence number). Such as O#100, N#120 are not permitted in programming.

**Note 2** Variable exceeding the max. limit of the parameter can't be used. When #30 = 120, M#30 exceeds the max. limit of the instruction.

**Note 3** Display and setting of variable: It can be displayed on LCD, or be set by MDI.

## 2. Typies of variables

Variables are classified into null variables, local variables, common variables and system variables with different applications and characteristics.

(1) Null variables: #0 (This variable always be null, no value can be assigned to it.)

(2) Local variables: #1~#50: They can only be used for data storage in a macro such as the results of operations. When the power off, reset or program end (execute M30, M02), they are cleared automatically. When a macro is called, arguments are assigned to local variables.

(3) Common variables: #100~#199, #500~#999: They can be shared among the main program and the custom macros called by the main program. Namely the variable #i in a custom macro program is identical with that in other macro program. So the common variable #i of operation result of a macro program can be used in other macro programs.

Common variables applications are not specified in this system and they can be used freely by user

Table 4-7-1-1

Variable number	Variable type	Function
# 100 ~ # 199	Common variables	Cleared at power-off, all reset for "null" at power-on
# 500 ~ # 999		Data saved in files and reserved even if power-off

(4) System variables:

They are used for reading and writing a variety of CNC data, which are shown as follows:

- 1) Interface input signal    #1000 --- #1047    (read signal input by PLC by bits)
- 2) Interface output signal    #1100 --- #1147    (write signal output to PLC by bits)
- 3) Tool length offset value    #1500~#1755    (readable and writable)

- 4) Tool length wear offset value #1800～#2055 (readable and writable)
- 5) Tool radius offset value #2100～#2355 (readable and writable)
- 6) Tool radius wear offset value #2400～#2655 (readable and writable)
- 7) Alarm #3000
- 8) User data list #3500～#3755 (read-only, unwritable)
- 9) Modal message #4000～#4030 (read-only, unwritable)
- 10) Position message #5001～#5030 (read-only, unwritable)
- 11) Workpiece zero offset #5201～#5235 (readable and writable)
- 12) Additional workpiece coordinate system #7001～#7250 (readable and writable)

### 3. Explanation for system variables

#### 1) Modal message

**Table 4-7-1-2**

Variables No.	Function	Group No.
#4000	G10,G11	00
#4001	G00,G01,G02,G03	01
#4002	G17,G18,G19	02
#4003	G90,G91	03
#4004	G94,G95	04
#4005	G54,G55,G56,G57,G58,G59	05
#4006	G20,G21	06
#4007	G40,G41,G42	07
#4008	G43,G44,G49	08
#4009	G73,G74,G76,G80,G81,G82,G83,G84,G85,G86,G87,G88,G89	09
#4010	G98,G99	10
#4011	G15,G16	11
#4012	G50,G51	12
#4013	G68,G69	13
#4014	G61,G62,G63,G64	14
#4015	G96,G97	15
#4016	To be expanded	16
#4017	To be expanded	17
#4018	To be expanded	18
#4019	To be expanded	19
#4020	To be expanded	20
#4021	To be expanded	21
#4022	D	
#4023	H	
#4024	F	
#4025	M	
#4026	S	
#4027	T	
#4028	N	
#4029	O	
#4030	P(current selected additional workpiece coordinate system)	

**Note 1** P code stands for the current selected additional workpiece coordinate system.

**Note 2** When G#4002 code is being executed, the value obtained in #4002 is 17, 18 or 19.

**Note 3** The modal message can be read, but cannot be written.

2) Current position message

**Table 4-7-1-3**

Variable No.	Position message	Relative coordinate system	Read operation as moving	Tool offset value	
#5001	Block end position of X axis (ABSIO)	Workpiece coordinate system	Allowed	Tool nose position not involved (program specified position)	
#5002	Block end position of Y axis (ABSIO)				
#5003	Block end position of Z axis (ABSIO)				
#5004	Block end position of 4th axis (ABSIO)				
#5006	Block end position of X axis (ABSMT)	Machine coordinate system	Unallowed	Tool basic position involved (Machine coordinate)	
#5007	Block end position of Y axis (ABSMT)				
#5008	Block end position of Z axis (ABSMT)				
#5009	Block end position of 4th axis (ABSMT)				
#5011	Block end position of X axis (ABSOT)	Workpiece coordinate system			
#5012	Block end position of Y axis (ABSOT)				
#5013	Block end position of Z axis (ABSOT)				
#5014	Block end position of 4th axis (ABSOT)				
#5016	Block end position of X axis (ABSKP)			Allowed	
#5017	Block end position of Y axis (ABSKP)				
#5018	Block end position of Z axis (ABSKP)				
#5019	Block end position of 4th axis (ABSKP)				
#5021	Tool length offset value of X axis			Unallowed	
#5022	Tool length offset value of Y axis				
#5023	Tool length offset value of Z axis				
#5024	Tool length offset value of 4th axis				
#5026	Servo position offset of X axis				
#5027	Servo position offset of Y axis				
#5028	Servo position offset of Z axis				
#5029	Servo position offset of 4th axis				

**Note:** 1、ABSIO: the last block end point coordinate in workpiece coordinate system.  
2、ABSMT: the current machine coordinate system position in machine coordinate system  
3、ABSOT: the current coordinate position in workpiece coordinate system.

4、ABSKP: effective position of G31 block skip signal in workpiece coordinate system。

3) Workpiece zero offset value and additional zero offset value

**Table 4-7-1-4**

Variable No.	Function
#5201	External workpiece zero offset value of the 1st axis
...	...
#5204	External workpiece zero offset value of the 4th axis
#5206	G54 workpiece zero offset value of the 1st axis
...	...
#5209	G54 workpiece zero offset value of the 4th axis
#5211	G55 workpiece zero offset value of the 1st axis
...	...
#5214	G55 workpiece zero offset value of the 4th axis
#5216	G56 workpiece zero offset value of the 1st axis
...	...
#5219	G56 workpiece zero offset value of the 4th axis
#5221	G57 workpiece zero offset value of the 1st axis
...	...
#5224	G57 workpiece zero offset value of the 4th axis
#5226	G58 workpiece zero offset value of the 1st axis
...	...
#5229	G58 workpiece zero offset value of the 4th axis
#5231	G59 workpiece zero offset value of the 1st axis
...	...
#5234	G59 workpiece zero offset value of the 4th axis
#7001	G54 P1 workpiece zero offset value of the 1st axis
...	...
#7004	G54 P1 workpiece zero offset value of the 4th axis
#7006	G54 P2 workpiece zero offset value of the 1st axis
...	...
#7009	G54 P2 workpiece zero offset value of the 4th axis
#7246	G54 P50 workpiece zero offset value of the 1st axis
...	...
#7249	G54 P50 workpiece zero offset value of the 4th axis

#### 4. Local variable

The correspondence of address and local variable:

**Table 4-7-1-5**

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

**Note 1** The assignment is done by an English letter followed by a numerical value. Besides the letters G, L, O, N, H and P, the other 20 letters can also be assigned for



arguments. Each letter from A-B-C-D... to X-Y-Z can be assigned once and they need not to be assigned by letter order. The addresses not assigned may be omitted.

**Note 2** G65 should be specified prior to argument using.

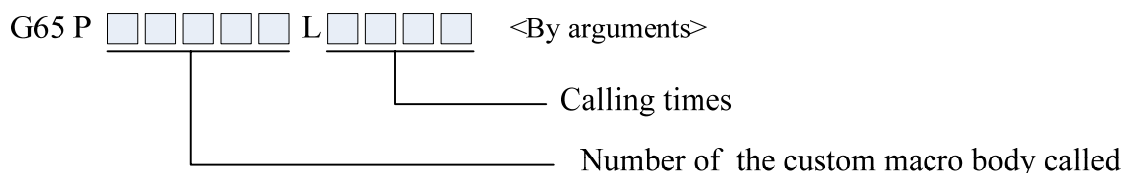
### 5. Precautions for custom macro body

- 1) Input by keys  
Press # key behind the parameter words G, X, Y, Z, R, I, J, K, F, H, M, S, T, P, Q for inputting "#"
- 2) Either operation or transfer instruction can be specified in MDI mode.
- 3) H, P, Q, R of the operation and transfer instructions preceding or behind G65 are all used as parameters for G65.  
H02 G65 P#100 Q#101 R#102 ;           Correct  
N100 G65 H01 P#100 Q10 ;           Correct
- 4) Variable range: -9999.9999~9999.9999
- 5) The result of the variable operation can be a decimal with a precision of 0.0001. All operations, except H11 (OR operation), H12 (AND operation), H13 (NOT operation), H23 (ROUNDING operation) with decimal portion neglected in operation, are done without the decimal portions abnegated.  
Example:  
#100 = 35, #101 = 10, #102 = 5  
#110 = #100÷#101           (=3.5)  
#111 = #110×#102           (=17.5)  
#120 = #100×#102           (=175)  
#121 = #120÷#101           (=17.5)
- 6) The execution time of operation and transfer instruction differs depending on different conditions, usually the average time is 10ms.

### 4.7.3 Custom macro call

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

**Format:**



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

If repetition is needed, specify the repetition times (from 1 to 9999 ) behind L code; if L is omitted, the default time is 1. If it is specified by arguments, the values will be assigned to the corresponding local variables.

**Note 1** If the subprogram number specified by address P is not found, an alarm (PS 078)

will be issued.

**Note 2** No. 90000~99999 subprograms are the system reserved programs, if they are called, they can be executed, but the cursor will rest on at M98 block, and the program interface displays the main program all the time ( the subprogram can be displayed and edited by modifying bit parameter N0: 27#4.)

**Note 3** Macroprogram can not be called in DNC mode.

#### 4.7.4 Custom macro function A

##### 1. Format:

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

m: 01~99 represent the operation or transfer function.

#i: Variable name for saving operation result.

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

#k: Variable name 2 for operation. Or a constant.

**Significance:**  $\#i = \#j \circ \#k$

\_\_\_\_\_ Operation sign, specified by Hm

**Example:** P#100 Q#101 R#102.....#100 = #101  $\circ$  #102 ;

P#100 Q#101 R15 .....#100 = #101  $\circ$  15 ;

P#100 Q-100 R#102.....#100 = -100  $\circ$  #102 ;

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

**Table 4-7-4-1**

G code	H code	Function	Definition
G65	H01	Value assignment	$\#i = \#j$
G65	H02	Addition	$\#i = \#j + \#k$
G65	H03	Subtraction	$\#i = \#j - \#k$
G65	H04	Multiplication	$\#i = \#j \times \#k$
G65	H05	Division	$\#i = \#j \div \#k$
G65	H11	Logic addition(OR)	$\#i = \#j \text{ OR } \#k$
G65	H12	Logic multiplication (AND)	$\#i = \#j \text{ AND } \#k$
G65	H13	AND-OR	$\#i = \#j \text{ XOR } \#k$
G65	H21	Square root	$\#i = \sqrt{\#j}$
G65	H22	Absolute value	$\#i =  \#j $
G65	H23	Compliment	$\#i = \#j - \text{trunc}(\#j \div \#k) \times \#k$
G65	H24	Algorism to binary	$\#i = \text{BIN}(\#j)$
G65	H25	Binary to algorism	$\#i = \text{BCD}(\#j)$
G65	H26	Compound multiplication and division	$\#i = (\#i \times \#j) \div \#k$
G65	H27	Compound square root	$\#i = \sqrt{\#j^2 + \#k^2}$
G65	H31	Sine	$\#i = \#j \times \text{SIN}(\#k)$
G65	H32	Cosine	$\#i = \#j \times \text{COS}(\#k)$
G65	H33	Tangent	$\#i = \#j \times \text{TAN}(\#k)$

G65	H34	Arctangent	#i = ATAN(#j/#k)
G65	H80	Unconditional transfer	Turning N
G65	H81	Conditional transfer 1	IF #j = #k, GOTO N
G65	H82	Conditional transfer 2	IF #j ≠ #k, GOTO N
G65	H83	Conditional transfer 3	IF #j > #k, GOTO N
G65	H84	Conditional transfer 4	IF #j < #k, GOTO N
G65	H85	Conditional transfer 5	IF #j ≥ #k, GOTO N
G65	H86	Conditional transfer 6	IF #j ≤ #k, GOTO N

## 2. Operation instruction:

- 1) Variable assignment: # I = # J

### G65 H01 P#I Q#J

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

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

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

- 2) Addition: # I = # J + # K

### G65 H02 P#I Q#J R#K

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

- 3) Subtraction: # I = # J - # K

### G65 H03 P#I Q#J R#K;

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

- 4) Multiplication: # I = # J × # K

### G65 H04 P#I Q#J R#K;

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

- 5) Division: # I = # J ÷ # K

### G65 H05 P#I Q#J R#K

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

- 6) Logic addition (OR): # I = # J .OR. # K

### G65 H11 P#I Q#J R#K;

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

- 7) Logic multiplication (AND): # I = # J .AND. # K

### G65 H12 P#I Q#J R#K;

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

- 8) AND-OR: # I = # J .XOR. # K

### G65 H13 P#I Q#J R#K

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

- 9) Square root: # I =  $\sqrt{\#J}$

### G65 H21 P#I Q#J ;

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

- 10) Absolute value: # I = | # J |

### G65 H22 P#I Q#J ;

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

- 11) Rounding: # I = # J - TRUNC(#J/#K) × # K, TRUNC: reserving or abnegating decimal portion

**G65 H23 P#I Q#J R#K**

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

- 12) Algorithm to binary: # I = BIN (# J)

**G65 H24 P#I Q#J ;**

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

- 13) Binary to algorism: # I = BCD (# J)

**G65 H25 P#I Q#J ;**

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

- 14) Compound multiplication and division: # I = (# I×# J) ÷# K

**G65 H26 P#I Q#J R# k;**

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

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

**G65 H27 P#I Q#J R#K**

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

- 16) Sine: # I = # J•SIN (# K) (Unit: degree)

**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: degree)

**G65 H32 P#I Q#J R# k;**

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

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

**G65 H33 P#I Q#J R# K;**

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

- 19) Arctangent: # I = ATAN (# J /# K) (Unit: degree)

**G65 H34 P#I Q#J R# k;**

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

**Note 1** The unit of angular variable is degree.

**Note 2** If Q, R required are not specified in operations above, they are defaulted for zero.

**Note 3** TRUNC: rounding operation, the decimal portion is abandoned.

### 3. Transfer command

- 1) Unconditional transfer

**G65 H80 Pn; n: Sequence number**

(Example) G65 H80 P120; (To N120 block)

- 2) Conditional transfer 1 #J.EQ.# K ( = )

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

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

When # 101 = #102, it goes to N1000 block; when # 101 ≠ #102, the execution proceeds by sequence.

- 3) Conditional transfer 2 #J.NE.# K ( ≠ )

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

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

When # 101  $\neq$  #102, it goes to N1000 block; when # 101 = #102, the execution proceeds by sequence.

- 4) Conditional transfer 3 #J.GT.# K ( > )

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

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

When # 101 > #102, it goes to N1000 block; when # 101  $\leq$  #102, the execution proceeds by sequence.

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

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

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

When # 101 < #102, it goes to N1000 block; when # 101  $\geq$  #102, the execution proceeds by sequence.

- 6) Conditional transfer 5 #J.GE.# K (  $\geq$  )

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

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

When # 101  $\leq$  #102, it goes to N1000 block; when # 101 < #102, the execution proceeds by sequence.

- 7) Conditional transfer 6 #J.LE.# K (  $\leq$  )

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

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

When # 101  $\leq$  #102, it goes to N1000 block; when # 101 > #102, the execution proceeds by sequence.

**Note** The sequence number can be specified by variables. Such as G65 H81 P#100 Q#101 R#102; if the conditions are met, it goes to the block whose number is specified by #10.

#### 4. Logic AND, logic OR and logic NOT instructions

##### Example:

G65 H01 P#100 Q0;

G65 H01 P#101 Q3;

G65 H01 P#102 Q5;

G65 H11 P#100 Q#101 Q#102;

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

G65 H12 P#100 Q#101 Q#102;

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

#### 5. Examples for custom macro

##### 1 Bolt hole cycle

To drill N equal-spaced holes on the circumference of the circle with the center regarded as the basic point (X0, Y0), radius R and the initial angle (A).

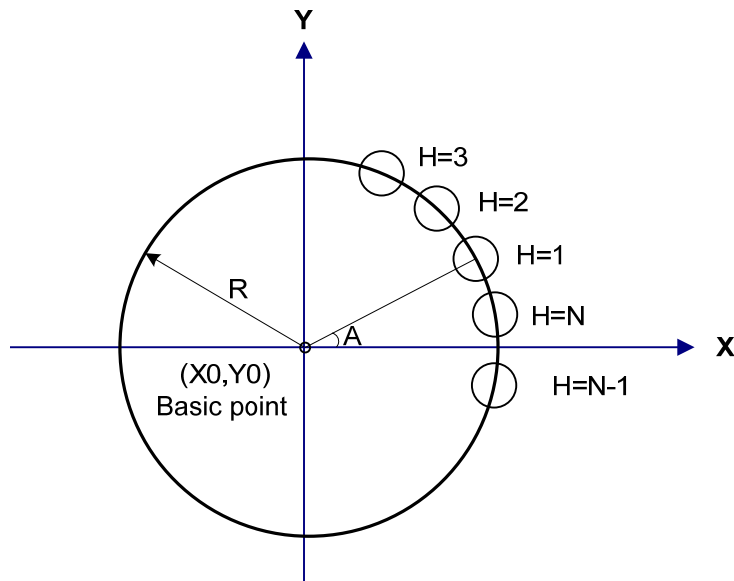


Fig. 4-7-5-1

X0, Y0 is the coordinate of the basic point in bolt hole cycle.

R: radius, A: initial angle, N: Number. Parameters above using the following variables:

#500: X coordinate value of basic point (X0)

#501: Y coordinate value of basic point (Y0)

#502: Radius (R)

#503: Initial angle (A)

#504: Number

If  $N > 0$ , for N holes in CCW direction;

If  $N < 0$ , for N holes in CW direction.

The following variables are used for the operation in macro.

#100: For the counting of the I hole machining (i)

#101: The final value of the counting(= | N | ) (IE)

#102: The angle of the i-th hole ( $\theta_i$ )

#103: X coordinate of the i-th hole ( $X_i$ )

#104: Y coordinate of the i-th hole ( $Y_i$ )

The custom macro body can be programmed as following:

O9010;

N100 G65 H01 P#100 Q#0;  $i=0$

G65 H22 P#101 Q#504;  $IE = | N |$

N200 G65 H04 P#102 Q#100 R3600 00;  $\theta_i = A + 360^\circ \times I/N$

G65 H05 P#102 Q#102 R#504;

G65 H02 P#102 Q#503 R#102;  $X_i = X_0 + R \cdot \cos(\theta_i)$

G65 H32 P#103 Q#502 R#102;

G65 H02 P#103 Q#500 R#103;  $Y_i = Y_0 + R \cdot \sin(\theta_i)$

G65 H31 P#104 Q#502 R#102;

G65 H02 P#104 Q#501 R#104;

G90 G00 X#103 Y#104; The i-th hole positioning

#### 4.7.5 Custom macro function B

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

Function	Format	Remarks
Definition	$\#i = \#j$	
Addition	$\#i = \#j + \#k$ ;	
Subtraction	$\#i = \#j - \#k$ ;	
Multiplication	$\#i = \#j * \#k$ ;	
Division	$\#i = \#j / \#k$ ;	
Sine	$\#i = \text{SIN}[\#j]$ ;	The angle is specified by degree. 90°30' indicates an angle of 90.5°.
Arcsine	$\#i = \text{ASIN}[\#j]$ ;	
Cosine	$\#i = \text{COS}[\#j]$ ;	
Arc cosine	$\#i = \text{ACOS}[\#j]$ ;	
Tangent	$\#i = \text{TAN}[\#j]$ ;	
Arc tangent	$\#i = \text{ATAN}[\#j] / [\#k]$ ;	
Square root	$\#i = \text{SQRT}[\#j]$ ;	
Absolute value	$\#i = \text{ABS}[\#j]$ ;	
Rounding-off	$\#i = \text{ROUND}[\#j]$ ;	
Rounding up to an integer	$\#i = \text{FUP}[\#j]$ ;	
Rounding down to an integer	$\#i = \text{FIX} [\#j]$ ;	
Natural logarithm	$\#i = \text{LN}[\#j]$ ;	
OR	$\#i = \#i \text{ OR } \#k$ ;	Logic operation is

Exclusive OR AND	#i = #j XOR #k; #i = #j AND #k;	executed by the binary system.
BCD to BIN Bin to BCD	#i = BIN[#j]; #i = BCD[#j];	Used for switching with PMC signal

**Explanation:****(1) Angle unit**

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

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

Ranging from -90° to 90°.

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

Variable #j can be replaced by constants

**(3) ARCCOS #i = ACOS [ #j ]**

Ranging from 180° to 0°.

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

Variable #j can be replaced by constants.

**(4) ARCTAN #i = ATAN [ #j ] / [ #k ]**

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

Ranging from 0° to 360°.

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

Variable #j can be replaced by constants.

**(5) Natural logarithm #i = LN [ #j ]**

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

Variable #j can be replaced by constants.

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

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

**(7) ROUND (rounding-off) function**

The round function rounds off at the first decimal place.

Example:

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

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

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

Example:

Suppose that #1=1.2, #2=-1.2.

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

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

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

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

**(9) The abbreviations of the arithmetic and logic instructions.**



When a function is specified in a program, the first two characters of the function name can be used to specify the function. (See table 4-7-5-1)

Example:

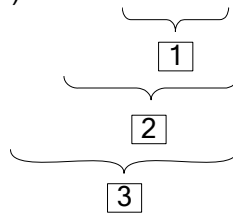
ROUND→RO

FIX→FI

#### (10) Operation sequence

- ① Function
- ② Multiplication and division operation (\* / AND)
- ③ Addition and subtraction operation (+ - OR XOR)

Example) #1 = #2 + #3 \* SIN[#4] ;



[1], [2] and [3] indicate the operation sequence.

#### (11) Restrictions

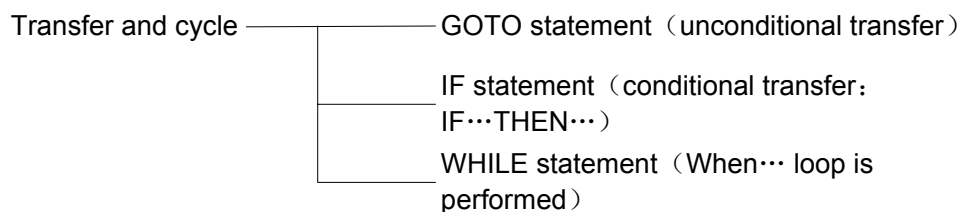
Brackets [, ] are used to enclose an expression.

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

## 2. Transfer and cycle

### 1) Transfer and loop

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



### 2) Unconditional transfer

#### ➤ GOTO statement

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

GOTOn; n: Sequence number (1~99999)

Example:

GOTO 1;

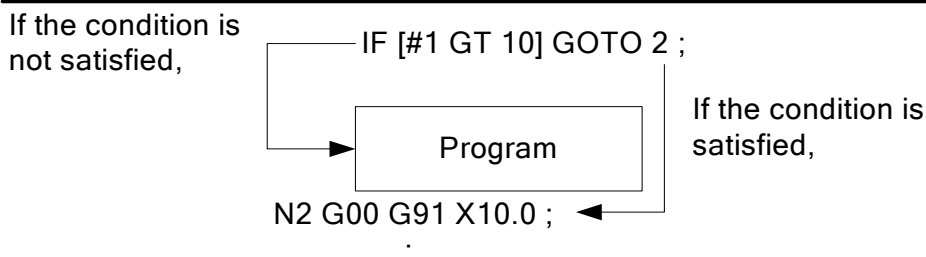
GOTO #10;

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

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

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

If the value of a variable is greater than 10, the system transfers to the block with sequence number N2.



### IF[<conditional expression>]THEN

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

If the values of #1 and #2 are the same, 0 is assigned to #3.

IF[#1 EQ #2] THEN #3=0;

### Explanation:

#### ➤ Conditional expression

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

#### ➤ Operator

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

Table 4-7-5-2 Operators

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

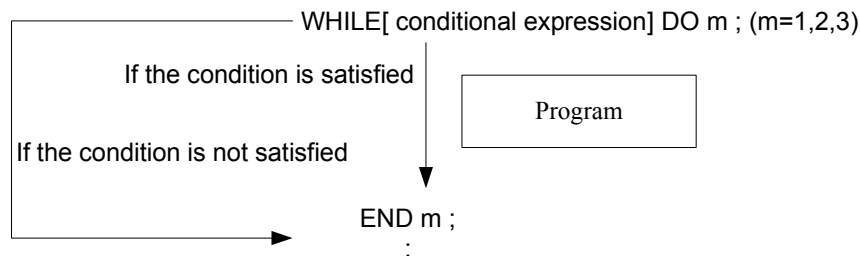
#### ➤ Typical program

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

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

#### 4) Loop (WHILE statement)

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



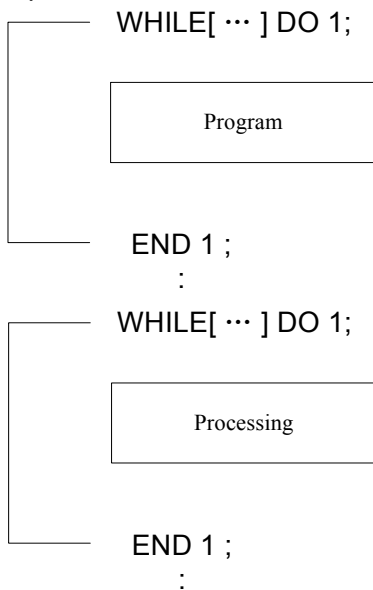
When the specified condition is satisfied, the program from DO to END is executed. Otherwise, program execution proceeds to the block after END. This kind of instruction format is applicable to IF statement. A number after DO and a number after END are the identification numbers for specifying the range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

#### Explanation:

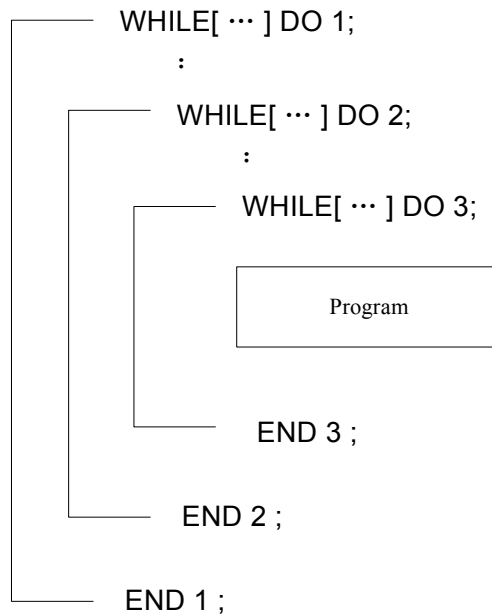
##### ➤ Nestling

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

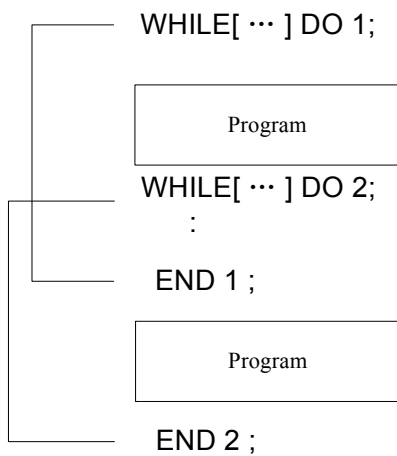
1. The identification numbers (1 to 3) can be used as many times as required.



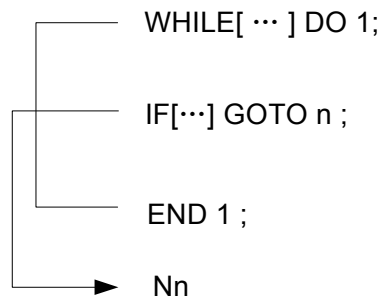
3. DO loops can be nested to 3 levels



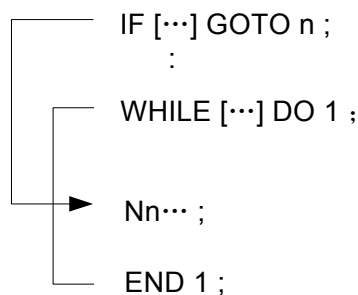
2. The ranges of DO cannot overlap



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



5. Transfer cannot enter the loop area.



### Explanation:

#### ➤ Infinite loop

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

#### ➤ Processing time

When a transfer to a sequence number in GOTO statement occurs, the sequence number is

searched for. Processing in the reverse direction is longer than the one in the forward direction. The processing time can be reduced by using WHILE statement for repetition.

➤ Undefined variables

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

➤ Typical program

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

```
O0001 ;  
#1=0;  
#2=1;  
WHILE [#1 LE 9] DO 1;  
#1=#1+#2;  
#1=#2+#1;  
END 1;  
M30;
```

**Precautions:**

- When a macro program is called by G65, and M, S, T, D and F are used for transferring variables, only positive integers can be transferred. This limitation does not apply to other letters.
- The line number N code cannot be in the same line with WHILE/DO/END, or the loop is ineffective.
- Loop and skip instructions cannot be used in DNC mode.
- A GOTO statement starts searching at the beginning of the program and skips when the first corresponding line number is retrieved. Try not to use the same N code in one program.
- When the variable number is expressed by a decimal fraction, the system will remove the decimal part with carry ignored.
- The values of local variables are retained before the main program ends. They are common to each subprogram.

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

# 5 Miscellaneous Function M code

The M codes available in this system are listed as following:

**Table 5-1**

	<b>M code</b>	<b>Function</b>
<b>M codes used by program</b>	M30	Program ends and returns to program beginning, workpieces added by 1
	M02	Program ends and returns to program beginning, workpieces added by 1
	M98	Calling subprogram
	M99	Subprogram ends and returns / execution repeated
<b>M codes controlled by PLC</b>	M00	Program dwell
	M01	Program optional dwell
	M03	Spindle CCW
	M04	Spindle CW
	M05	Spindle stop
	M06	Tool change
	M08	Cooling on
	M09	Cooling off
	M10	A axis release
	M11	A axis clamp
	M16	Tool release
	M17	Tool clamping
	M19	Spindle orientation
	M21	Tool search instruction in retraction
	M22	Tool search instruction for a new tool
	M23	Magazine to spindle instruction
	M26	Chip flushing water valve ON
	M27	Chip flushing water valve ON
	M28	Rigid tapping cancel
	M24	Magazine retraction instruction
	M29	Rigid tapping
	M35	Helical chip remover on
	M36	Helical chip remover off
	M44	Spindle blowing on
	M45	Spindle blowing off
	M50	Auto tool change start
	M51	Auto tool change finish
	M53	Tool judging after tool change
	M55	Tool judging on the spindle

When move instruction and miscellaneous function are specified in the same block, the instructions are executed in one of the following two ways:

- (1) Simultaneous execution of the move instruction and miscellaneous function instruction.
- (2) Executing miscellaneous function instructions on completion of the move instruction execution.

The selection of either sequence depends on the machine tool builder's specification. Refer to the manual by the machine builder for details.

When a numerical value is specified following address M, code signal and strobe signal are sent to the machine. The machine uses these signals to turn on or off these functions. Usually only one M code can be specified in a block. In some cases, up to three M codes can be specified in a block by bit parameter No.33#7. Some M codes can't be specified simultaneously for the restrictions of the mechanical operation. See the machine manual by the builder for the restrictions to specify multiple M codes for the same block in mechanical operation.

## 5.1 M codes controlled by PLC

If an M code controlled by PLC is in a same block with a move instruction, they are executed simultaneously.

### 5.1.1 Forward and reverse rotation instructions (M03, M04)

**Instruction:** M3 (M4) Sx x x;

**Explanation:** Viewed from the positive Z axis to negative, the spindle counterclockwise (CCW) rotation is defined as forward rotation, clockwise (CW) as reverse rotation.

The instruction of Sx x x specifies the spindle speed, it is the gear in gear mode.

Unit: r/min

When it is controlled by frequency converter, Sx x x specifies the actual speed. e.g. S1000 specifies the spindle to rotate by a speed of 1000r/min.

### 5.1.2 Spindle stop (M05)

**Instruction:** M5 When M5 is executed in auto mode, the spindle stops but the speed specified by S instruction is reserved. The deceleration at spindle stop is set by the machine builder. It is usually by energy consumption brake.

### 5.1.3 Cooling on and off (M08, M09)

**Instruction:** M8 (M9) t is used to control the the cooling pump. If the miscellaneous functions are locked in auto mode, this instruction is not executed.

### 5.1.4 A axis release and clamping (M10, M11)

**Instruction:** M10 (M11) It is used for A axis release and clamping.

### 5.1.5 Tool release and clamping (M16, M17)

Instruction: M16 (M17) It is used for tool release and clamping.

### 5.1.6 Spindle orientation (M19)

Instruction: M19 It is specified for spindle orientation which is used for tool change and positioning.

### 5.1.7 Tool search instruction (M21, M22)

Instruction: M21 It is used to search tool in retraction; M22, it is used to search a new tool for clamping.

### 5.1.8 Magazine rotation instruction (M23, M24)

Instruction: M23 It is used to rotate the tool magazine to the spindle; M24 It is used to rotate the tool magazine back.

### 5.1.9 Rigid tapping (M29)

Instruction: M29 It is used for rigid tapping.

### 5.1.10 Lubricating on and off (M32, M33)

Instruction: M32 (M33) It is used to control the lubricating pump. If the miscellaneous functions are locked in auto mode, this instruction is not executed.

### 5.1.11 Helical chip remover on and off (M35, M36)

Instruction: M35 (M36) It is used to control the helical chip remover.

### 5.1.12 Mirror image instructions (M40, M41, M42, M43)

Instructions: M40 is used to specify X axis mirror image; M41 is used to specify Y axis mirror image; M42 is used to specify Z axis mirror image; M43 is used to cancel mirror image.

### 5.1.13 Spindle blowing on and off (M44, M45)

Instruction: M44 (M45) It is used to control the spindle blowing.

### 5.1.14 Auto tool change start and end (M50, M51)

Instruction: M50 (M51) It is used to control the start and end of auto tool change.



### 5.1.15 Tool judging after tool change (M53)

Instruction: M53 It is used to judge the tool after the tool change.

## 5.2 M codes used by program

M codes used by program are classified for main program type and macro type. If the M code for program and the move instruction are in a same block, the move instruction will be executed before M code.

- Note: 1. M00、M01、M02、M06、M30、M98、M99 code can't be specified with other M code together, or an alarm is issued by system. When they share a same block, the other non-M instructions are executed before M codes.
2. This kind of M codes include the codes that are to make CNC to send M code to the machine tool and make the CNC to perform the internal operation, e.g. the M code to make the block prereading inactive. Moreover, the codes to make CNC to send M code to machine without performing the internal operation may share the same block.

### 5.2.1 Program end and return (M30, M02)

When M30 (M02) in the program is executed in auto mode, the auto mode is cancelled. The blocks following them are not executed and the spindle and cooling stops, the workpiece machined added by 1. M30 can be set by bit parameter NO.33#4 to return to program beginning; M02 can be set by bit parameter NO.33#2 to return to program beginning. If M02 or M30 is at the end of the subprogram, the control returns to the program that call the subprogram and goes on executing the following block.

### 5.2.2 Program dwell (M00)

In Auto running, automatic operation pauses after a block containing M00 is executed. And the previous modal information will be saved. The automatic operation can be continued by pressing cycle start key, which is equivalent to pressing down feed hold key.

### 5.2.3 Program optional stop (M01)

Automatic operation is stopped optionally after a block containing M01 is executed. If the "optional stop" on-off is set for ON, M01 is equivalent to M00; if the "optional stop" on-off is set for OFF, M01 is ineffective. See *OPERATION MANUAL* for its operation.

### 5.2.4 Subprogram calling (M98)

This code is used to call a subprogram in the main program.

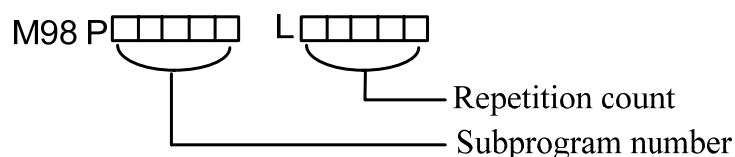


Fig. 5-2-4-1

### 5.2.5 Program end and return (M99)

1. In auto mode, if M99 is used at the end of the main program, the control returns to the program beginning to continue automatic operation after the block containing M99 is executed. The blocks followed are not to be executed, and the number of the workpieces machined is not accumulated.
2. If M99 is used at the end of a subprogram, the control returns to the the main program block containing M98 after the block containing M99 is executed.
3. In DNC mode, if M99 is regarded as M30, the cursor will stop at the end of the program.

## 6 S codes for Spindle Function

The code signal converted to analog signal by S code and the numerical value followed is sent to the machine to control the machine spindle.

S is a modal value.

### 6.1 Spindle analog control

When the bit parameter NO.1#2 SPT=0, the spindle speed is controlled by analog voltage specified by address S and the numerical value followed. See details about it in *OPERATION* manual.

**Format:** S\_

**Explanation:**

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

### 6.2 Spindle switch volume control

When the bit parameter NO.1#2 SPT=1, the spindle speed is controlled by switch volume specified by address S and two digits number followed.

Four gears are available in this system as spindle speed is controlled by switch volume. See details on the correspondence of S code and the spindle speed as well as the gears in the manual by machine builder.

**Format:** S01 (S1) ;  
S02 (S2) ;  
S03 (S3) ;

**Explanation:**

1. There are 8 gears in the software at present, and 3 gears in the ladder diagram. When S codes beyond the codes above are specified, the system displays "Miscellaneous function being executed".
2. If a four-digit number is specified behind S, the latter two digits are effective.

### 6.3 Constant surface speed control (G96/G97)

**Format:**

Constant surface speed control instruction

G96 S\_ Surface speed (m/min or inch/min)

Constant surface speed control cancel instruction

G97 S\_ Spindle speed (r/min)

Constant surface speed controlled axis instruction

G96 Pn P1 X axis; P2 Y axis; P3 Z axis; P4 4<sup>th</sup> axis

Max. spindle speed clamping

G92 S\_ S specifies the max. spindle speed (r/min)

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

**Explanation:**

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

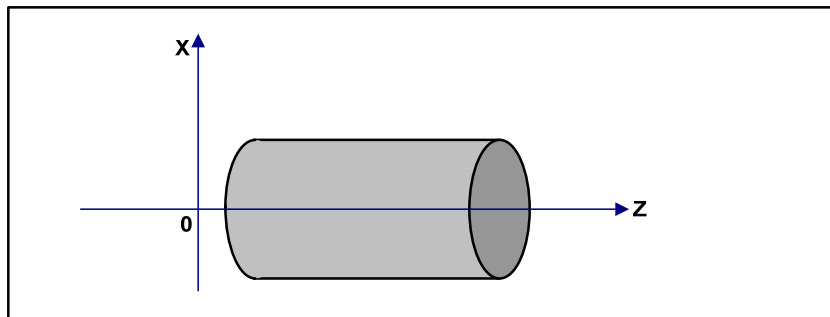


Fig. 6-3-1 Workpiece coordinate system for constant surface speed control

- 4 When constant surface speed control is applied, if a spindle speed higher than the value specified in G 92 S\_, it is clamped at the maximum spindle speed. When the power is switched on, the maximum spindle speed is not yet set, the S in G96 is regarded as zero till M3 or M4 appears in program.

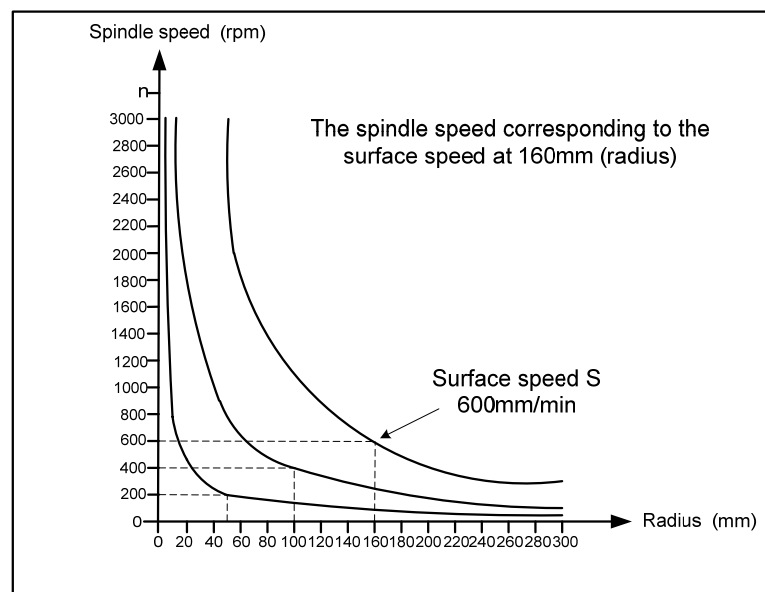


Fig. 6-3-2 Relations of workpiece radius, spindle speed and surface speed

- 5 Surface speed is specified in G96 mode

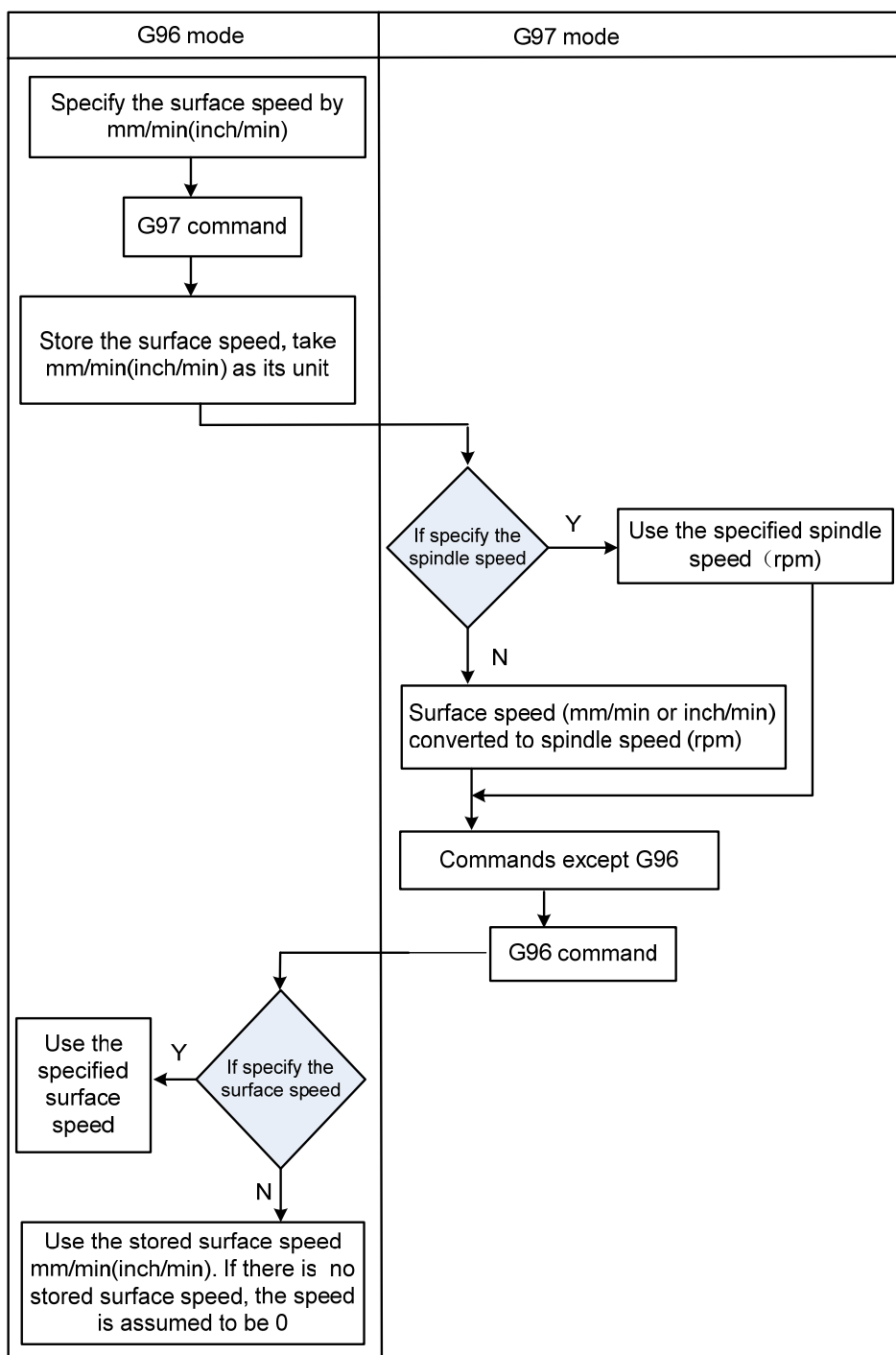


Fig. 6-3-3

**Restriction**

- 1 Because the response problem in the servo system may not be considered when the spindle speed changes, while the constant surface speed is still effective during threading, so it is recommended to cancel the constant surface speed by G97 before threading.
- 2 In a traverse block specified by G00, the constant surface speed control is not made by calculating the surface speed by a transient change of the tool position, but is made by calculating the surface speed based on the position at the end point of the traverse block, on the condition that cutting is not performed during traverse. Therefore, the constant surface cutting speed is not needed.

# 7

## Feed Functions: F code

The feed functions are used to control the feedrate of the tool. They are used as following:

### 7.1 Traverse

G00 instruction is used for rapid positioning. And the traverse speed can be set by number parameter P88~P92. Override can be applied to a traverse speed by the OVERRIDE adjusting

keys on the operator panel as follows:



**Fig. 7-1-1 rapid traverse federate key**

In which, F0 is set by date parameter P93.

The acceleration of rapid positioning (G0) can be set by number parameter P105~P124. It can be properly set depending on the machine and the motor response.

**Note** A feedrate F instruction is ineffective even it is specified in a block containing G00 and the system performs positioning at the speed specified by G0.

### 7.2 Cutting feedrate

Feedrate of linear interpolation(G01), circular interpolation(G02, G03) are specified with the numbers after F code. The unit of it is mm/min. The tool moves by the feedrate programmed. Override can be applied to feedrate using the override key on the operator panel(Override range: 0 %~150%).

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

In non-forecast mode, the maximum cutting feedrate is set by number parameter P94 and in forecast mode, it is set by number parameter P96. If the feedrate is more than that, use the feedrate set by that parameter.

In non-forecast mode, the minimum cutting feedrate is set by number parameter P95 and in forecast mode, it is set by parameter P97. If the feedrate is less than that, use the feedrate set by that parameter.

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

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

- A).** Feed per minute (G94): it is used to specify the feed amount per minute after F code.
- B).** Feed per revolution (G95): it is used to specify the feed amount per revolution after F code.

### 7.2.1 Feed per minute (G94)

**Format:** G94 F\_

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

**Explanation:**

- 1、 After specifying G94 (in feed per minute mode), the feed amount of the tool per minute is directly specified by a number after F.
- 2、 G94 is a modal code. Once specified, it is effective till G95 is specified. The default at power-on is feed per minute mode. The default cutting feedrate is set by data parameter P87.
- 3、 An override from 0% to 150% can be applied to feed per minute with the override key on the operator panel.

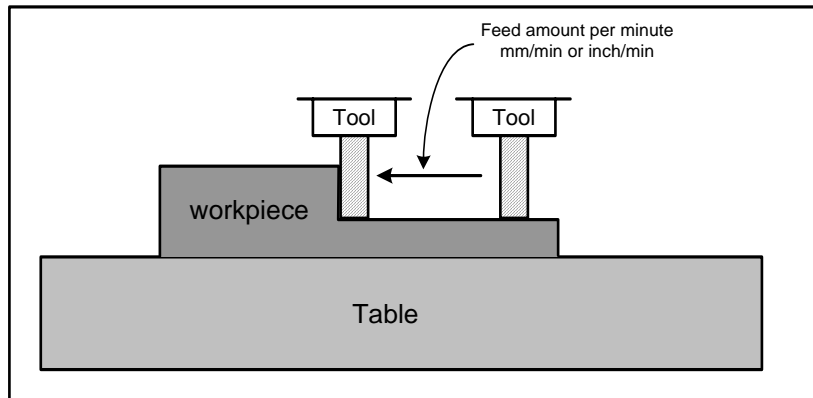


Fig. 7-2-1-1 Feed per minute

**Restriction**

Feed per minute mode can't be applied for some instructions such as threading.

### 7.2.2 Feed per revolution (G95)

**Format:** G95 F\_

**Function:** It specifies the tool feed amount in a revolution. Unit: mm/r or inch/r

**Explanation:**

- 1 The function can be applied only the encoder is fixed on the machine
- 2 After specifying G95 (feed per revolution mode), the feed amount of the tool per revolution is directly specified by a number after F.
- 3 G95 is a modal code. Once specified, it is effective till G94 is specified. The feedrate per revolution is defaulted to be 0 when initializing.
- 4 An override from 0% to 150% can be applied to feed per revolution with the override key on the operator panel.

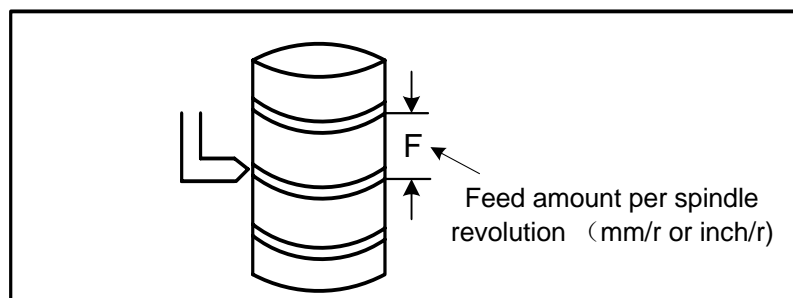


Fig. 7-2-2-1 Feed per revolution

**Note** Feedrate fluctuation may occur if the spindle speed is low. The slower the spindle rotates, the more frequently the feedrate fluctuation occurs.

### 7.3 Tangential speed control

Usually the cutting feed of the tool is made by controlling the speed along the tangent of the contour path to reach a value specified.

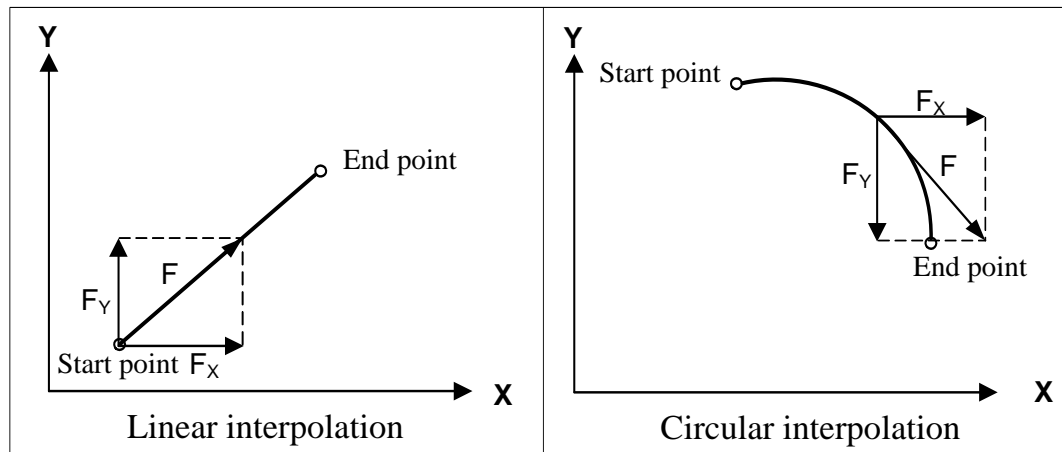


Fig. 7-3-1

- $F$ : The speed along the tangent :  $F = \sqrt{F_x^2 + F_y^2 + F_z^2}$   
 $F_x$ : The speed along X axis  
 $F_y$ : The speed along Y axis  
 $F_z$ : The speed along Z axis

### 7.4 Feedrate override keys

The feedrate in JOG mode and AUTO mode can be overridden by the override keys on the operator panel. The override range from 0~150%(16 gears with 10% per gear). In AUTO mode, if the feedrate override is adjusted for zero, the feeding is stopped by the system with 0 cutting override displayed. The execution is continued if the override is readjusted.

### 7.5 Auto acceleration/deceleration

The stable start and stop can be done by auto acceleration/deceleration at the beginning and the end of the moving controlled by the system motor. And the auto acceleration/deceleration can also be done when the moving speed is changed, so the speed can be changed steadily. Therefore the acceleration/deceleration needn't to be considered for programming.

Rapid traverse: Pre-acceleration/deceleration ( 0 : linear type ; 1 : S type ) hind acceleration/deceleration (0: linear type; 1: exponential type)

Cutting feed: Pre-acceleration/deceleration (0: linear type; 1: S type) hind acceleration/deceleration (0: linear type; 1: exponential type)

JOG feed: Hind acceleration/deceleration (0: linear type; 1: exponential type)

(Set the universal time constant for each axis by parameters)



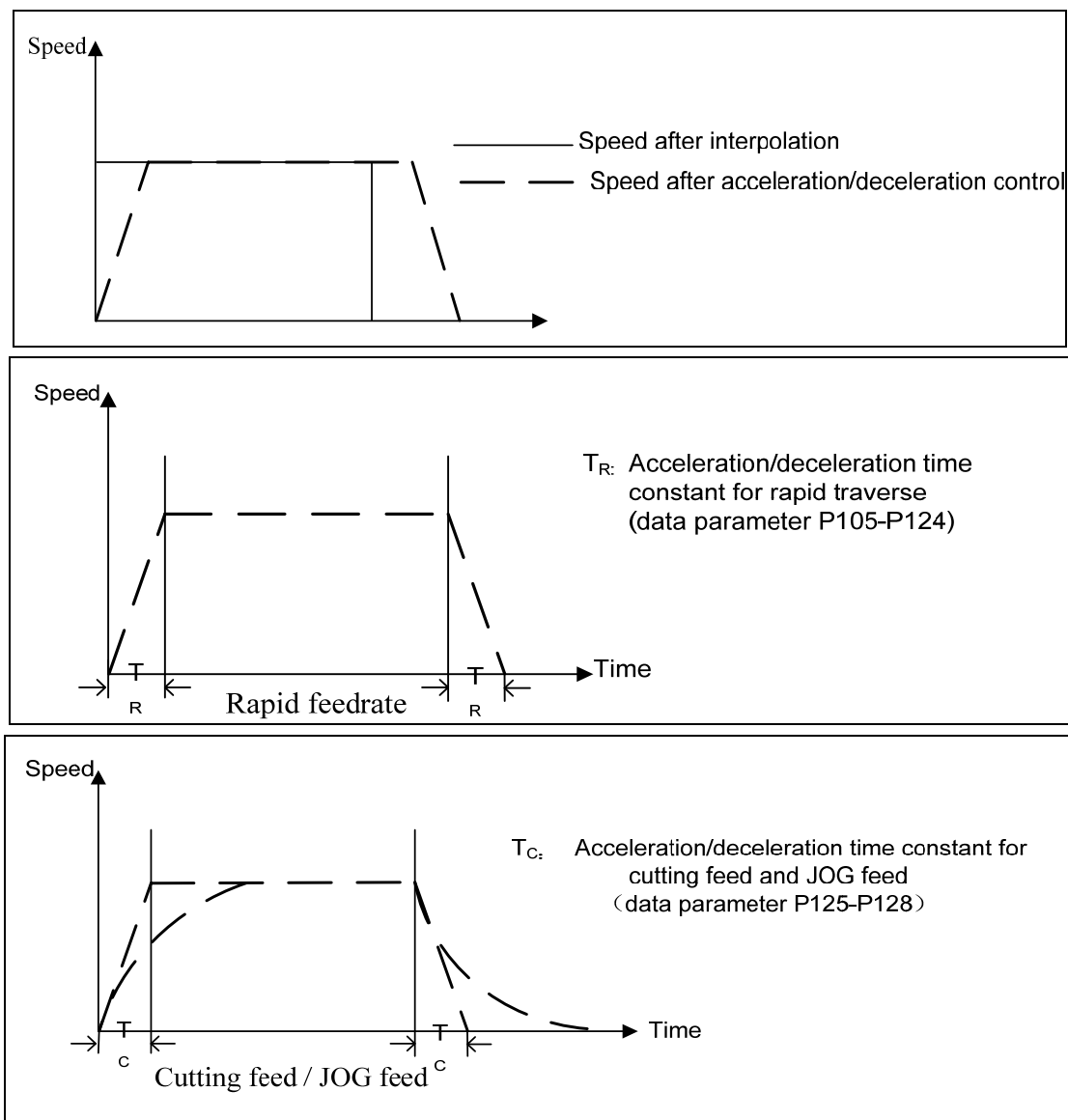


Fig.7-5-1

## 7.6 Acceleration/deceleration for corner of a block

For example: if only Y axis moves in a block, and X axis moves in the block following, the tool path is as following during the Y axis deceleration and the X axis acceleration:

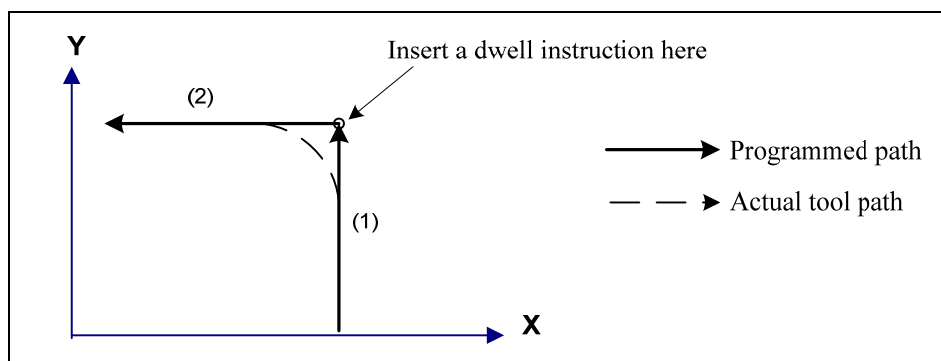


Fig. 7-6-1

If the dwell (exact stop) instruction is inserted, the tool moves along the real line as in above figure by the program. Otherwise the bigger the cutting feedrate is, or the longer the time constant of the acceleration/deceleration, the bigger the arc at the corner is. For circular instruction, the actual arc radius of the tool path is smaller than the radius given by the program. Under the condition allowed by mechanical system, reduce the time constant of the acceleration/deceleration as far as possible to decrease the error at the corner.

# 8

## Tool Function

### 8.1 Tool function

By specifying a numerical value (up to 128) following address T, tools can be selected on the machine.

Only one T code can be specified in a block in principle, if the codes in a same group are specified in a block, no alarm is issued, and then the latter T code is valid. Refer to the machine builder's manual for the number with address T and the corresponding machine operation of T code.

When a move instruction and a T code are specified in a same block, the instructions are executed in the following two ways:

- 1 Simultaneous execution of the move and T instructions.
- 2 Executing T instruction upon completion of the move instruction.

The selection of either 1 or 2 depends on the machine builder's specifications. Refer to the machine builder's manual for details.

When the T code and tool change instruction share a block, the T code is executed before tool change. If they are not in a same block, M06 executes the T code specified by last program.

Example:

```
O00010;  
N10 T2M6;           Spindle tool number is T2  
N20 M6T3;           Spindle tool number is T3  
N30 T4;             Spindle tool number is T3  
N40 M6;             Spindle tool number is T4  
N50 T5;             Spindle tool number is T4  
N60 M30  
%
```

The spindle tool number is T4 after the tool change operation

~

# III OPERATION

## Operator Panel

### 1.1 Panel layout

The 990MA machine center has an integrated operator panel, which is comprised by LCD area, edit area, and machine control area. The layout of it is shown as following:

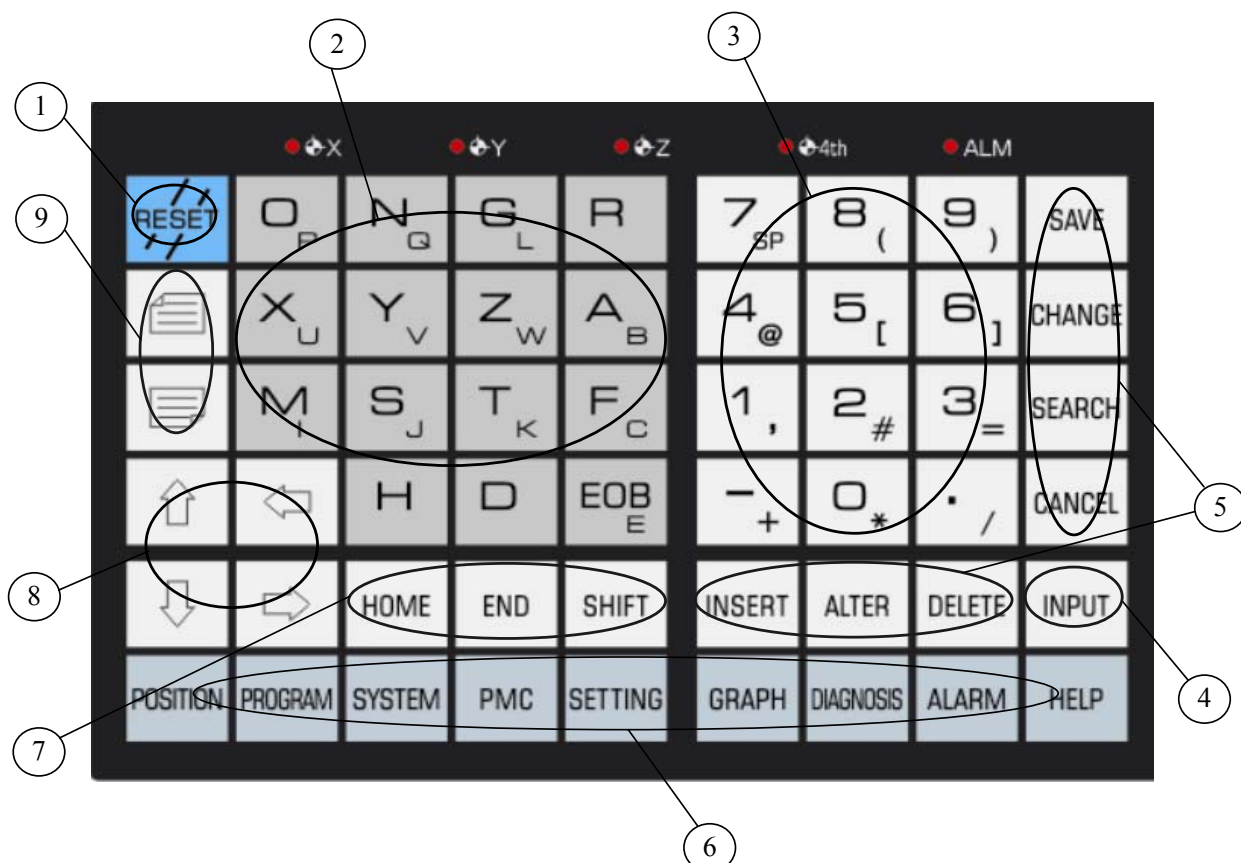


## 1.2 Explanation of the panel function

### 1.2.1 LCD area

The display area of this system is applied with a 7 inch chromatic LCD that has 480×320 resolution.

### 1.2.2 Edit area



The edit keyboard divided for 9 small areas is explained as follows:

Number	Name	Explanation
1	Reset key	Press this key to reset the system, feed and stop output
2	Address key	Press these keys to input address
3	Numerical keys	Press these keys to input numerical numbers
4	Input key	For Inputting numerical values, addresses or data into the buffer area; confirming the operation result
5	Editing and Search key	Use for block inser, modify, delete, cancel, save and chage in program editing; For searching data and addresses to view and modify
6	Screen operation keys	Press any key of them to enter the corresponding interface (introduced below)
7	Cursor positioning keys	Press these keys to move the cursor to the the beginning or end of the line or that of the program.
8	Cursor Moving keys	Press these keys to move the cursor in four directions
9	Page keys	Press these keys to change the page on the screen in the same display mode

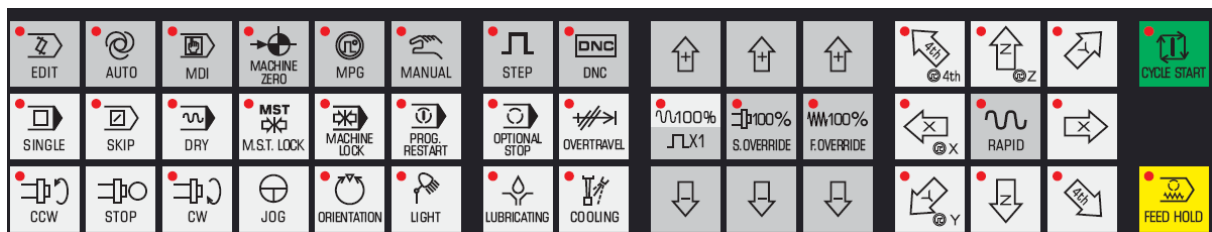
### 1.2.3 Screen operation keys

8 page display keys are laid out in this operator panel, which are as following:













Position page	Program page	System page	PMC page	Setting page	Graphics page	Diagnosis page	Alarm page	Help page

Name	Explanation	Remark
Position page	Press this key to enter position page	Relative coordinate, absolute coordinate of current point, comprehensive display, program monitoring display by changing soft keys
Program page	Press this key to enter program page	Program,MDI, current/mode, program directory display by changing soft keys,
System page	Press this key to enter system page	bit parameter, data parameter, common variables, system variables, offset and screw pitch display by changing soft keys to view and modify the parameters, variables, tool offset and pitch compensation.
PLC page	Press this key to enter PLC page	PMC counter address, Keep type relay address and data list address display by pressing soft keys to view and modify.
Setting page	Press this key to enter program page	setting, parameter on-off, coordinate, panel, servo system, data and password setting display by changing soft keys
Graphic page	Press this key to enter setting page	For graphic parameter, graphic display page and the graphic center, dimension, ratio and display page setting by changing soft keys
Diagnosis page	Press this key to enter diagnosis page	To view the I/O signals in the system, PMC → CNC, machine, PMc→ machine by changing soft keys
Alarm page	Press this key to enter alarm page	To view alarm display pages ,history alarm display pages and operation display pagesby changing soft keys
Help page	Press this key to enter help page (it is invalid temporarily )	Help message about the system can be viewed in this page by switching corresponding soft keys.









## 1.2.4 Control area





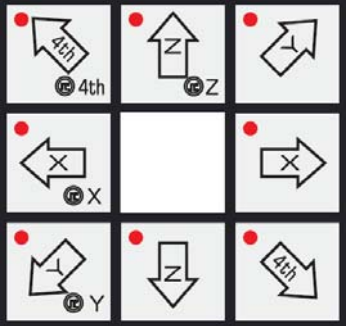





Keys	Name	Function explanation	Remarks and operation explanation
	dit key	To enter Edit mode	Switching to Edit mode in Auto mode, system slowing down to stop if current block is executed

	Auto mode key	To enter Auto mode	In this mode, internal memory program is selected
	MDI mode key	To enter MDI mode	Switching to MDI mode in Auto mode, system slowing down to stop if current block is executed
	Machine zero mode key	To enter Machine zero mode	Switching to Machine zero mode in Auto mode, system immediately slowing down to stop
	MPG mode key	To enter MPG mode	Switching to MPG mode in Auto mode, system immediately slowing down to stop
	MANUAL mode key	To enter MANUAL mode	Switch to MANUAL mode in Auto mode, system immediately slowing down to stop
	Step mode key	To enter Step mode	Switching to Step mode in Auto mode, system immediately slowing down to stop
	DNC mode key	To enter DNC mode	Switching to DNC mode in Auto mode, system slowing down to stop if current block is executed
	Single Block key	For switching of program block/blocks, if it is on, the indicator lights up	Auto mode, MDI mode, DNC mode
	Block Skip key	For block preceding with “/” sign skipping, if it is set for on, the indicator lights up	Auto mode, MDI mode, DNC mode
	Dry Run key	If it is effective, the indicator lights up	Auto mode, MDI mode, DNC mode
	M.S.T. Lock key	If the miscellaneous function is on, its indicator lights up and M,S,T functions are ineffective	Auto mode, MDI mode, DNC mode
	Machine Lock key	If it is on, its indicator lights up, and the axis output is ineffective.	Auto mode, MDI mode, Machine zero, MPG mode, Step mode, JOG mode, DNC mode



	Program Restart key	Cursor moving to the beginning of the starting block to restart the machine, also for rapid program check	Auto mode, MDI mode, DNC mode
	Optional Stop key	For stop of the program with "M01"	Auto mode, MDI mode, DNC mode
	Overtravel release key	An alarm occurs if the hard limit is reached. Press this key with its indicator lighting up to move the machine reversely till the indicator goes off.	MANUAL mode, MGP mode
	Spindle control keys	Spindle CCW Spindle stop Spindle CW	MPG mode, Step mode, JOG mode
	Spindle override keys	Spindle speed adjusting (spindle speed analog control effective)	Any mode
	Spindle JOG key	Spindle JOG on/off	JOG mode
	Spindle Orientation key	Spindle orientation on/off	JOG mode
	Rapid override keys	For rapid override, selection	Auto mode, MDI mode, Machine zero, MPG mode, Step mode, JOG mode, DNC mode

	Feedrate Override keys	For adjustment of the feedrate	Auto mode, MDI mode, Edit mode, Machine zero, MPG mode, Step mode, JOG mode, DNC mode
	Lubricating key	For machine lubrication on/off	Any mode
	Cooling key	For coolant on/off	Any mode
	Working Light key	For machine working light on/off	Any mode
	Manual Feed key	X, Y, and Z axis positive / negative move in JOG and Step mode, the positive of the axis is MPG axis	Machine zero, Step mode, JOG mode, MPG mode
	Rapid key	For rapid on/off	Any mode
	Cycle Start key	Auto running begins by pressing this key	Auto mode, MDI mode, DNC mode
	Feed Hold key	Auto running stops by pressing this key	Auto mode, MDI mode, DNC mode

**Note:** A block with more than 1 “/” sign at its beginning is skipped by the system even if the skip function is OFF.

## 2 System Power On/Off and Safety Operations

### 2.1 System power on

Before this GSK990MA is powered on, ensure that:

- 1 The machine is normal.
- 2 The voltage of the power supply conforms to the requirement of the machine.
- 3 The wiring is correct and reliable.

The current position (relative) is displayed after system normality check and initiation are finished.

CURRENT POS (RELATIVE)		000002 N0120
<b>O0002</b>	<b>N00000</b>	PRG SPEED: 0
<b>X</b>	<b>16.000</b>	ACT SPEED: 0
<b>Y</b>	<b>16.000</b>	FEED OVRD: 100%
<b>Z</b>	<b>56.000</b>	RAPID OVRD: 100%
		OFFSET: H0000 D0000
		PRT CNT: 0000/0000
		RUN TIME: 000: 00: 00
		CURRENT: 10:06:00
		G00 G17 G54 G21 G40 G49
		Sx 100% S0000 T0100
		MDI

Fig. 2-1-1

### 2.2 System power off

Before power is off, ensure that:

1. The X, Y, Z axis of the CNC is at halt;
2. Miscellaneous functions(spindle, pump etc.) are off.
3. CNC power is cut off prior to cutting off machine power.

Check the following items before power-off:

1. The LED indicating the cycle start on the operator panel is off.
2. All the movable parts of the CNC machine tool is at halt.
3. Press POWER OFF button to turn off the power.

#### Cut off power at emergency

The power should be cut off immediately to prevent from incident in emergency situation during the machine running. And the zero return and tool setting should be performed again because of the error between the system coordinate and actual coordinate of the position after power-off.

**Note:** Refer to the machine builder's manual for cutting off the machine power.

## 2.3 Safety operations

### 2.3.1 Reset operation



The system is in reset mode after pressing key:

1. All axes movement stops;
2. The M functions are ineffective;
3. Whether to save G codes or not after modifying bit parameters **NO:35#1**~**NO:35#7** and **NO:36#0**~**NO:36#7** and resetting;
4. Whether to clear F, H, D codes or not after modifying bit parameters **NO:34#7** and resetting;
5. Whether to delete the program or not after modifying bit parameters **NO:28#7** and resetting in MDI mode;
6. Whether to cancel local coordinate or not after modifying bit parameters **NO:10#3** and resetting.
7. In non-Edit mode, whether the cursor returns to the beginning of the program after resetting is determined by bit parameter **NO:10#7**;
8. Whether macro local variables #1~#50 are cleared after resetting is determined by bit parameter **NO:52#7**;
9. Whether macro common variables #100~#199 are cleared after resetting is determined by bit parameter **NO:52#6**;
10. Resetting can be used during abnormal system output and coordinate axis action.

### 2.3.2 Emergency stop

If the emergency button is pressed during machine running, the system enters into emergency status and the machining movement is stopped immediately. All the outputs such as the spindle running, coolant are also cut off. If the emergency button is released (varying by machine builders, usually the button bumps up by rotating it left-handedly), the emergency is cancelled.

**Note 1 Ensure the cause of the fault is eliminated before the emergency is cancelled.**

**Note 2 Perform the reference point return operation to ensure the position coordinate after the emergency is cancelled.**

The common emergency is a normal-closed signal. When the trigger point is broken off, the system enters into emergency status and the machine stops immediately. The wiring of the emergency signal circuit is as following:

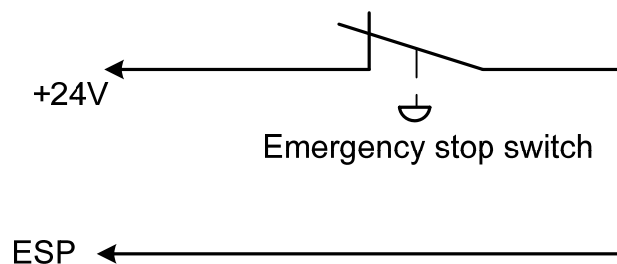


Fig. 2-3-2-1

### 2.3.3 Feed hold



The **FEED HOLD** key can be pressed during the machine running to make the running to dwell. But in rigid tapping, cycle running, the machine dwells till current instruction is executed.

### 2.4 Cycle start and feed hold



The **CYCLE START** and **FEED HOLD** keys are used for the program start and dwell operations in Auto mode, MDI mode and DNC modes. The external start and dwell are set by bit parameter **NO:59#7**, and they can also be set by modifying the address K5.1 of the PLC. These two methods are equivalent.

### 2.5 Overtravel protection

Overtravel protection must be employed to prevent the damage to the machine due to the overtravel of the feed axis.

#### 2.5.1 Hardware overtravel protection

The overtravel limit switches are fixed at the positive and negative maximum stroke of the machine X, Y, Z axis respectively. If the overtravel occurs, the moving axis slows down and stops after it touches the limit switch. And the overtravel alarm is issued.

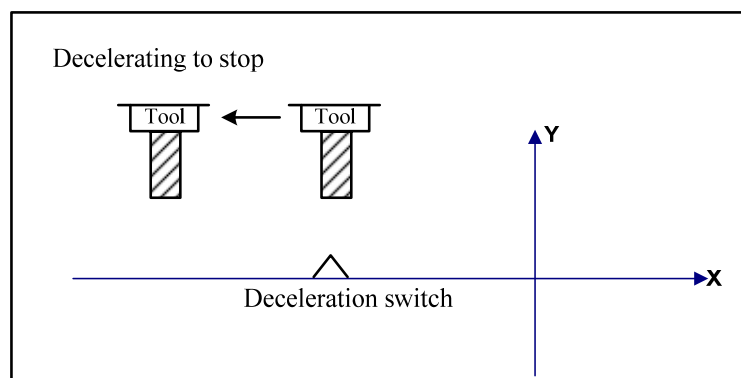


Fig. 2-5-1-1

#### Explanation:

##### Overtravel in Auto mode

In Auto mode, if the tool contacts the stroke limit switch during moving along an axis, all the axes movement are slowed down to stop with the overtravel alarm being issued. The program execution is stopped at the block where the overtravel occurs.

##### Overtravel in JOG mode

In JOG mode, any axis contacts the stroke limit switch, all axes will slow down immediately and stop.

### 2.5.2 Software overtravel protection

The software strokes of the machine are set by the number parameters **P66~P75**, referring to machine coordinate values. Overtravel alarm occurs if the machine position (coordinate) exceeds the setting software stroke. The alarm issued before or after (0: before, 1: after) overtravel for software limit overtravel is set by bit parameter **NO:11#7**. After the overtravel alarm, move the axis reversely in JOG mode, the alarm will be cancelled after the axis is moved out of the overtravel range.

### 2.5.3 Release of the overtravel alarm

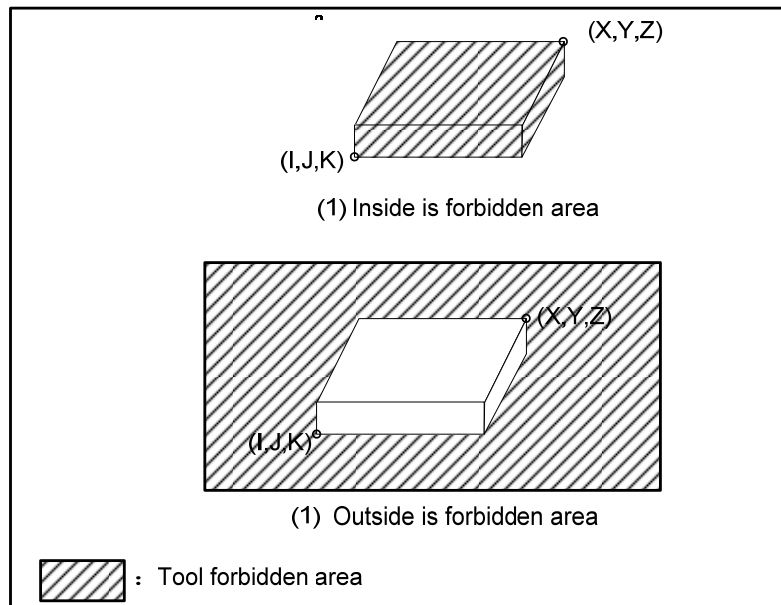
The method for releasing hardware overtravel alarm is: in JOG mode or MPG mode, press the



key on the panel, then move the axis out reversely (for positive overtravel, move negatively; for negative overtravel, move positively).

## 2.6 Stroke detection

By storage stroke detection 1 and 2, the system can specify 3 areas that the tool can't enter.



**Fig. 2-6-1 Stroke detection**

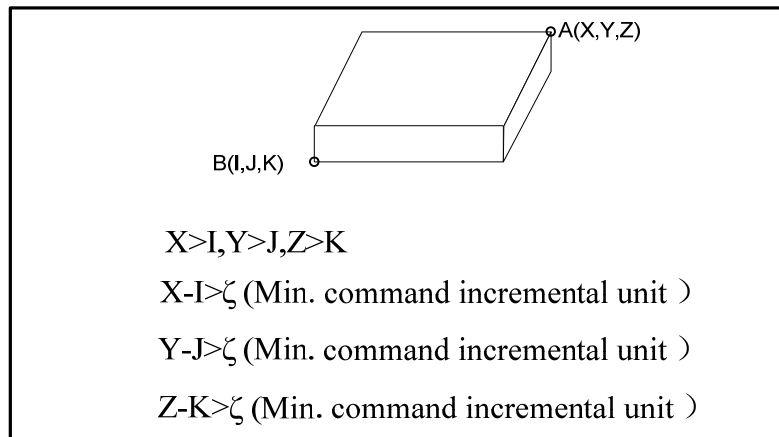
When the tool goes beyond the stroke, alarm is issued and the machine decelerates and stops.

When the tool enters the forbidden area with alarm issued, the tool may move in the reverse direction that the tool enters.

#### Explanation:

1. Storage stroke detection 1: its boundary is set by number parameter P66~P75, the outside of this area is forbidden area, which is usually set for the machine maximum travel by the machine builder.
2. Storage stroke detection 2: its boundary is set by number parameter P76~P85 or program instructions, the inside or outside of this area can be set for a forbidden area, which is set by bit parameter **NO:11#0**.

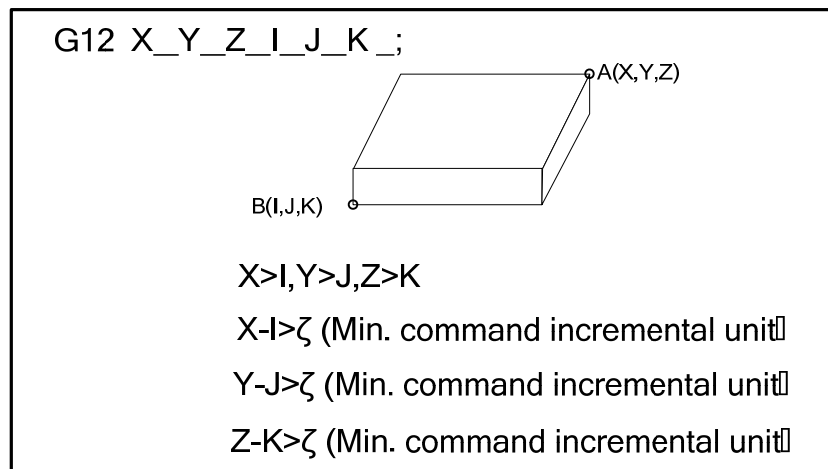
- 1.) When the forbidden area is set by parameters: the A and B points in the following figure must be set.



**Fig. 2-6-2 To use parameters set up or change forbidden area**

When the forbidden area is set by number parameter P76~P85, the distance (output increment) in the machine coordinate system must be given by a min. command incremental unit via data.

- 2.) When the forbidden area is specified by program instructions: by G12 it forbids the tool to enter forbidden area; by G13 it allows the tool to enter the forbidden area. Each G12 or G13 code must be specified by a single block in program. The following commands are used for setting up or changing the forbidden area.



**Fig.2-6-3 To set up or change forbidden area by program**

If the distance (input increment) in the machine coordinate system is specified by min. input incremental unit via G12, the programming data will be converted to the value of min. command unit by min. incremental unit. And the value will be saved in parameter P73~P83.

Example 1: The inside is the forbidden area (bit parameter **NO:11#0=0**):

N1 G12 X50 Y40 Z30 I20 J10 K15;	Setting point A (50, 40, 30) and point B (20, 10, 15) for the tool forbidden area
N2 G01 X30 Y30 Z20;	Linear interpolation to (30, 30, 20)
N3 G13;	Cancelling stored stroke check
N4 G01 X50;	

Example 2: The outside is the forbidden area (bit parameter **NO: 11#0=1**):

N1 G12 X50 Y40 Z30 I20 J10 K15;      Setting point A (50, 40, 30) and point  
B (20, 10, 15) for the tool forbidden  
area  
N2 G01 X10 Y-10 Z-10;      Linear interpolation to (10, -10, -10)  
N3 G13;      Cancelling the stored stroke check  
N4 G01 X50;

3. Detection point in the forbidden area: prior to the programming for forbidden area, please confirm the detection point (the top of the tool nose or tool collet). As is shown in Fig.2-6-4, if the detection point is A(tool nose), the distance "a" should be used as the data for storage detection; if the detection point is B(tool collet), the distance "b" should be used as the data for storage detection. When the detection point is A (tool nose), and the tool length varies with the tool, the forbidden area should be set up according to the longest tool, as such may ensure the safe running.

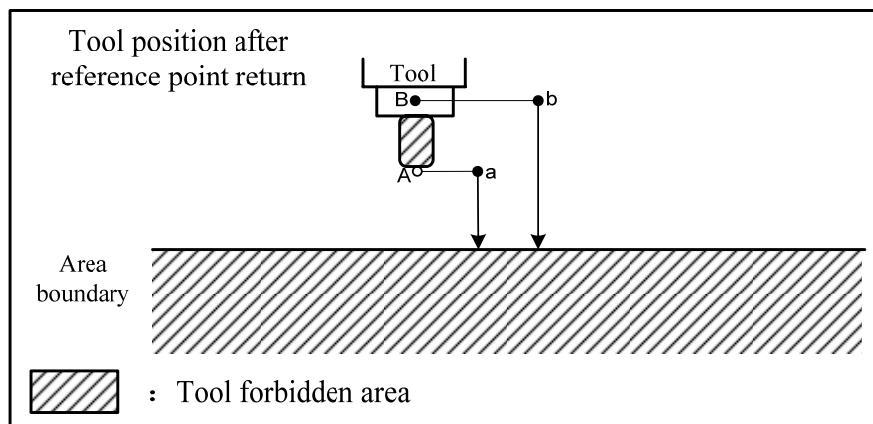


Fig. 2-6-4 To setup forbidden area

4. The overlapping of tool forbidden areas: The forbidden areas can be set by overlapping, as is shown in following figure:

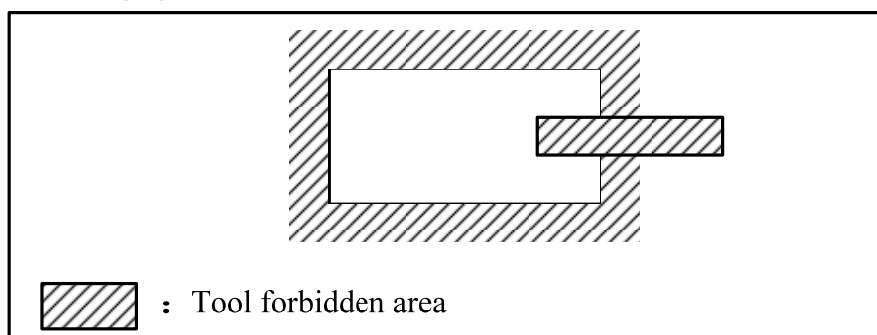


Fig. 2-6-5 The overlapped forbidden areas

Unnecessary limit should be set beyond the machine stroke.

5. Forbidden area to be effective: the forbidden area boundary takes effective immediately after the manual reference point return or auto reference point return by G28 instruction at power on. If the reference point is within the forbidden area at power on, an alarm will be issued (effective only for G12 of storage stroke 2).
6. Alarm cancellation: if the tool enters the forbidden area with an alarm being issued, it can only move reversely.  
To cancel alarm, move the tool reversely till it exits the forbidden area and the system resets. If



the alarm is cancelled, the tool can move forward or backward freely.

7. G13 changed by G12 in forbidden area: the following results may occur if G13 changed by G12 in forbidden area:
8. Whether the stroke check is performed is set by bit parameter NO:10#1. When bit parameter NO:10#1=0, the stroke check is not performed before movement; when bit parameter NO:10#1=1, the stroke check is performed before movement.


# 3

## Interface Display and Data Modification and setting

### 3.1 Position display

#### 3.1.1 Four types of position display



Press  key to enter the position page that includes four types: **【REL】**, **【ABS】**, **【All】**, **【MONI】**. They can be viewed by the corresponding soft keys, as is shown in the following:

- 1) Relative mode: It displays the coordinate of the current tool in relative coordinate system, see Fig.3-1-1-1:

ACT POS (RELATIVE)		000002 N0120
<b>O0002</b>	<b>N00000</b>	PRG SPEED: 0
<b>X</b>	<b>16.000</b>	ACT SPEED: 0
<b>Y</b>	<b>16.000</b>	FEED OVRD: 100%
<b>Z</b>	<b>56.000</b>	RAPID OVRD: 100%
		OFFSET: H0000 D0000
		PRT CNT: 0000/0000
		RUN TIME: 000: 00: 00
		CURRENT TIME: 10:06:00
		G00 G17 G54 G21 G40 G49
		Sx 100% S0000 T0100
		MDI

Fig. 3-1-1-1

- 2) Absolute mode: It displays the coordinate of the current tool in absolute coordinate, see Fig.3-1-1-2

ACT POS (ABSOLUTE)		000002 N0120
<b>O0002</b>	<b>N00000</b>	PRG SPEED: 0
<b>X</b>	<b>116.000</b>	ACT SPEED: 0
<b>Y</b>	<b>116.000</b>	FEED OVRD: 100%
<b>Z</b>	<b>156.000</b>	RAPID OVRD: 100%
		OFFSET: H0000 D0000
		PRT CNT: 0000/0000
		RUN TIME: 000: 00: 00
		CURRENT TIME: 10:06:00
		G00 G17 G54 G21 G40 G49
		Sx 100% S0000 T0100
		MDI

Fig. 3-1-1-2

- 3) All mode

in **【ALL】** interface, the coordinates in the following coordinate system can be displayed together:

- (A) The position in relative coordinate system;
- (B) The position in absolute coordinate system;
- (C) The position in machine coordinate system;
- (D) The offset amount in Handle interruption (displacement);
- (E) Subspeed;
- (F) Remaining distance (only displayed in Auto, MDI, DNC mode)

The display is as follows (Fig.3-1-1-3) :

ACTUAL POSITION			000002 N0120		
(RELATIVE)		(ABSOLUTE)		(MACHINE)	
X	0.0000	X	0.0000	X	0.0000
Y	0.0000	Y	0.0000	Y	0.0000
Z	0.0000	Z	0.0000	Z	0.0000
(HANDLE INTR)		(SUBSPEED)		(REM DIST)	
X	0.0000	X	0.0000	X	0.0000
Y	0.0000	Y	0.0000	Y	0.0000
Z	0.0000	Z	0.0000	Z	0.0000
			S00000 T0100		
			AUTO		

Fig. 3-1-1-3

#### 4) Monitoring mode

In 【MONI】 interface, the absolute coordinate, relative coordinate of the current position as well as the current running program modal message and blocks can be displayed together: (See Fig. 3-1-1-4):

MONITOR			000002 N0120		
(ABSOLUTE)		(REM DIST)		G00 G17 G90 G94	
X	0.0000	X	0.0000	G54 G21 G40 G49	
Y	0.0000	Y	0.0000	G80 G98 G15 G50	
Z	0.0000	Z	0.0000	F 1000 AF 0	
			S 1000 M 3		
00002;					
N0060 X100;					
N0120 X0;					
N0180 Y100;					
			LN: 3		
			S00000 T0100		
			AUTO		

Fig. 3-1-1-4

- Note 1** The display in 【MONI】 mode can be set by BIT6 of the parameter NO.023. When BIT6=0, the machine coordinate but the modal instruction is displayed at the original position.
- Note 2** In <MACHINE ZERO>, <STEP>,<JOG>,<MPG>modes, the intermediate coordinate system is a relative one; while in <AUTO>,<MDI>,<DNC> modes, it is a remaining distance.

### 3.1.2 Display of run time, part count, programming speed and override, actual speed etc.

The programming speed, the actual speed, feedrate and rapid override, G codes, tool offset, run time, part count, spindle override, spindle speed, tools etc. can be displayed in【absolue】or【relative】mode in<POSITION> interface(see Fig.3-12-1).

ACT POS (RELATIVE)		00002 N0120
<b>O0002</b>	<b>N00000</b>	PRO SPEED: 0
<b>X</b>	<b>16.000</b>	ACT SPEED: 0
<b>Y</b>	<b>16.000</b>	FEED OVRD: 100%
<b>Z</b>	<b>56.000</b>	RAPID OVRD: 100%
		OFFSET: H0000 D0000
		PRT CNT: 0000/0000
		RUN TIME: 000: 00: 00
		CURRENT TIME: 10:06:00
		G00 G17 G54 G21 G40 G49
		Sx 100% S0000 T0100
		MDI

Fig. 3-1-2-1

The meaning of them is as following:

Programming speed: Speed specified by F code

Actual speed: The actual cutting rate overridden

Feedrate override: Feed override selected by feedrate override keys

Rapid override: Rapid override selected by rapid override keys

G codes: The value of the G code in block being executed

Tool offset: H0000, tool length compensation for current program;D0000, tool radius compensation for current program

Parts count: Plusing 1 when M30 is executed

Run time: Time counting start when Auto run starts, time units are hour, minute and second

Sx: Spindle override for spindle speed


S00000: Actual feedback speed of spindle encoder

T0000: Tool number specified by T code in program

**Note The parts counting is reserved after the power-down.**

The clearing ways for parts counting and run time:

1) Switchover to POSITION mode.

2) Press  key, the cursor locates to the PRT CNT item, input data and press



key to confirm; if



key is pressed directly, the parts counting will be cleared.

3) Shift UP or DOWN key to RUN TIME.



4) Press  key to clear the RUN TIME.


- Note**
- 1 To display the actual spindle speed, the encoder must be applied to the spindle.
  - 2 The actual speed= the programming speed F × override; in G00 mode the axes speeds are set by number parameter P88~P92 and they can be overridden by rapid override; the dry run speed is set by number parameter P86.
  - 3 The programming speed for feed per revolution is displayed when the block involving feed per revolution is being executed.
  - 4 The total workpieces machined is set by number parameter P356, total workpieces to be machined is set by number parameter P357.

### 3.1.3 Relative coordinate clearing and mediating

The steps of relative coordinate position clearing are as follows:


- 1) Enter a mode that displays the relative coordinate;(Fig. 3-1-2)



- 2) **Clearing operation:** Press and hold “X” key till X in the display flickers, then press  key, the relative coordinate in X axis will be cleared; (Fig. 3-1-2-1).


- 3) **Mediating operation:** Press and hold “X” key till “X” in the display flickers, then press



“” key and the relative coordinate in X axis will be mediated. (The relative coordinate of the axis divided by 2)

- 4) **Coordinate setting:** Press and hold “X” key till “X” in the display flickers, input the setting data



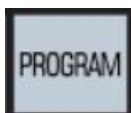
and press  key, the data will be entered into the coordinate system.

ACT POS (RELATIVE)		00002 N0120
<b>O0002</b>	<b>N00000</b>	PRG SPEED: 0
<b>X</b>	<b>0.000</b>	ACT SPEED: 0
<b>Y</b>	<b>16.000</b>	FEED OVRD: 100%
<b>Z</b>	<b>56.000</b>	RAPID OVRD: 100%
		OFFSET: H0000 D0000
		PRT CNT: 0000/0000
		RUN TIME: 000: 00: 00
		CURRENT TIME: 10:06:00
		G00 G17 G54 G21 G40 G49
		Sx 100% S0000 T0100
		MDI

Fig. 3-1-2-2

- 5) The clearing of Y and Z axes are the same as above.

## 3.2 Program display



Press  key on the panel to enter program display that have 5 modes: **【◆PRG】** ,

【MDI】、【CUR/MOD】and 【DIR】. They can be viewed and modified by pressing this key continuously.  
See Fig.3-2-1 as following:

1) Program display:

Press 【◆PRG】 soft key to enter program page, in this mode, a page of the blocks being executed in the memory can be displayed(See Fig. 3-2-1).

PROGRAM	000002	N00120
00002;		
N0060	X100;	
N0120	X0;	
N0180	G01 X50 Y50 F2000 ;	
N0240	G41 X100 D1	
N0300	G01 Y100	
N0360	G02 X200 R50	
N0420	G01 Y0 F2500	
N0180	X0	
N0180	Y50	
DATA:	Ln: 2	S00000 T0000
		EDIT

Fig. 3-2-1

2) MDI display

Press 【MDI】 soft key to enter MDI page, in this mode, multiple blocks can be edited and executed. The program format is the same as the editing program. MDI mode is applicable to simple program testing operation. (See Fig. 3-2-2)

PROGRAM (MDI)	00002	N00000
00000;		
%		
DATA:	Ln: 2	S00000 T0000
MDI		

Fig. 3-2-2

3) Program (CUR/MOD) display

Press 【CUR/MOD】 soft key to enter current/mode interface, it displays the instructions of the blocks being executed and the current mode. MDI data input and running are available in MDI mode. (See Fig. 3-2-3).

PROGRAM CURRENT/MODAL)						00002	
N00000							
(CURRENT)				(MODAL)			
X	P	G00	G11	F	300		
Y	Q	G17	G98	S	0		
Z	F	G90	G15	M	30		
*	L	G94	G50	T	0000		
*	S	G54	G69	H	0000		
R	M	G21	G64	D	0000		
I	T	G40	G97				
J	H	G49					
K	D						
DATA:				S00000	T0000		

Fig. 3-2-3

## 4) Program (DIR) display

Press **【DIR】** soft key to enter directory interface, it displays as follow(Fig.3-2-4):

(a) SYS VERSION: hardware and software

(b) PRG USED: The programs saved (including subprogram)

FREE: number of the programs that can be saved.

(c) MEM USED: the capacity occupied by the programs saved (expressed by characters)

FREE: the capacity available for program storage.

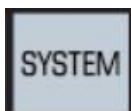
(d) PROGRAM DIR: number of the program saved displayed by sequence. Press **CHANGE** key, the program display changes by the name sequence and time sequence.

PROGRAM (DIR)			00002 N00000		
SYS VERSION: 1.1 (HARD)			1.96 (SOFT)		
PROGRAM USED: 2			FREE: 398		
MEM USED: 224 (K)			FREE: 58112 (K)		
PROGRAM DIR:					
00001		9		2008-08-16 09: 10	
00002		114		2008-08-17 11: 23	
No. :			S00000 T0000		
MDI					

Fig. 3-2-4

**Explanation:** The program numbers in memory can be displayed by the page keys.

### 3.3 System display



Press **SYSTEM** key to enter system page. There are **【BITPAR】**, **【NUMPAR】**, **【COMVAR】**

**【SYSVAR】**, **【COMVAR】**, **【OFFSET】** and **【PITCH】** 6 display pages, and they can be viewed

and modified by pressing this key continuously, the steps are as following:

### 3.3.1 Display. Modification and setting for parameter

#### 3.3.1.1 Parameter display

1) Bit parameter page the bit parameter page is shown as follows (see Fig. 3-3-1-1):

BIT PARAMETER										00002 N00120	
No.	DATA										
0000	0	0	0	0	0	0	1	0			
	****	****	SEQ	****	****	INI	ISO	****			
0001	0	0	0	0	0	0	0	0			
	SJZ	****	MIRz	MIRy	MIRx	SPT	****	****			
0002	0	0	0	0	0	0	1	1			
	ND3	IOP	****	****	ASI1	SB1	ASIO	SB0			
0003	0	0	1	1	1	0	1	0			
	****	****	DIR5	DIR4	DIRZ	DIRY	DIRX	INM			
DATA										S00000 T0010	
MDI											

Fig. 3-3-1-1-1

See details about this parameter in Appendix 1.

2) Number parameter page the number parameter page is shown as follows (see Fig. 3-3-1-1):

NUM	PARAMETER	00002 N00000
NO.	MEANING	DATA
000	I/O chanel,select input/output device	0
001	communication chanel 0 baudrate (DNC)	38400
002	communication chanel 1 baudrate	115200
003	screen protection wait time (min)	0
004	system interpolation period (1, 2, 4, 8ms)	1
005	CNC contol axes	3
006	program axis name of each axis	0
007	set each axis name in the basic coordinate system	0
DATA:		S00000 T0000
MDI		

Fig. 3-3-1-1-2

See details about this parameter in Appendix 1.

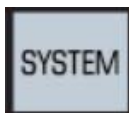
#### 3.3.1.2 Modification and setting of the parameter values

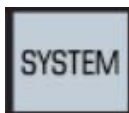
1) Select <MDI > mode;



2) Enter mode, set the parameter on-off for ON in 【ON-OFF】 interface.

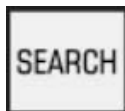




3) Press  key to enter the parameter page;

4) Move the cursor to the parameter number to be changed:

Method 1: Press page keys to display the parameter to be set; then move the cursor to the place to be modified;

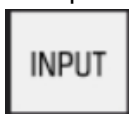


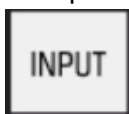
Method 2: Press  key in the panel, input parameter number to be modified and



press  key to confirm.

5) Input the new parameter values by numerical keys (different level parameters modification need corresponding level authority).



6) Press  key to confirm, the parameter value will be input and displayed.

7) After all parameters are set and entered, set the parameter on-off (switch) for OFF.

### 3.3.2 Display, modification and setting for macro variables

#### 3.3.2.1 Macro variable display

1) **User variable page** the page is shown as follows (see fig. 3-3-2-1-1):

COMMON VARIABLES		O0002 00120	
NO.	DATA	NO.	DATA
0000		0009	
0001		0010	
0002		0011	
0003		0012	
0004		0013	
0005		0014	
0006		0015	
0007		0016	
0008		0017	
COM VARIABLES (Memory variables )			
NO. :		S00000	T0000
		MDI	

Fig. 3-3-2-1-1

2) **System variable page** the page is shown as follows (see fig. 3-3-2-1-2):

SYS VARIABLES		O0002 00120	
NO.	DATA	NO.	DATA
1000	0	1009	0
1001	0	1010	0
1002	0	1011	0
1003	0	1012	0
1004	0	1013	0
1005	0	1014	0
1006	0	1015	0
1007	0	1016	0
1008	0	1017	0
INPUT SIGNAL		S00000 T0000	
NO. :		MDI	

Fig. 3-3-3-1-2

Refer to SECTION 4.7.2 in PROGRAMMING for the explanation and use of macro variables.

### 3.3.2.2 Modification and setting for macro variables

- 1) Select <MDI> mode.



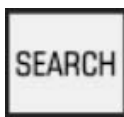
- 2) Enter **SETTING** page, turn on parameter switch in **【ON/OFF】** page (do not need to turn on the parameter switch when modifying common variables #100~#199 and #500~#999).



- 3) Press **SYSTEM** key to enter macro variable display page.

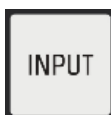
- 4) Move the cursor to the variable number to be modified.

Method 1: Press page keys to display the page where the variable is to be modified; move the cursor to the variable to be modified.



Method 2: Press key **SEARCH**, and press **INPUT** key to positioning after inputting the variable number.

- 5) Input a new value using number keys.



- 6) Press key **INPUT** for confirmation, and then the value will be input and displayed.

### 3.3.3 Display, modification and setting for offset

#### 3.3.3.1 Offset display

- 1) Offset page the page is shown as follows (see fig.3-3-3-1-1).

OFFSET			O0002	00120
NO.	GEOM (H)	WEAR (H)	GEOM (D)	WEAR (D)
001	0.000	0.000	0.000	0.000
002	0.000	0.000	0.000	0.000
003	0.000	0.000	0.000	0.000
004	0.000	0.000	0.000	0.000
005	0.000	0.000	0.000	0.000
006	0.000	0.000	0.000	0.000
007	0.000	0.000	0.000	0.000
008	0.000	0.000	0.000	0.000
CUR POSITION (RELATIVE)				
X	-266.667	Y	-117.111	Z 622.674
NO.			S00000	T0000
				MDI

Fig. 3-3-3-1-1

The offset value can be input directly or added to or subtracted from the actual position value. GEOM (H) stands for tool length compensation, WEAR (H) for tool length abrasion; GEOM (D) stands for tool radius compensation and WEAR (D) for tool radius abrasion.

### 3.3.3.2 Modification and setting for offset value

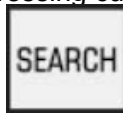
The steps for setting tool offset in offset page are as follows:



- 1) Press  key to enter offset page


- 2) Move the cursor to the target offset number.

Step 1: Press page keys to display the page where the offset value is to be modified, move the cursor by pressing cursor keys to the offset number to be modified.



Step 2: Press key , and press  key to positioning after inputting offset number.



- 3) Input offset value in any mode, after pressing  key, the offset value will be worked out automatically and displayed on LCD.

**Note 1:** During the tool offset modification, the new offset value is ineffective till the T code which specifies its offset number is specified.

- 2: The offset value can be modified anytime during the program execution. If the value is required to take effect in time during the program execution, the modification must be completed before the tool offset number is executed.

- 3: If the length offset value needs to be added to the relative coordinate value of Z axis, the offset value should be specified behind Z code, then they will be automatically added up in the system.

For example, if Z 10 is input, the offset value is the one obtained by adding 10 to the current relative coordinate value of Z axis.

3.3.4 Display, modification and setting for screw pitch offset

3.3.4.1 Screw pitch offset display

1) Screw pitch page pitch offset page is shown as follows (fig. 3-3-4-1-1 ):

Pitch Offset (Max. offset points: 256)				O0002
00120				
NO.	PITCH X	PITCH Y	PITCH Z	PITCH TH4
0000	0	0	0	0
0001	0	0	0	0
0002	0	0	0	0
0003	0	0	0	0
0004	0	0	0	0
0005	0	0	0	0
0006	0	0	0	0
0007	0	0	0	0
			S00000	T0000

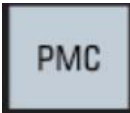
Fig. 3-3-4-1-1

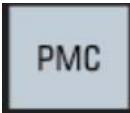
3.3.4.2 Modification and setting of pitch offset

- 1) The axes offset points, pitch error offset interval and pitch error offset multiplier are set by the number parameters **P221~P225**, **P226~P230** and **P231~P235** respectively.
- 2) In MDI mode, Enter offset value for every point by sequence.

**Note** Refer to Volume Four Connection of GSK990MA Connection and PLC manual for the pitch setting.

3.4 PMC display



Press the key  to display the PMC page. There are 3 subpages, including **【PLCPara0】**, **【PLCPara1】**, **【PLCPara2】** , which can be viewed by pressing the key continuously (See Fig.3-4-1 to Fig.3-4-3) .

1) **PLCPara0** page PLCPara0 page is shown as following fig.3-4-1:

PLCPara 0			O0002	N00120
NO.	ADDR	CURRENT	SET	
000	C000	00000	00000	
001	C001	00000	00000	
002	C002	00000	00000	
003	C003	00000	00000	
004	C004	00000	00000	
005	C005	00000	00000	
006	C006	00000	00000	
007	C007	00000	00000	
008	C008	00000	00000	
NO. :		S00000	T0000	
			MDI	

Fig. 3-4-1

2) **PLCPara1** page PLCPara1 page is shown as following fig.3-4-2:

PLCPara1						O0002		N00120	
ADDR	N. 7	N. 6	N. 5	N. 4	N. 3	N. 2	N. 1	N. 0	
K000	0	0	0	0	0	0	0	0	
K001	0	0	0	0	0	0	0	0	
K002	0	0	0	0	0	0	0	0	
K003	0	0	0	0	0	0	0	0	
K004	0	0	0	0	0	0	0	0	
K005	0	0	0	0	0	0	0	0	
K006	0	0	0	0	0	0	0	0	
K007	0	0	0	0	0	0	0	0	
NO. :					S00000	T0000			
					MDI				

Fig. 3-4-2

3) **PLCPara2** page PLCPara2 page is shown as following fig.3-4-3:

PLCPara 2			O0002 N00120		
NO.	ADDR	DATA	NO.	ADDR	DATA
000	D000	00000	009	D009	00009
001	D001	00000	010	D010	00010
002	D002	00000	011	D011	00011
003	D003	00000	012	D012	00012
004	D004	00000	013	D013	00013
005	D005	00000	014	D014	00014
006	D006	00000	015	D015	00015
007	D007	00000	016	D016	00016
008	D008	00000	017	D017	00017
NO. :			S00000 T0000		
			MDI		


Fig. 3-4-3

3.5 Setting display

3.5.1 Setting page

1 Entry of page

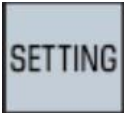


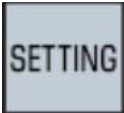
Press  key to enter the SETTING page, there are 【SETTING】、【SWITCH】、【G54-G59】、【DATA】 , 【PSW】 sub-modes in this page. They can be viewed or modified by pressing this key continuously, the pages are shown as following (see Fig. 3-5-1-1):

SETTING		00002 N00120	
MIRROR X =	1	( 0: OFF 1: ON)	
MIRROR Y =	1	( 0: OFF 1: ON)	
MIRROR Z =	1	( 0: OFF 1: ON)	
INCH PRG =	0	( 0: METRIC, 1: INCH)	
I/OCHAN. =	0	( 0—3 CHANNEL NO. )	
ABS PRG =	0	( 0: ABS, 1: INC)	
AUTO SEQ =	0	( 0: OFF 1: ON)	
SEQ STOP =	0000	( PRG NO. )	0000 (SEQ NO. )
2007 Y 08 M 08 D		08 H 08 M 08 S	
DATA		S00000 T0010	
MDI			

Fig. 3-5-1-1

2 【SETTING】 explanation



Press  soft key to enter the 【SETTING】 page shown as Fig.3-5-1. After entering the page, the user can view and modify the parameters. The operation steps are as following:

- (a) Enter < MDI> mode;

- (b) Press UP or DOWN keys to move the cursor to the items to be altered;
- (c) Key in 1 or 0 by following steps:
- 1) X, Y, Z axis mirror image  
0: Mirror image OFF    1: Mirror image ON
  - 2) Inch unit  
Set the input unit of the program for inch or mm  
0: mm                      1: inch
  - 3) I/O channel  
To be set by user's requirement. If U flash disk is used to perform the DNC machining, set the channel to 1.  
0,2: RS323 (0: select Xon/Xoff protocol; 2: select Xmodem protocol ), 1: USB
  - 4) Absolute programming  
0: Absolute programming                      1: Incremental programming
  - 5) Automatic sequence number  
0: The number is not inserted by system automatically when inputting program by keyboard in Edit mode.  
1: The number is inserted automatically by system when inputting program by keyboard in Edit mode. The number increment of blocks can be set by number parameter **P210**.
  - 6) Stop number  
This function can be used to specify the program execution to stop at a block specified, but the program number and the block number should be specified together for this function. E.g. 00060 (program number) means program number O00060; 00100 (sequence number) means block number N00100.
  - 7) Date and time  
User can set the system date and time herein.



- (d) Press  key to confirm the entry.

### 3.5.2 Parameter and program on-off page -

- 1 The -PAR SWITCH and PRG SWITCH page are shown as following (see Fig. 3-5-2-1):

SETTING	000002 N00120
_PAR SWITCH: ◆ OFF	ON
PRG SWITCH: OFF	◆ ON
S00000 T0010	
EDIT	

Fig. 3-5-2-1

## 2 Operation explanation

In page above, the user can set the parameter and program switch. The operation steps are as following:

- Enter the <MDI> mode, the parameter ON should be in MDI mode; Parameter OFF and program ON and OFF may be in any mode.
- Press UP or DOWN direction key to move the cursor to the items to be altered in the parameter or program;
- Set the parameter or program switch by pressing Left or Right cursor key. When the parameter switch is set for "OFF", the system parameter modification and setting are unallowed; when the program switch is set for "OFF", the program editing is unallowed either.
- When the parameter switch is set for "ON", alarm occurs (0100: parameter write –in

available ), so ,the alarm can be cancelled by pressing



key.

### 3.5.3 Coordinate setting interface

- The 【SETTING (G54-G59)】 interface is shown as following (Fig.3-5-3-1):

SETTING (G54-G59)		00002 N00120
CURRENT COORDINATE SYSTEM: G54		
(MACHINE)	(G54)	(G55)
X 0.000	X 0.000	X 0.000
Y 0.000	Y 0.000	Y 0.000
Z 0.000	Z 0.000	Z 0.000
(EXT)	(G56)	(G57)
X 0.000	X 0.000	X 0.000
Y 0.000	Y 0.000	Y 0.000
Z 0.000	Z 0.000	Z 0.000
DATA	S00000	T0010
		MDI

Fig. 3-5-3-1



Besides 6 (from G54 to G59) workpiece coordinate system (standard), 50 additional workpiece coordinate systems can also be used in this system, as are shown in Fig. 3-5-3-2. And each coordinate system can be viewed or modified by page keys. See details for these additional workpiece coordinate systems in *PROGRAMMING* Section 4.2.9. Additional workpiece coordinate system.

SETTING (G54-G59)			00002 N00120		
CURRENT COORDINATE SYSTEM: G54					
(MACHINE)		(G58)		(G59)	
X	0.000	X	0.000	X	0.000
Y	0.000	Y	0.000	Y	0.000
Z	0.000	Z	0.000	Z	0.000
(EXT)		(G54.001)		(G54.002)	
X	0.000	X	0.000	X	0.000
Y	0.000	Y	0.000	Y	0.000
Z	0.000	Z	0.000	Z	0.000
DATA				S00000	T0010
MDI					

Fig. 3-5-3-2

2. There are two ways for coordinate entry:

- 1) After entering this page in <MDI> mode, move the cursor to the coordinate system to be



altered. Press the axis name to be assigned and then press key for confirmation, the value in current machine coordinate system will be set for the origin of



the G coordinate system. e.g. If "X" is pressed and then key, the X machine coordinate of a point is entered automatically by system. Another example, if



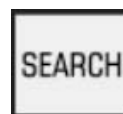
"X10" is entered, and then press key, which means X machine coordinate is +10; and "X-10" may also be entered.

- 2) After entering this page in <MDI> mode, move the cursor to the coordinate axis to be altered, input the machine coordinates or other values directly to define the G

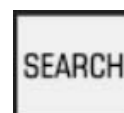


coordinate system origin, press key for confirmation.

3. Coordinate system search method



- 1) In any mode, positioning can be performed by pressing key after coordinate system is inputted. e.g. input "56".



- 2) In any mode, input the data such as "6", "06" or "006" and press key, then the

cursor will positioning on additional workpiece coordinate system “G54.006”.

3.5.4 Data processing interface

The data processing interface is as Fig.3-5-4-1 shows. The user data (such as mode parameter, number parameter, tool parameter, pitch data, ladder and programs) can be backup (saved) and reverted (read); and the data input or output to PC is also available in this system. The part programs saved in CNC are unaffected during the data backup and reversion.

(See Fig.3-5-4-1)

SETTING (DATA)

00002 N00120

	BACKUP	REVERT	OUTPUT	INPUT
SYS PAR :				
TOOL COMP :				
PITCH COMP:				
MACRO VAR :				
LADDER (PLC) :				
CNC PRG :				

S00000 T0010

MDI

Fig.3-5-4-1

Operation:

1 In 【PSW】 page of <SETTING> mode, set the corresponding password by pressing 【PSW】 soft key. The ladders, parameters can be only operated under the machine builder’s authority level. System parameters, tool offset, pitch compensation and system macro variables can be operated under the system debugger level or above.

2

Data	Password authority
ladder (PLC)、parameters	password for machine tool builder level, password for system manufacturer level
System parameter, tool compensation value,screw pitch offset value, system macro variable	password for machine tool builder level, password for system manufacturer level, Password for system debugging level
CNC part pogram	Without a password

Return to 【DATA】 page, after the cursor moves to the target position, the backup or reversion



of the data can be finished by pressing key.

Note 1: When performing data output/input operation, ensure the setting for the I/O channel is correct. When using a U flash disk, set the I/O channel to 1; when using transmission software via PC, set the I/O channel to 0 or 2.

Note 2: Bit parameter N0:27#0: for setting whether the editing for subprograms with program numbers from 80000-89999 is forbidden.

Note 3: Bit parameter N0:27#4: for setting whether the editing for subprograms with program numbers from 90000-99999 is forbidden.

### 3.5.5 Password authority interface

To prevent the part programs and CNC parameter from malicious modification, the password authority setting is available in this GSK990MA system. It is classified for 5 levels, which are the 1<sup>st</sup> level (system manufacturer), the 2<sup>nd</sup> level (machine builder), the 3<sup>rd</sup> level (system debugger), the 4<sup>th</sup> level (terminal user), the 5<sup>th</sup> level (operator) by descending sequence. The system defaults the lowest level at power-on (See Fig.3-5-5-1) .

The 1st and the 2nd level: The modifications of mode parameters, number parameters, tool offset data and PLC ladders transfer etc. are allowed in these levels.

The 3rd level: The modifications of CNC mode parameters, number parameters, tool offset data etc. are allowed in this level.

The 4th level: The modifications of macro variables, tool offset data are allowed in this level. But the modifications of CNC mode parameters, number parameters, pitch compensation data are not allowed in this level.


The 5th level: No password. The operation of the machine operator panel is allowed in this level, but the modifications of tool offset parameters, CNC mode parameters, number parameters, pitch compensation data are not allowed.

SETTING (PASSWORD)	000002 N0000
SYSTEM PSW :	_____
NEW: _____ AGAIN: _____	
MACHINE PSW:	_____
NEW: _____ AGAIN: _____	
DEBUG PSW :	_____
NEW: _____ AGAIN: _____	
USER PSW :	_____
NEW: _____ AGAIN: _____	
	S0000 T0100
	EDIT

**Fig. 3-5-5-1**

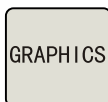
- 1) After entering this page in MDI mode, move the cursor to the item to be altered;



- 2) Key in the password under the corresponding level, then press  key. If the password is correct, the message "Password is correct." is issued by the system. If not, "Password is not correct." is issued.
- 3) Modify the corresponding parameters and setting for the system password;

- a The program on-off is required to be set for ON during the parameter modification.
  - b K parameter is needed to be set for ON during the ladder modification.
- 4) After modification, the password is automatically deregistered after the system power-off or input the wrong password again.

### 3.6 Graphic display



Press  key to enter the graphic page that has two display modes:

【G. PARA】 and 【◆GRAPH】. They can be switched over by pressing this key continuously (See Fig.3-6-1).

GRAPH ( PARA )				000002 N00120			
AXES = 0 (0: XY 1:XZ 2:ZX 3:YZ 4: XYZ 5:ZXY)							
GRPH MOD =				0 (0:GRPH CENT 1:MIN&MAX)			
AUTO ERA =				0 (0: OFF 1: ON)			
SCALE =				1.0000			
GRPH CEN =				0.0000 (X COORDINATE)			
GRPH CEN =				0.0000 (Y COORDINATE)			
GRPH CEN =				0.0000 (Z COORDINATE)			
MAX X		=	240.0000	MIN X		=	-240.0000
MAX Y		=	240.0000	MIN Y		=	-240.0000
MAX Z		=	240.0000	MIN Z		=	-240.0000
DATA				S0000 T0010			
MDI							

**Fig. 3-6-1**

1) Graphic parameter page, see Fig.3-6-1.

#### A. Graphic parameter meaning

- ① Coordinate selection: set drawing plane that has 6 types as shown in the figure above.
- ② Graphic mode: set graphic display mode
- ③ Scaling: set drawing ratio
- ④ Graphic center: set the coordinate of the LCD center in workpiece coordinate system
- ⑤ The maximum and minimum value: The scaling and the graphic center are automatically set when the maximum and minimum value of the axis are set.

Maximum value of X axis: the maximum value along X axis in graphics (unit: 0.001mm)

Minimum value of X axis: the maximum value along X axis in graphics (unit: 0.001mm)

Maximum value of Y axis: the maximum value along X axis in graphics (unit: 0.001mm)

Minimum value of Y axis: the maximum value along X axis in graphics (unit: 0.001mm)


Maximum value of Z axis: the maximum value along X axis in graphics (unit: 0.001mm)

Minimum value of Z axis: the maximum value along X axis in graphics (unit: 0.001mm)

#### B. The graphic parameters setting methods:

- a. Move the cursor to the parameter to be set;
- b. Key in the value desired;



c. Press  key to confirm it.

2) Graphic page is shown as Fig.3-6-2 :

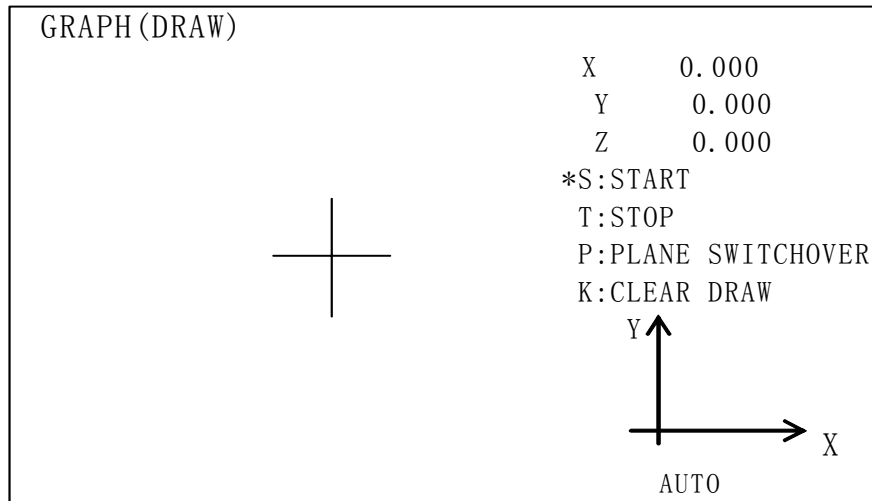



Fig. 3-6-2

The figure machined can be monitored in graphic page,





A Press  key to enter the DRAW START mode, then the sign '\*' is headed to **S: START**;



B Press  key to enter the DRAW STOP mode, then the sign '\*' is headed to **T: STOP**;



C Press  +  soft key to switch over the graph in the corresponding **0~5** coordinate display page;



D Press  +  soft key to erase the graph drawn.


### 3.7 Diagnosis display

The status of DI/DO signals between CNC and the machine, the signals transferred between CNC and PLC, PLC internal data and CNC internal status etc. are shown in the diagnosis display. Refer to *GSK990MA CNC System Connection and PLC Manual* for the meaning and setting of the corresponding diagnosis number.

The diagnosis of this part is used to detect the CNC interface signals and the internal running signals, and it can't be modified.

### 3.7.1 Diagnosis data display



Press  key to enter the Diagnose page, which has 4 modes: **【NC】**, **【PLC→NC】**, **【MT】** and **【PLC→MT】**. They can also be viewed by pressing this key continuously (See Fig.3-7-1-1 to Fig.3-7-1-5)

1 NC interface is shown as follow. Fig.3-7-1-1:

DIAGNOSE (NC)				O0002 00120	
NO.	DATA			NO.	DATA
000	0	0	0	008	0 0 0 0 0 0 0 0
001	0	0	0	009	0 0 0 0 0 0 0 0
002	0	0	0	010	0 0 0 0 0 0 0 0
003	0	0	0	011	0 0 0 0 0 0 0 0
004	0	0	0	012	0 0 0 0 0 0 0 0
005	0	0	0	013	0 0 0 0 0 0 0 0
006	0	0	0	014	0 0 0 0 0 0 0 0
007	0	0	0	015	0 0 0 0 0 0 0 0
NO. :				S00000 T0000	
				MDI	

**Fig.3-7-1-1**

This is the signal sent to PLC by CNC system. See *GSK990MA CNC System Connection and PLC Manual* for the meaning and setting of the corresponding diagnosis number.

2 PLC→NC interface , it is shown in Fig.3-7-1-2:

DIAGNOSE (PMC→NC)				O0002 00120	
NO.	DATA			NO.	DATA
000	0	0	1	008	0 0 0 0 0 0 0 0
001	0	0	0	009	0 0 0 1 0 0 0 0
002	0	0	0	010	0 0 0 0 0 0 0 0
003	0	0	0	011	0 0 0 0 0 0 0 0
004	0	0	0	012	0 0 0 0 0 0 0 0
005	0	0	0	013	0 0 0 0 0 0 0 0
006	0	0	0	014	0 0 0 0 0 0 0 0
007	0	0	0	015	0 0 0 0 0 0 0 0
NO. :				S00000 T0000	
				MDI	

**Fig.3-7-1-2**

This is the signal sent to CNC system by PLC. See *GSK990MA CNC System Connection and PLC Manual* for the meaning and setting of the corresponding diagnosis number.

3 MT In <DIAGNOSE> page, it is shown in Fig.3-7-1-3:

DIAGNOSE ( MACHINE )		O0002 00120	
NO.	DATA	NO.	DATA
000	0 0 0 0 0 0 0 0	008	0 0 0 0 0 0 0 0
001	0 0 0 0 0 0 0 0	009	0 0 0 0 0 0 0 0
002	0 0 0 0 0 0 0 0	010	0 0 0 0 0 0 0 0
003	0 0 0 0 0 0 0 0	011	0 0 0 0 0 0 0 0
004	0 0 0 0 0 0 0 0	012	0 0 0 0 0 0 0 0
005	0 0 0 0 0 0 0 0	013	0 0 0 0 0 0 0 0
006	0 0 0 0 0 0 0 0	014	0 0 0 0 0 0 0 0
007	0 0 0 0 0 0 0 0	015	0 0 0 0 0 0 0 0
NO. :		S00000 T0000	
		MDI	

Fig.3-7-1-3

This is the signal sent to PLC by machine. See *GSK990MA CNC System Connection and PLC Manual* for the meaning and setting of the corresponding diagnosis number.

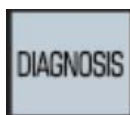
4 PLC—>MT interface it is shown in Fig.3-7-1-4:

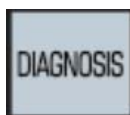
DIAGNOSE ( PLC—>NC )		O0002 00120	
NO.	DATA	NO.	DATA
000	0 0 0 1 0 0 0 1	008	0 0 0 0 0 0 0 0
001	0 0 0 0 0 1 0 0	009	0 0 0 0 0 0 0 0
002	0 0 0 0 0 0 0 1	010	0 0 0 0 0 0 0 0
003	0 0 0 0 0 0 0 0	011	0 0 0 0 0 0 0 0
004	0 0 0 0 0 0 0 0	012	0 0 0 0 0 1 0 0
005	0 0 0 0 0 0 0 0	013	0 0 0 0 0 1 0 0
006	0 0 0 0 0 0 0 0	014	0 1 0 0 0 0 0 0
007	0 0 0 0 0 0 0 0	015	0 0 0 0 0 0 1 0
NO. :		S00000 T0000	
		MDI	

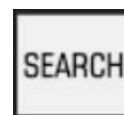
Fig.3-7-1-4

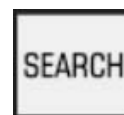
This is the signal sent to machine by PLC. See *GSK990MA CNC System Connection and PLC manual* for the meaning and setting of the corresponding diagnosis number.

### 3.7.2 Signal viewing



- 1) Press  key to select the DIAGNOSE page.
- 2) The respective address explanation and meaning are shown at the down-left of the screen when moving the cursor to the left or right.



- 3) Move the cursor or key in the parameter address to be searched, then press  key, the target address will be found.

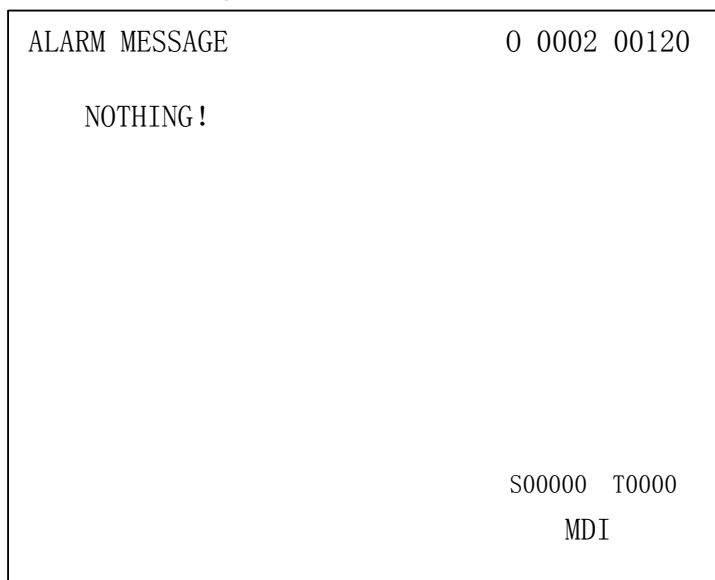
### 3.8 Alarm display

When an alarm is issued, "ALARM" is displayed at the bottom line of the LCD. Press the



key to display the alarm page, there are 3 modes **【ALARM】**, **【HISTORY】**, **【OPERATE】** in this page, which can be viewed by this (See Fig.3-8-1 to Fig.3-8-4) . They can also be set by parameter No.24.6 for switching to alarm interface if an alarm is issued.

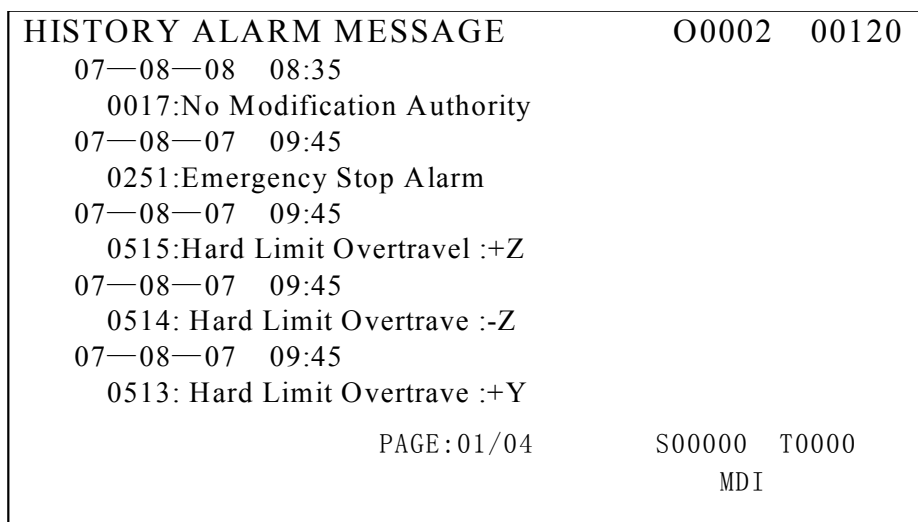
1 Alarm interface it is shown in Fig.3-8-1:



**Fig.3-8-1**

In alarm page, it displays the message of current P/S alarm number. See details for the alarm in Appendix 2.

2 HISTORY interface it is shown in Fig.3-8-2:



**Fig.3-8-2**

To view the message conveniently, the time is listed by descending sequence in this page.

3 OPERATE interface it is shown in Fig.3-8-3:



OPERATE HISTORY	O0002	00120
07—08—08 15:35		
Modify System Bit PAR	0037	
07—08—07 14:45		
Modify System Bit PAR	0026.6	
07—08—07 14:45		
Modify System Bit PAR	0026.7	
07—08—07 09:45		
Modify System Bit PAR	0034.5	
07—08—07 09:45		
Modify System Bit PAR	0034.6	
PAGE:01/04	S00000	T0000
	MDI	

**Fig.3-8-3**

The OPERATE page displays the modification message on the system parameters and ladders, e.g. the modification content and time.

OPERATE and HISTORY alarm interface can display 34 pages of alarm history message, such as the alarm time, alarm number, alarm message and page numbers and they can be viewed by page keys.

The recording of the HISTORY and OPERATE can be deleted by pressing <DELETE> key (system debugger level or above).

### 3.9 Help display (this function is invalid temporarily )

# 4 Manual Operation

The MANUAL mode that contains MANUAL feed, spindle control and machine panel control can

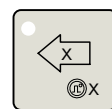


be entered by pressing key.

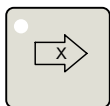
## 4.1 Coordinate axis movement

In JOG mode, the 3 axes can be moved by JOG feed or manual rapid traverse respectively.

### 4.1.1 Manual feed



X axis can be moved to positive or negative direction by pressing and holding the



key in Feed Axis and Direction Selection area. If the key is released, the X axis movement stops. And the feedrate can be overridden to change the feed rate; that of the Y and Z axes are the same as X axis. The three axes simultaneous moving are not available in this system, but the simultaneous zeroing of three axes is supported by the system.

**Note** The axis JOG feedrate is set by parameter P98;

### 4.1.2 Manual rapid traverse



Press key till the indicator for rapid traverse on panel lights up. Then press manual RAPID key, each axis will traverse rapidly.

**Note 1** The manual rapid speeds are set by the parameter P88~ P92.

**Note 2** The effective manual rapid traverse before reference return is set by the bit parameter N0:12#0.

### 4.1.3 JOG feedrate and manual rapid traverse speed selection

The manual feedrate override classified for 16 gears (0%--150%) is available in JOG feed by



pressing key for selection.



The traverse speed can be selected by pressing keys in manual rapid traverse. The override for rapid traverse includes four gears: F0, 25%, 50%, 100% (F0

override by number parameter P93).

**Note1 The rapid overrides are effective for the following speed:**

- (1) G00 rapid traverse
- (2) Rapid traverse in canned cycle
- (3) Rapid traverse in G28
- (4) Manual rapid traverse

Example: If the rapid traverse speed is 6m/min and override is 50%, speed is 3m/min.

**Note2 The adjusting by override keys during the axis moving is ineffective.**

#### 4.1.4 Manual intervention

While a program run in Auto, MDI or DNC modes shifts to JOG mode after a dwell operation, the



manual operation is available. Move the axes manually then shift to Auto mode, press the key to run the program, the axes traverse to the original intervention point by G00 and go on the program execution.

Explanation:

- 1 If the single block is executed during the returning, the tool will stop at a halt position. When the cycle start is put on, the running is restored.
- 2 If alarm or resetting occurs during the manual intervention or returning, this function will be cancelled.
- 3 Don't use machine lock, mirror image, scaling functions during manual intervention.
- 4 Processing and workpiece figure should be taken into consideration to prevent tool or machine damage prior to manual intervention.

The manual intervention operation is shown in the following figure:

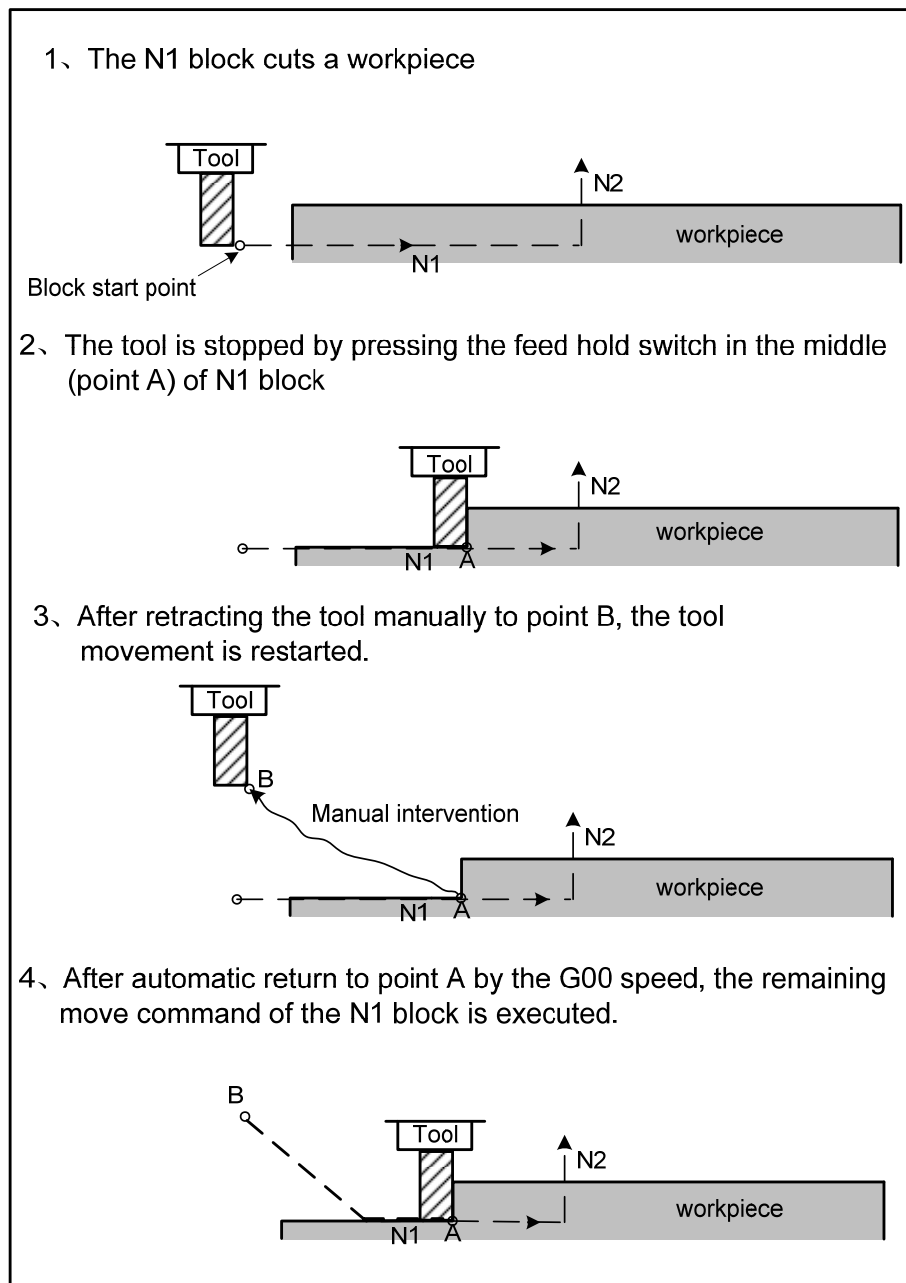


Fig. 4-1-4-1

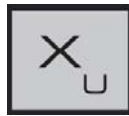
#### 4.1.5 Workpiece alignment

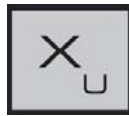

To ensure the part precision (dimensions, profile, and position precision) and surface machining quality, it must do alignment for the part or the fixture that clamps the part.

Methods for alignment: drawing, trial cut etc, according to the alignment characteristic, 990MA CNC system specially does alignment by a tool, for example, to position a center of the rectangle part in X-Y plane by the method of trial cut halving (called centering alignment) alignment, steps are as following:

- 1) Positioning the fixture that clamps the part on the machine tool.
- 2) Chuck the tool into the spindle and rotate the spindle by a certain speed.
- 3) The system switchovers to the display page of relative coordinate. Firstly, align the X direction:

positioning each axis to X direction of workpiece by manual mode. Shift down Z axis to make the tool tip position lower than the workpiece surface, then move to the workpiece negative direction by lower speed (usually use the MPG mode ) till the tool touches the workpiece. At





the moment, press  key on the edit panel area, then press  key to set the coordinate to zero (if set the coordinate to the other value, use the similar method, such as



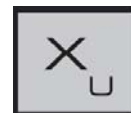
input "X20" then press  key to conform it)

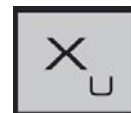

- 4) Ibidem, move the tool to cut the workpiece at the positive side, after positioning, press



 key, and then press  key to finish the centering operation. Note that the centering setting does not change the absolute coordinate and machine coordinate.

- 5) Move the tool to the position whose axis relative coordinate is displayed for "zero", i.e. the center of the X axis.



- 6) Select "G54-G59" page in "SETTING" interface, press  key and then  key to finish the zero setting of X axis.
- 7) In the center of XY plane (i.e. when the XY values of the relative coordinate are 0, the machine positioning point ), the floating coordinate system can be set up by G92, also, the XY machine coordinate for the point can be recorded to parameter of G54~G59 the workpiece coordinate system, and called by the system
- 8) Till now, the operation of aligning the center of the rectangle workpiece by trial cut is finished.

Mastering the relative coordinate assignment method and using the centering function setting will enhance the alignment and the operation convenience

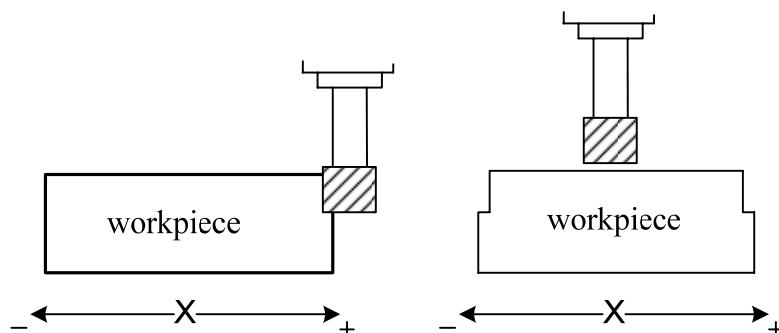


Fig. 4-1-5-1

#### Note

- 1: The system can only set the input for the coordinate displayed in relative position. (the position of relative coordinate can be set in the position can be modified)
- 2: Operation function available. It may do the addition or subtraction for coordinate value displayed, and set the result as the displayed coordinates.
- 3: After the coordinate is set, the coordinate system set by G92 will be lost because of the

power-off, machine zero return or workpiece coordinate system G54~G59 calling, but the coordinate system recorded to G54~G59 workpiece coordinate system by parameter will not lost, the user should set them agilely by required, and the last one is suggested to be used.

## 4.2 Spindle control

### 4.2.1 Spindle CCW



: The spindle is started for CCW rotation if this key is pressed in JOG./MPG/Step mode after S speed is specified in MDI mode.

### 4.2.2 Spindle CW



: The spindle is started for CW rotation if this key is pressed in Manual./MPG/Step mode after S speed is specified in MDI mode.

### 4.2.3 Spindle stop



: The spindle is stopped if this key is pressed in Manual./MPG/Step mode.

### 4.2.4 Spindle auto gear shift

The frequency conversion control or I/O point control for spindle is set by the bit parameter NO:1#2. If parameter NO:1#2=1, the spindle auto gears are controlled by PLC. Three gears(1 to 3 gear) are available in this system, the maximum speed of each gear is set by parameter (P246,P247, P248) respectively, which can be output by modifying the ladder. During the spindle CW or CCW rotation in JOG or Auto mode, the increase or decrease for the corresponding spindle gear can be adjusted by pressing positive/negative override keys. In MDI mode, the system will automatically select the corresponding gear as the specified speed is entered.

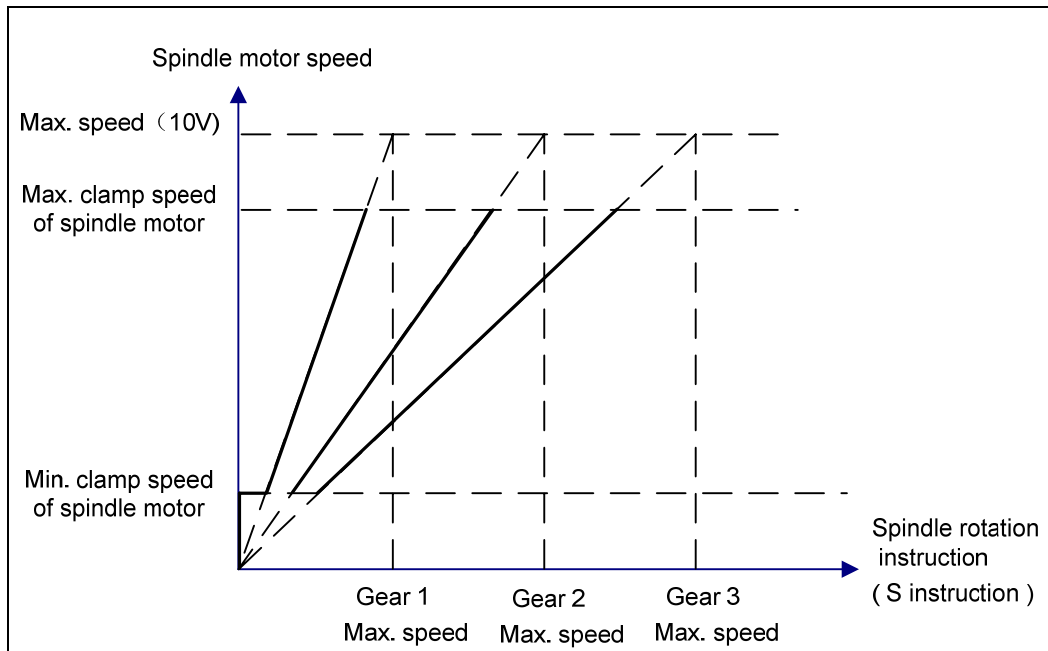


Fig. 4-2-4-1

**Note** When the spindle auto gear is effective, the spindle gear is detected by gear in-position signal and S instruction is executed.

### 4.3 Other manual operations

#### 4.3.1 Cooling control



: Compound key. The cooling function is switched between ON and OFF by pressing this key. The indicator lighting up is for ON, gone out for OFF.

#### 4.3.2 Lubricating control



: Compound key. The lubricating function is switched between ON and OFF by pressing this key. The indicator lighting up is for ON, gone out for OFF.

#### 4.3.3 Work light control




: compound key: The work light is switched between ON and OFF by pressing this key, the indicator lighting up is for ON, gone out for OFF.

# 5

Step Operation

## 5.1 Step feed



Press  key to enter the Step mode, in this mode, the machine moves by the system defined step each time.

### 5.1.1 Selection of moving amount



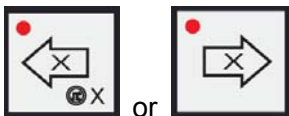
Press a key to select a moving increment in    keys, the increment will be shown on the screen, in <POSITION> interface it displays a step: 0.100 (See Fig. 5-1-1-1):



ACT POS (RELATIVE)		00002	N0120
<b>O0002</b>	<b>N00000</b>	STEP W. :	0.100
<b>X</b>	<b>16.000</b>	ACT SPEED:	0
<b>Y</b>	<b>16.000</b>	FEED ORVRRD:	100%
<b>Z</b>	<b>56.000</b>	RAPID OVRRD :	100%
		OFFSET:	H0000 D0000
		PRT CNT:	0000/0000
		RUN TIME:	000: 00: 00
		CUR TIME:	10:06:00
		G00 G17 G54 G21 G40 G49	
		Sx 100% S0000 T0100	
		STEP	

Fig. 5-1-1-1

The machine axis moves 0.1mm when pressing this key once.

### 5.1.2 Selection of moving axis and direction



pressing axis direction key  or . X axis direction key can move X axis positively or negatively. Press the key once, the corresponding axis will be moved for a step distance defined by system. And the feedrate can be overridden by pressing override keys. The operation for X or Z axis is identical with that of X axis. The manual synchronous 3 axes moving is not supported in this system, but the synchronous 3 axes zero returning is.

### 5.1.3 Step feed explanation

The max.calmping speed for step feed is set by data parameter P155.  
The step federate is not controlled by override and rapid override.



## 5.2 Step interruption

While the program running in Auto, MDI, DNC mode is shifted to Step mode by dwell, the control will execute the step interruption. The coordinate system of step interruption is consistent with that of MPG, and the operation of it is also the same as that of MPG ( electronic handwheel, i.e. standard Manual Pulse Generator—MPG, the following is the same). See details in the Section 6.2 Controlling in MPG interruption.

The step interruption coordinate system clearing steps: press CTRL+X till “X” flickers, then press <CANCEL> key, the coordinate system will be cleared. The operations of Y, Z are the same as above; while the zero returning is being performed, the coordinate system is cleared automatically.


## 5.3 Auxiliary control in Step mode

The auxiliary control in Step mode is the same as that in JOG mode. See details in section 4.2 and 4.3 of this manual.

# 6 MPG Operation




## 6.1 MPG feed



Press  key to enter the MPG mode, in this mode, the machine movement is controlled by a handwheel.

### 6.1.1 Moving amount selection



The moving increment will be displayed if a key in    is pressed., it displays the MPG increment in <POSITION> interface: 0.100 (See Fig.6-1-1-1) :


ACT POS (RELATIVE)		00002	N0120
<b>O0002</b>	<b>N00000</b>	WHEEL INC:	0.100
<b>X</b>	<b>16.000</b>	ACT SPEED:	0
<b>Y</b>	<b>16.000</b>	FEED OVRD:	100%
<b>Z</b>	<b>56.000</b>	RAPID OVRD:	100%
		OFFSET:	H0000 D0000
		PRT CNT:	0000/0000
		RUN TIME:	000: 00: 00
		CUR TIME:	10:06:00
		G00 G17 G54 G21 G40 G49	
		Sx 100% S0000 T0100	
		MPG	

Fig. 6-1-1-1

### 6.1.2 Selection of moving axis and direction

In MPG mode, select the moving axis to be controlled by handwheel, press the corresponding key, then the axis can be moved by handwheel.



In MPG mode, if X axis is to be controlled by handwheel, press  key, then X axis can be moved by rotating the handwheel (See Fig.6-1-2-1) :

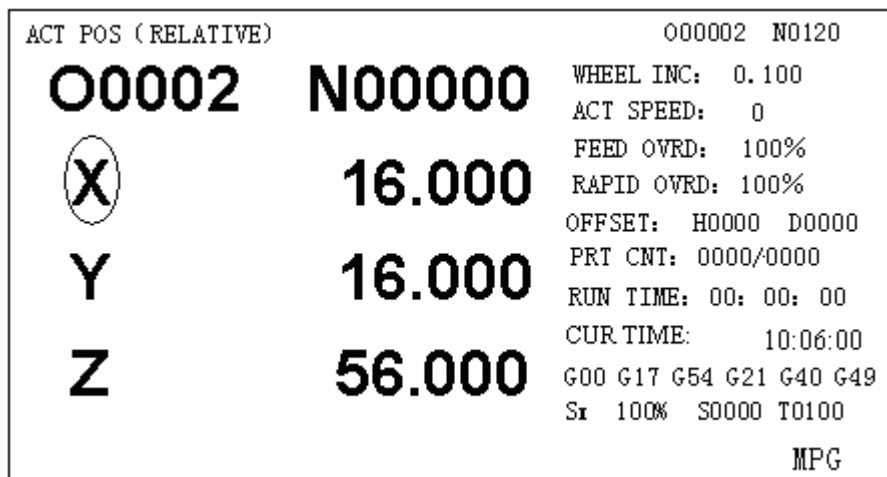


Fig. 6-1-2-1

The MPG feed direction is decided by the handwheel rotation direction. See details in the machine builder's manual. Usually, the CW of handwheel is the positive feed, and CCW for negative feed.

### 6.1.3 Explanation of MPG feed

- 1 The relation of the handwheel scale and the machine moving amount are as following table:

Table 6-1-3-1

	Moving amount of a handwheel scale		
MPG increment (mm)	0.001	0.01	0.1
Machine moving amount (mm)	0.001	0.01	0.1

- 2 The value in the table varies with the mechanical transmission. See details in the machine builder's manual;
- 3 The speed of the handwheel rotated should be less than 5 r/s. If not, there may be inconsistent between the scale and the moving amount.

## 6.2 Control in MPG interruption

### 6.2.1 MPG interruption operation

MPG interruption operation can be overlapped with the automatic movement in Auto mode.

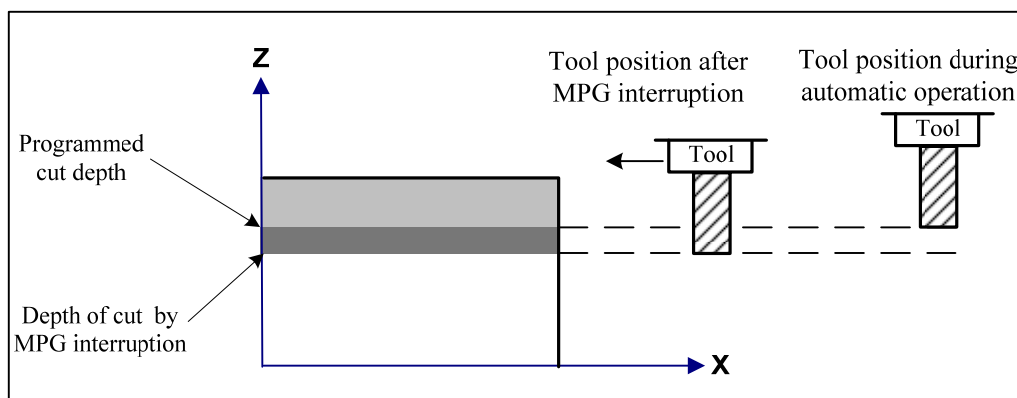


Fig. 6-2-1-1

Operation steps:

- 1) After the dwell of the program execution in Auto mode, switch over the control to MPG mode.
- 2) Move the tool position by handwheel, move Z axis downward or X, Y axis parallel to modify the coordinate system.
- 3) It starts after the control is switched to Auto mode, the workpiece coordinates remain unchanged till the coordinates restore to their actual values after the machine zero return operation.

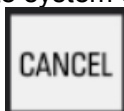
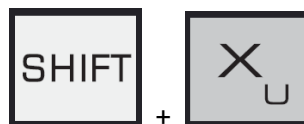
**Note: whether the MPG /Step interruption function is used is set by bit parameter NO:56#3.**

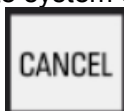
As the program run in Auto, MDI, DNC mode is shifted to MPG mode by dwell, the control will execute the MPG interruption. The coordinate system for MPG interruption is shown in Fig.6-2-1-2.

ACTUAL POSITION				00002 N0120	
(RELATIVE)		(ABSOLUTE)		(MACHINE)	
X	0.0000	X	0.0000	X	0.0000
Y	0.0000	Y	0.0000	Y	0.0000
Z	0.0000	Z	0.0000	Z	0.0000
(HANDLE INTR)		(SUBSPEED)			
X	0.0000	X	0.0000		
Y	0.0000	Y	0.0000		
Z	0.0000	Z	0.0000		
				S00000 T0100	
				MPG	

Fig.6-2-1-2

The MPG interruption coordinate system clearing steps: press



till "X" flickers, then press  key, the coordinate system will be cleared. The operations of Y, Z are the same as above; while the zero returning is being performed, the coordinate system is cleared automatically.

## 6.2.2 Relation of MPG interruption with other functions

Display	Relation
Machine lock	If machine lock is effective, the machine move is ineffective in MPG interruption.
Absolute coordinate value	MPG interruption does not change the absolute coordinates.
Relative coordinate value	MPG interruption does not change the relative coordinates.
Machine coordinate value	The changing amount of machine coordinate is the displacement amount induced by MPG rotation.

**Note** The moving amount of MPG interruption is cleared when the manual reference point return is performed by each axis.

## 6.3 Auxiliary control in MPG mode

The auxiliary operation in MPG mode is identical with that in JOG mode. See Section 4.2 and 4.3 for details.

## 6.4 MPG drive function

### **Operation method:**

Enable the electronic MPG drive function by setting bit parameter NO:59#1. In Auto mode, turn on Dry Run, press key <CYCLE START>, and control the execution of the part program by rotating the MPG. The execution speed of the program becomes faster as the MPG is rotated faster, and vice versa. This function is usually used for workpiece trial cutting and machining program detection.

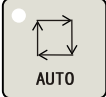
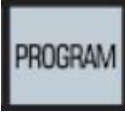

Note 1: The Dry Run is ineffective after the electronic MPG drive function is enabled.

Note 2: Single block stop execution is effective in single block mode.


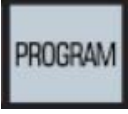


# 7 Auto Operation

## 7.1 Selection of the auto run programs


### 1 Program loading in auto mode

- (a) Press  key to enter the Auto mode;
- (b) Press  key to enter the program page, move the cursor to find the target program;
- (c) Press  key for confirmation.

### 2 Program loading in Edit mode


- (a) Press  key to enter the Edit mode;
- (b) Press  key to enter the program page, move the cursor to find the target program;
- (c) Press  key for confirmation.
- (d) Press  key to enter the Auto mode;

## 7.2 Auto run start


After select the program by the two ways of section 7.1 above, press  key to execute the program, the program execution can be viewed by switching to <POSITION>, <MONI><GRAPH> etc. interfaces.

The program execution is started from the line where the cursor locates, so check that



whether the cursor is located at the program to be executed before pressing the  key. If the cursor is not located at the start line from which the program is to be executed, press reset key



, then press  key to run the program automatically from the start line.

**Note: the workpiece coordinate system and basic offset value can not be modified during the program running in Auto mode**


### 7.3 Auto run stop

In Auto run, to make the program being executed automatically to be stopped, five ways are provided in this system:

#### 1 Program stop (M00)

After the block containing M00 is executed, the auto running pauses and the modal message




is saved. If  key is pressed, the program execution is continued.

#### 2 Program optional stop (M01)




If the  key is pressed during the program execution, the automatic running pauses

and the modal message is saved when the block containing M01 is being executed. If  key is pressed, the program execution is continued.




#### 3 Press key



If the  key is pressed during the auto running, the machine status is:

- 1) Machine feeding slows down and stops;
- 2) Dwell continues if Dwell is being executed (G04 instruction) ;
- 3) The remaining modal message is saved;

- 4) The program execution is continued after  key is pressed.



- 4 Press  key

See Section 2.3.1 of this manual.

- 5 Press EMERGENCY STOP button


See Section 2.3.2 of this manual.

In addition, if the control is switched to other mode from Auto mode, DNC mode, MDI interface of MDI mode in which the program being executed, the machine can also be stopped. The steps are as following:

- 1) If the control is switched to Edit, MDI, DNC mode, the machine stops after the current block is executed.
- 2) If the control is switched to JOG, MPG, Step mode, the machine interruption stops immediately.
- 3) If the control is switched to Machine zero interface, the machine slows down to stop.


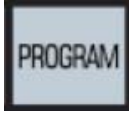
## 7.4 Auto running from an arbitrary block

This system permits the current program to be executed from an arbitrary block of it. The steps are as following:

1. Start spindle and other miscellaneous functions by pressing  key to enter JOG mode;

2. Ensure that the program modal values are correct when they are being ran In MDI mode



3. Press  key to enter Edit mode, then press  key to enter program page, select the program to be executed in 【DIR】 ;

4. Open the program and move the cursor to the block to be executed;



5. Press  key to enter Auto mode;



6. Press  key to execute the program automatically.





- Note 1** Before execution, confirm the current coordinate point to be the end of the block preceding to the block to be executed (confirmation of the current coordinate point is unnecessary if the block to be executed is absolute programming and contains G00/G01);
- Note 2** If the block to be executed is tool change operation etc, ensure that the interference between the tool and the workpiece at current position, which may cause machine damage or personnel hurt, will not occur.

## 7.5 Dry run

Before the program execution, a dry run can be performed to have a check for the program, which is usually used together with "MACHINE LOCK", "M.S.T. LOCK".



Press  key to enter Auto mode, press  key (the Dry Run indicator in panel lighting up means the current mode is DRY RUN).

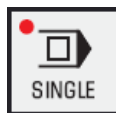
In rapid feed, the program speed is Dry Run speed × rapid override

In cutting feed, the program speed is Dry Run speed × feedrate override

- Note 1** The dry run speed is set by the number parameter P86;
- Note 2** In rigid taping, whether the Dry Run is effective is set by bit parameter NO:12#5;
- Note 3** The effectiveness of dry run in cutting feed is set by the bit parameter NO:12#6.
- Note 4** The effectiveness of dry run in rapid positioning is set by the bit parameter NO:12#7.


## 7.6 Single block running

"Single Block" can be selected for checking a block execution.



In Auto, DNC or MDO mode, press  key (The SINGLE BLOCK indicator in panel lighting up means the current mode is Single Block). In this mode, the system stops after a block is




executed. Press  key to go on next block execution, perform the operation repeatedly till the whole program is executed.

- Note 1** In G28 mode, the single block stop can be performed at an intermediate point.

## 7.7 Running with machine lock



In Auto mode, press  key (The MACHINE LOCK indicator in panel lighting up means the current mode is Machine lock). In this mode, the machine axes don't move. But the position coordinates displayed are the same as that during machine moving. And M, S, T are effective too. This function is used for program verification.


**Note** Due to that the machine position is not consistent with its coordinate position



after  key is pressed and program running, the machine zero operation is needed to be performed.

## 7.8 Running with M.S.T. lock



In <AUTO> mode, press  key (The M.S.T. LOCK indicator in panel lighting up means the current mode is M.S.T. LOCK). In this mode, the M, S, T instructions are not executed. This function is used for program verification together with the Machine Lock.

**Note** M00, M30, M98, M99 is executed by convention.

## 7.9 Feedrate and rapid override in Auto run


In <AUTO> mode, the feedrate and rapid traverse speed can be overridden by the system.

In Auto run, the feedrate override classified for 16 gears can be selected by pressing




keys.



The feedrate override ascends for a gear (10%) till 150% each time the  key is pressed;



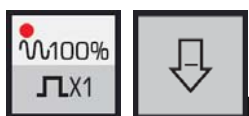
The feedrate override descends for a gear (10%) till 0 each time the  key is pressed. If the override is F0, the axis stop is set by bit parameter **N0:12#4**. If it is set for non-stop 0, the actual rapid traverse speed is set by number parameter P93 (for all axes).


**Note 1** F value in feedrate override program:

**The actual feedrate = F value specified × feedrate override**



During auto running, the rapid traverse speed can be selected by pressing



 key. The 4 gears override of F0, 25%, 50%, 100% are available for the rapid traverse.

**Note 2** The rapid traverse speed value overridden by rapid override and number parameters P88、P89、P90 can be obtained by following equation:

**The actual rapid traverse speed of X axis = the value set by parameter P88 × rapid override**

The actual rapid traverse speed of Y or Z axis is as above.

## 7.10 Spindle override in Auto run

In auto run, the spindle speed can be overridden if it is controlled by analog quantity.




The spindle speed can be overridden by pressing spindle override keys




 in auto mode, which are classified for 16 gears from 0%~150%.



The spindle override ascends for a gear(10%) till 150% each time the  key is pressed;



The spindle override descends for a gear(10%) till 0% each time the  key is pressed.



The actual spindle speed=speed specified× spindle override

The max. spindle speed is set by number parameter P258, if the spindle speed exceeds the max. value set, it uses the max. speed.

## 7.11 Background edit in Auto run

The background edit function in processing is supported in this system.


During the program execution in Auto mode, press <PROGRAM> key to enter the program

page, then press  +  soft key to enter the background edit interface, as is shown in Fig.7-11-1.



**Fig. 7-11-1**

Enter the program background edit interface, and the program editing operation is the same

as that in Edit mode (Refer to Chapter 10 **Program Edit** in this manual). Then press  soft key to save the edited program and exit this interface.

Note: it is suggested that the background editing file size should be not more than 3000 lines, otherwise, the machining effecte will be affected.

## 8 MDI Operation

Except the input, modification, offset operations in MDI mode, the MDI running function is also available in this system. By this function the instructions can be input directly for execution. The input, modification, offset operations etc. are introduced in Chapter 3 “Page display as well as data modification and setting”. This chapter will describe the MDI running function in MDI mode.


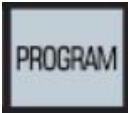


### 8.1 MDI instructions input

The input in MDI mode is classified for two types:

- 1 By **【MDI】** type, multiple blocks can be input continuously;
- 2 By **【CUR/MOD】** type, only one block can be input.

The input in **【MDI】** is identical with the program input in Edit mode, see Chapter 10 Program Edit in this manual for details. The **【CUR/MOD】** input is introduced as following.



Example: To input a block “G00 X50 Y100” in **【CUR/MOD】** page, the steps are:

- 1) Press  key to enter the MDI mode;
- 2) Press  key to enter the Program page, press **【CUR/MOD】** soft key to enter the **【CUR/MOD】** page (see Fig.8-1-1):  

- 3) Key in the block “G00X50Y100” by sequence and press  key to confirm, then the block will be displayed on the page (see Fig. 8-1-1):

PROGRAM ( CURRENT/MODAL )				00002 N00000			
( CURRENT )				( MODAL )			
G00	X	50.000	P	G00	G11	F	300
	Y	100.000	Q	G17	G98	S	0
	Z		F	G90	G15	M	30
	*		L	G94	G50	T	0000
	*		S	G54	G69	H	0000
	R		M	G21	G64	D	0000
	I		T	G40	G97		
	J		H	G49			
	K		D				
DATA:				S00000 T0000			
				MDI			



Fig. 8-1-1

8.2 Run and stop of MDI instructions

After the instructions are input by the steps in section 8.1, press  key to run the MDI instructions. During the running, the instructions execution can be stopped by pressing  key.

- Note 1 MDI running must be performed in MDI mode.
- Note 2 The program input in 【CUR/MOD】 interface is executed prior to that input in MDI mode.

8.3 Words modification and clearing of MDI instructions

If an error occurs during word inputting, press  key to cancel the whole block input; if the error is found after the input is finished, reinput the correct words to replace the wrong ones or press  key to clear all for reinputting.

## 8.4 Modes changing

When the control is switched to MDI, DNC, Auto, Edit mode during the program execution in Auto, MDI, DNC mode, the system will stop the program execution after the current block is executed.

When the control is switched to Step mode by a dwell during the program execution in Auto, MDI, DNC mode, it will execute the step interruption, see section 5.2 Step interruption. If the control is switched to MPG mode by a dwell, it will execute MPG interruption, see section 6.2 MPG interruption. If the control is switched to JOG mode by a dwell, it will execute manual intervention, see section 4.1.4 Manual interruption.

When the control is switched to Step, MPG, JOG, Machine Zero mode during the program execution in Auto, MDI, DNC mode, the system will execute deceleration and stop.

# 9 Machine Zero Operation

## 9.1 Conception of machine zero

The machine coordinate system is the inherent coordinate system by machine. Its origin is called mechanical zero (or machine zero), as is called reference point in this manual. It is usually fixed at the maximum stroke point of X axis, Y axis or Z axis. This origin that is a fixed point is set after the machine is designed, manufactured and adjusted. As the machine zero is not confirmed by the CNC system at power-on, the auto or manual machine zero return is usually performed.

The machine zero return has two types: one-revolution-signal, non-one-revolution-signal. It is set by bit parameter No.6#6. For the zero return of the non-one-revolution-signal by the motor, it is classified for the A, B two types. It is set by bit parameter No.6#7.

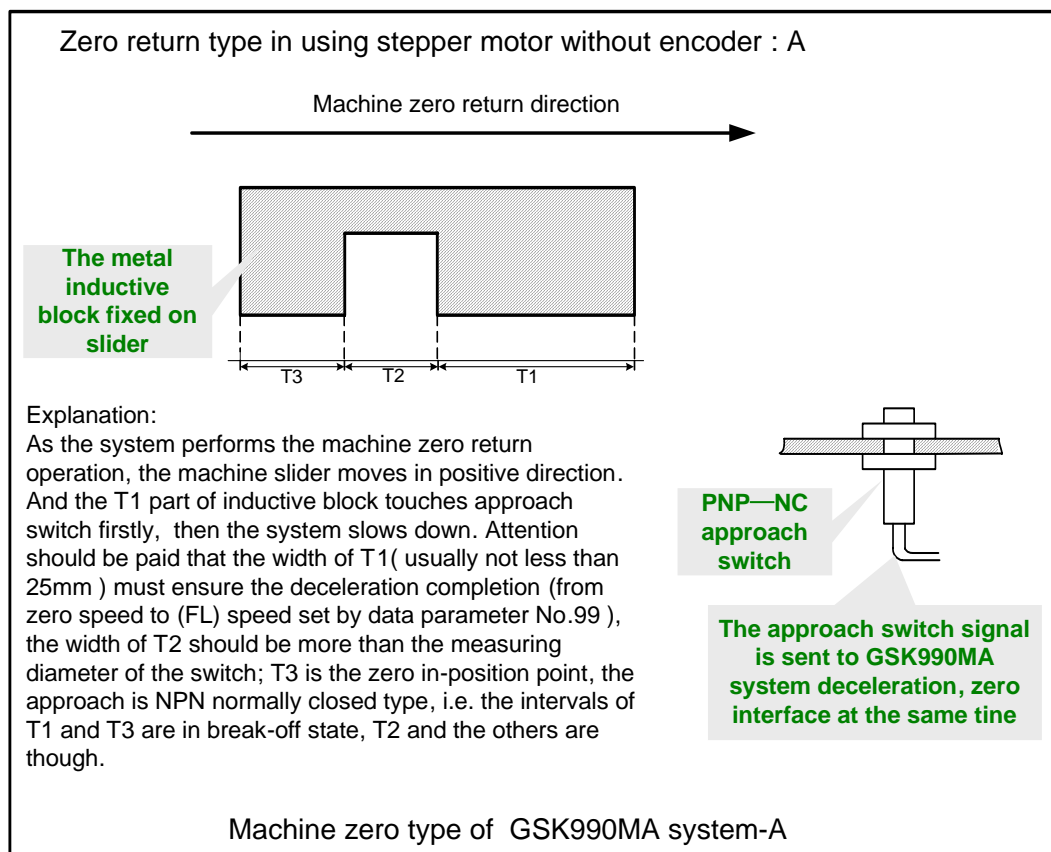


Fig. 9-1-1



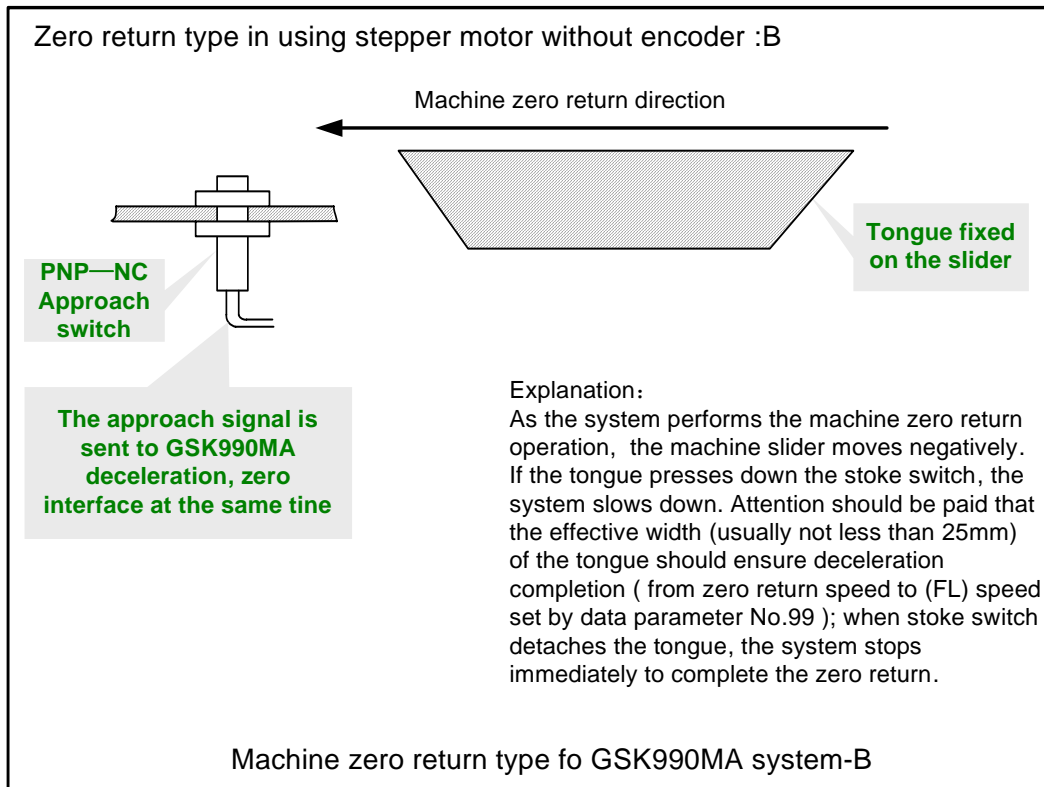
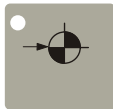
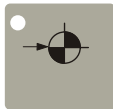


Fig. 9-1-2

## 9.2 Steps for machine zero



- 1 Press  to enter Machine Zero mode, the characters“machine zero”will be displayed at the down-right of the LCD screen;
- 2 Select the axis X, Y, or Z for machine zero and its direction is set by bit parameter **N0:7#3~N0:7#5**;
- 3 The machine moves towards the machine zero. Before the deceleration point is reached the machine traverses rapidly(traverse speed set by number parameter P100~P102), then moves to the machine zero point(i.e. reference point ) by a speed of FL(set by number parameter P99) if the machine touches the deceleration switch. As the machine zero is reached, the corresponding axis moving stops and the Machine Zero indicator lights up.

### 9.3 Machine zero steps by program

After the bit parameter No:4#3 is set for 0, the machine zero can be specified by G28 instruction. Because it detects the stroke tongue, this instruction is equivalent to manual machine zero.

**Note 1** If the machine zero is not fixed on your machine, don't perform the machine zero operation.

**Note 2** The indicator of the corresponding axis lights up when the machine zero is finished.

**Note 3** The indicator is gone out on condition that the axis is moved out from the machine zero by the operator.


**Note 4** Refer to the machine builder's manual for the direction of the machine zero (reference point).


# 10

## Edit Operation

### 10.1 Program edit

The part program edit should be operated in Edit mode. The Edit mode can be entered by

pressing  key.

Press  key to enter the program edit and modification interface, as is shown in Fig.10-1-1:

```

PROGRAM                                00002 N00120
000002;
N0060 X100;
N0120 X0;
N0180 G01 X50 Y50 F2000 ;
// N0240 G41 X100 D1
N0300 G01 Y100
N0360 G02 X200 R50
N0420 G01 Y0 F2500
N0180 X0
N0180 Y50
DATA:                                Ln: 2          S00000 T0000
                                           EDIT

```

**Fig.10-1-1**

The switch of the program must be opened before program edit. See the section 3.5.2 Parameter and program switch in this manual for its operation.

**Note1:** The maximum lines a program file contains are 200,000.

**Note2:** As is shown in fig. 10-1-1, if there is more than 1 sign “/” ahead of a block, the system will skip the block even if the block skip function is not turned on.

#### 10.1.1 Program creation

##### 10.1.1.1 Auto creation of the sequence number

Set the “auto sequence number” for 1 by the steps in section 3.5.1 (See Fig. 10-1-1-1) :

SETTING	00002	N00120
MIRROR X =	1	(0: OFF 1: ON)
MIRROR Y =	1	(0: OFF 1: ON)
MIRROR Z =	1	(0: OFF 1: ON)
INCH PRG =	0	(0: METRIC, 1: INCH)
I/O CHAN. =	0	( 0—3 CHANNEL NO. )
ABS PRG =	0	(0: ABS, 1: INC)
AUTO SEQ =	0	(0: OFF 1: ON)
SEQ STOP =	0000 (PROGRAM NO.)	0000 (SEQUENCE NO.)
2007 Y 8 M 8 D	8 H 8 M 8 S	
DATA	S00000	T0010
	MDI	

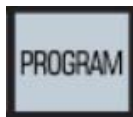
Fig. 10-1-1-1

Therefore the sequence number will be automatically inserted into the blocks during editing.  
The incremental amount of the sequence number is set by number parameter P210.

### 10.1.1.2 Program input and save







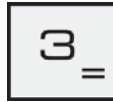
1、 Press key to enter Edit mode;



2、 Press key to enter program page (See Fig. 10-1-1-2-1);

PROGRAM	00002	N00120
000002;		
N0060 X100;		
N0120 X0;		
N0180 G01 X50 Y50 F2000 ;		
N0240 G41 X100 D1		
N0300 G01 Y100		
N0360 G02 X200 R50		
N0420 G01 Y0 F2500		
N0180 X0		
N0180 Y50		
DATA:	Ln: 2	S00000 T0000
		EDIT

Fig. 10-1-1-2-1

3. Press address key , then key in numerical keys     by sequence (an example by setting up a program named O00003), it displays O00003 behind the data column (See Fig. 10-1-1-2-2) :

```

PROGRAM                                00002 N00120
00002;
N0060 X100;
N0120 X0;
N0180 G01 X50 Y50 F2000 ;
N0240 G41 X100 D1
N0300 G01 Y100
N0360 G02 X200 R50
N0420 G01 Y0 F2500
N0180 X0
N0180 Y50
DATA: 00003                Ln: 2        S00000 T0000
                                EDIT

```

Fig. 10-1-1-2-2


- 4 Press  key to set up the new program name, it displays (Fig. 10-1-1-2-3):


```

PROGRAM                                00003 N00120
00003;
%
DATA:                                Ln: 2        S00000 T0000
                                EDIT

```

Fig. 10-1-1-2-3

- 5 Input the blocks programmed word by word, after inputing, the program will be saved automatically while switching to other modes. If switching to the other interface ( e.g. 

- page), press  key to save the program before the operation and finish the block inputing.

**Note 1:** In Edit mode, only the complete word can be entered. Single letter and numerical number input is not supported by system.



**Note 2:** If a wrong instruction word is detected during program inputting, press key to cancel the instruction.

**Note 3:** No more than 74 characters can be input in one block each time.

### 10.1.1.3 Search of sequence number, word and line number


Sequence number search operation is usually used to search for a sequence number in a program so that the execution and edit can be started from the block containing this sequence number. Those blocks that are skipped because of the the search do not affect the CNC. (This means that the data in the skipped blocks such as coordinates, M, S, T and G codes does not affect the CNC coordinates and modal values.)



If the execution needs to be done from a searched block in a program, specify M, S, T and G codes, coordinates and so forth as required (by MDI) after closely checking the machine and CNC states at that point.

The word search function is used to search a special address word or number in a program, and it is usually used for editing.

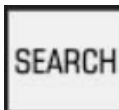
Steps for sequence number, line number or word search:


- 1 Enter <Edit > mode
- 2 Look up the target program in 【PROGRAM (DIR)】 page;

- 3 Press  key to enter the target program;

- 4 Key in the word or sequence number to be searched and press  or  keys to look for it.

- 5 If the line number in program is needed to be searched, press

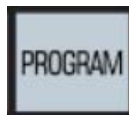


key, key in the line number to be searched and press  key for confirmation.


**Note 1** The search function is automatically cancelled when the search for sequence number and word is performed to the end of a program.


- 2 The searching for sequence number, word and line number can be performed in either 【AUTO】 or 【EDIT】 mode, but in 【AUTO】 mode, it can only be performed in the background edit page.


#### 10.1.1.4 Location of the cursor





Select Edit mode, then press  key to enter the 【PROGRAM】 page.


- a) Press  key to shift the cursor upward for a line, if the column where the cursor locates exceeds the end column of the last line, the cursor moves to the end of the last line.


- b) Press  key to shift the cursor downward for a line, if the column where the cursor locates exceeds the end column of the next line, the cursor moves to the end of the next line.



- c) Press  key to move the cursor for a column to the right, if the cursor is at the line end, it moves to the beginning of the next line.

- d) Press  key to move the cursor for a column to the left, if the cursor is at the beginning of the line, it moves to the end of the last line.




- e) Press  key to page up, and the cursor will locate to the same position of the previous page

- f) Press  key to page down, and the cursor will locate to the same position of the next page

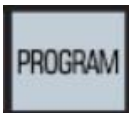
- g) Press  key, the cursor moves to the beginning of the line it locates.

- h) Press  +  keys, the cursor moves to the beginning of the

program.

- i) Press  key, the cursor moves to the end of the line it locates.
- j) Press  +  keys, the cursor moves to the end of the program.


### 10.1.1.5 Insertion, deletion and modification of word

Select <EDIT> mode, then press  key to display the program page. Locate the cursor to the position to be edited.

#### 1. Word insertion


After keying in the data, press  key, the data will be inserted to the left of the cursor.

#### 2. Word deletion

Locate the cursor to the word to be deleted, press  key, the word will be deleted.


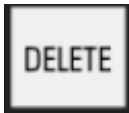
#### 3. Word modification

Move the cursor to the place to be modified, and key in the new content, then press

 key to replace the old content by the new one.

### 10.1.1.6 Deletion of a single block

Select <EDIT> mode, then press  key to display the program page. Locate the

cursor to the beginning of the block to be deleted. Press  +  keys to delete the block where the cursor locates.



**Note** N could be keyed in to delete the block whether the block is headed with sequence number (cursor heading the line).

### 10.1.1.7 Deletion of multiple blocks

The blocks from the currently displayed word to the specified sequence number block can be deleted.

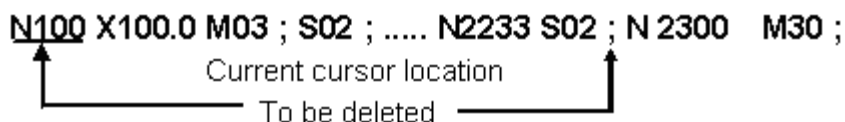
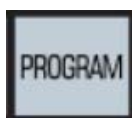
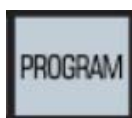
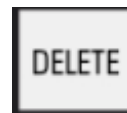
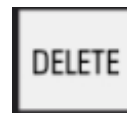


Fig. 10-1-1-7-1



Select <EDIT> mode, press  key to display the program. Locate the cursor to the beginning of the target to be deleted (position of word N100 as figure above), then key in the last



word of the multiple blocks to be deleted, e.g. **S02** (as Fig.10-1-1-7 above), press  key to delete the blocks from the current cursor location to the address specified.


**Note 1** The blocks that can be deleted are two hundred thousand lines at most.


**Note 2** If several words to be deleted are same in program, it will delete the blocks to the word nearest to the cursor location.

### 10.1.2 Deletion of a single program

The steps for deleting a program in memory are as follows:

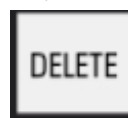
- a) Select <EDIT> mode;
- b) there are two ways to delete program;

Method1: Enter the PROGRAM display page and Key in the address key ; key in

the program name (e.g. For O0002 program, key in the numerical key .




); press



key, the corresponding

program in memory will be deleted.

Method 2: Select 【DIR】 page in program interface, then select the program name to be


deleted by moving cursor and press  key, the program selected will be deleted.

Note: For program O0001, just the program can be deleted, the program number can not be deleted.

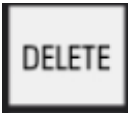
### 10.1.3 Deletion of all programs

The steps for deleting all programs in the memory are as follows:

- a) Select <EDIT> mode;
- b) Enter the program page;

c) Key in the address key ;


d) Key in the address keys , , , ,  by sequence;

e) Press  key, all the programs in the memory will be deleted.


### 10.1.4 Copy of a program

Steps for copying current program and saved for a new name:

- a) Select <EDIT> mode;
- b) Enter the 【PROGRAM ( DIR )】 page and select the program to be copied by cursor keys,

then press  key to enter the program page;

c) Press address key  and key in the new program number;


d) Press the  key, the file will be copied and the control enters the new program edit page.



- e) Return to 【PROGRAM ( DIR )】 page, the name of the new program copied can be viewed.

### 10.1.5 Copy and paste of blocks

The steps for program copy and paste are as following:

- a) Locate the cursor to the beginning of the blocks to be copied;  
b) Key in the last character of the blocks to be copied;

- c) Press  key, the blocks from the cursor to the character keyed in will be copied.

- d) Locate the cursor to the position to be pasted, press  +  key to complete the paste.

**Note1:** If several words to be cut are same in program, it will cut the blocks to the word nearest to the cursor location.

**Note 2:** If the blocks are copied with method N+sequence number, the blocks from the cursor to the N + sequence number are copied.



**Note 3:** 10,000 lines of blocks can be copied at most.

### 10.1.6 Cut and paste of block

Steps of block cut are as following:

- a) Enter the program edit page(as Fig.10-1-1);  
b) Locate the cursor to the beginning of the block to be cut;  
c) Key in the last character of the block to be cut;

- d) Press the  +  key , the block will be cut into clip board.

- e) Locate the cursor to the position to be pasted, and press  +  key, the block will be pasted.

**Note1:** If several words to be cut are same in program, it will cut the blocks to the word nearest to the cursor location.


**Note 2:** If the blocks are cut with method N+sequence number, the blocks from the cursor to the N sequence number are cut.

### 10.1.7 Replacement of the blocks

Steps of block replacement are as following:

- a) Enter the program edit page (Fig.10-1-1);
- b) Locate the cursor to the character to be altered;
- c) Key in the new character;



- d) Press the  key, the character where the cursor locates will be replaced by the new one.

Note: This operation is just for the characters, but not for the whole block.

### 10.1.8 Rename of a program


Rename the current program:

- a) Select <EDIT> mode;
- b) Enter the program page(cursor specifies the program name );



- c) Key in address key , key in the new name;



- d) Press  key to complete the renaming.



### 10.1.9 Program restart

The function is used in the event of an accident such as tool fracture, system restarting after power-off or emergency stop during program execution. After the accident is eliminated, the system returns to the program breakpoint by program restart to continue the program execution, and then it retracts to original point by Dry Run.

Steps for program restart are as follows:

1. Solve the machine accident such as tool change, offset changing, machine zero return and so on.



2. Return to <AUTO> mode, press key , and then key  on the panel. Then the program moves to the start point (i.e. the end point of the last block) of the interrupted block at the dry run speed and the execution continues . The operation can be restarted anywhere.



**Note 1:** The sequence in which the axes moves to the program restart position. They are set by data parameter P376.

**Note 2:** Check whether the collision occurs when the tool moves to the program restart position. If such a possibility exists, move the tool to the place where no obstruction occurs and then perform restart.

**Note 3:** When the coordinate axis restarts the position moving to switch on the single block running, the tool stops each time it finishes an axis movement.

**Note 4:** If there is no absolute position detector, the reference point return must be performed before the restart after power-on.

**Note 5:** Do not perform the resetting during the program execution from block research at restarting to restarting, or the restarting must be done from the first step.

**Note 6:** The restart function of the system does not support the program containing subprograms currently.

## 10.2 Program management

### 10.2.1 Program directory search

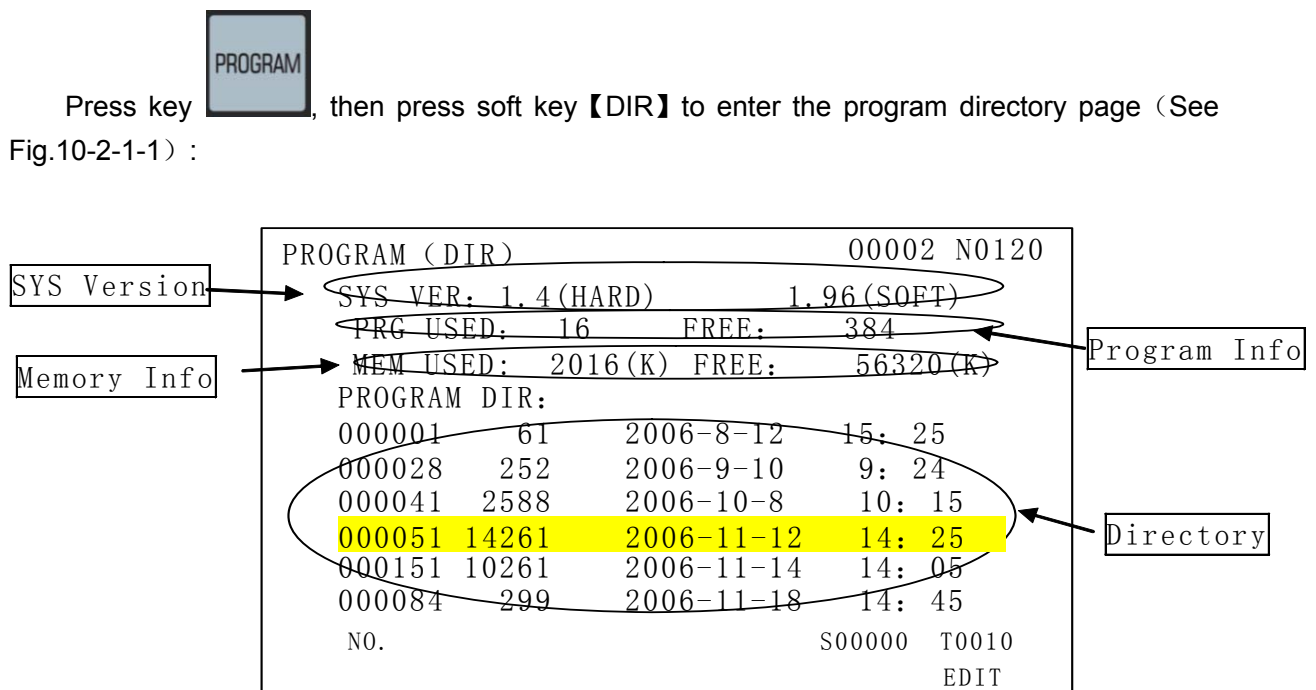



Fig.10-2-1-1

1) Open the program

Open the program specified:  + sequence number +  key (or



key) or sequence number +



key (or



key)

In Edit mode, if the sequence number inputted does not exist, a new program will be created.

## 2) Deletion of the program:

1. Edit mode Press key to delete the program where cursor locates
2. Edit mode + sequence number + key or sequence number + key to delete sequence program.

### 10.2.2 Number of the program stored

The maximum number of the programs stored in this system is 400. Look up in Fig. 10.2.1 above for the message on the number of the program currently stored in the program directory page.

### 10.2.3 Memory capacity

Look up in Fig.10.2.1 above for the message on memory capacity in the program directory page.

### 10.2.4 Viewing of the program list

A program directory page can display 6 CNC program names at most. If the CNC programs are over 6, they can't be fully displayed in a page, so press the PAGE key to display the program names on the next page. If the page key is pressed continuously, all the CNC program names will be displayed by cycle on LCD.



Because the programs are listed by their name sizes, press key to view them and the programs will be listed by the date sequence with the latest modified program headed.

### **10.2.5 Program lock**

The program switch is set in this system to protect the user programs to be modified by unauthorized personnel. After the program editing, set the program switch for OFF to lock the program. And the program edit is disabled. See Section 3.5.1 for its explanation.

# 11

## Communication

This system can communicate with PC or USB via its own interfaces to realize data transmission and DNC on-line machining.

### 11.1 Serial communication

Preparation for serial communication of GSK-990MA CNC system:

1. Connect the PC serial port (COM port ) and system RS232 interface with a serial line.
2. Open GSK Com serial communication software on PC side.

**Note: GSK Com serial communication software uses Windows interfaces. This software is suitable for Win98, WinMe, WinXP and Win2000 system.**

3. Setting for GSK Com serial communication software:
  - (1) Select "Suitable for GSK990MA";
  - (2) Click "Series Port" menu, and set baudrate in "Serial Setting" dialog. For data transmission, select the baudrate of 115200 (corresponding to the default set by data parameter P002); For DNC on-line machining, select the baudrate of 38400 (corresponding to the default set by data parameter P001)

#### 11.1.1 Program start

Run the CommGSK990MA.exe program directly. The interface of it is as following:



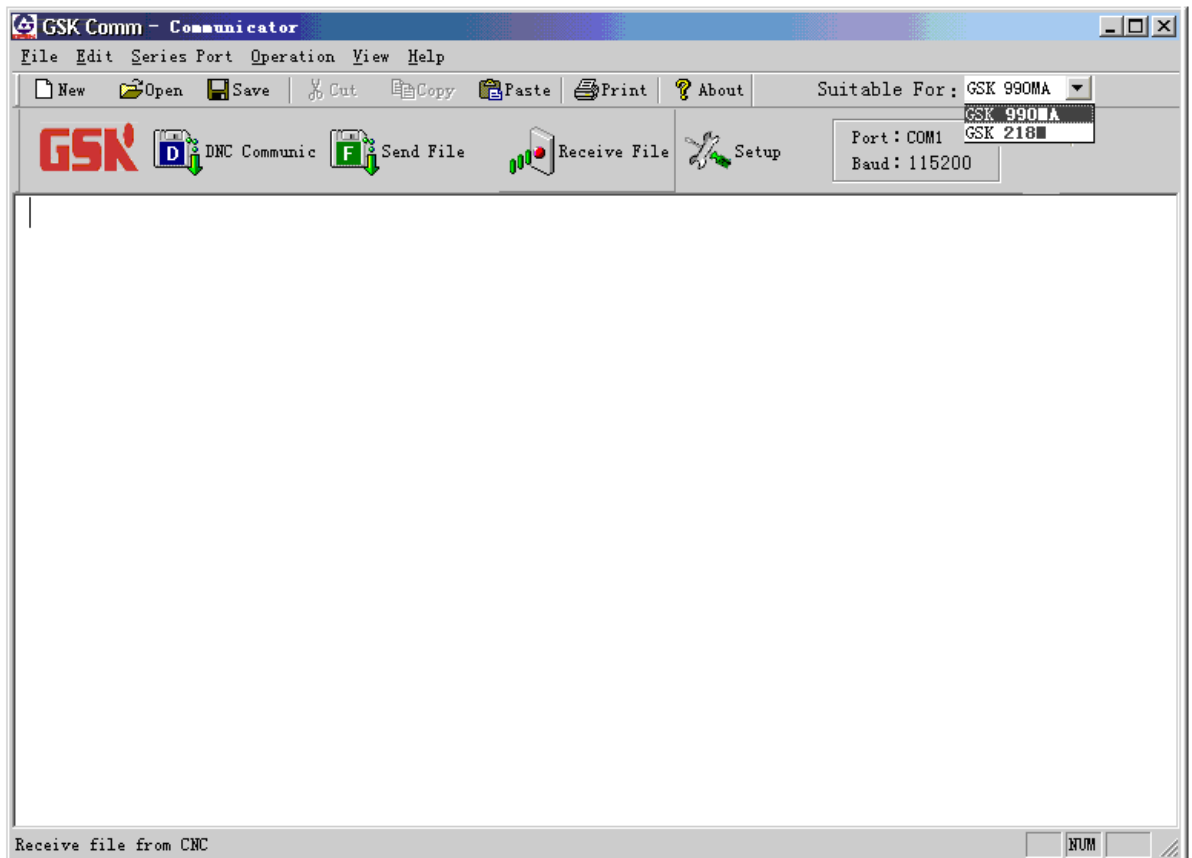


Fig. 11-1-1-1

### 11.1.2 Function introduction

#### 1 File menu

The file menu involves the functions of File Creation, Open, Save, Print and Print setting and the latest the file list etc.

#### 2 Edit menu

The edit menu involves the function such as Cut, Copy, Paste, Retraction, Find, and Replace.

#### 3 Serial menu

It is mainly used for the open and setting of the serial ports.

#### 4 Transfer menu

It involves the transfer types of DNC, file sending, and file receiving.

#### 5 View menu

It is used for the display and hiding of the tool column and state column.


#### 6 Help menu


It is used to view the software version.

### 11.1.3 Serial port data transmission

Operation steps are shown as follows:


- 1) Select <MDI> mode;

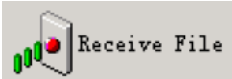
- 2) Press  key to enter setting page, set the I/O channel to 0 or 2. (With I/O channel set to 0, select Xon/Xoff for DNC protocol; with I/O channel set to 2, select XModem for DNC protocol)

- 3) Press soft key  to switch to Setting (Password) page, and then input corresponding password authority.

Target file	Password authority
System parameter value	Password for system debugging level, password for machine tool builder level, password for system manufacturer level
Tool offset value	Password for system debugging level, password for machine tool builder level, password for system manufacturer level
Pitch offset value	Password for system debugging level, password for machine tool builder level, password for system manufacturer level
Custom macro variable	Password for system debugging level, password for machine tool builder level, password for system manufacturer level
Parameters	Password for machine tool builder level, password for system manufacturer level
Ladder (PLC)	Password for machine tool builder level, password for system manufacturer level
CNC part program	Without a password

#### A. Data output (CNC→PC):

1. Press  key to enter setting (data processing ) page, according to the content being transferred, move red symbol to “Data input” dialog box by direction key, and press **【INPUT】** key, then the system prompts “Transfer waiting”

2. Click button  on GSK Com serial communication software, then “Receive File” dialog pops up, as is shown in fig. 11-1-3-1.

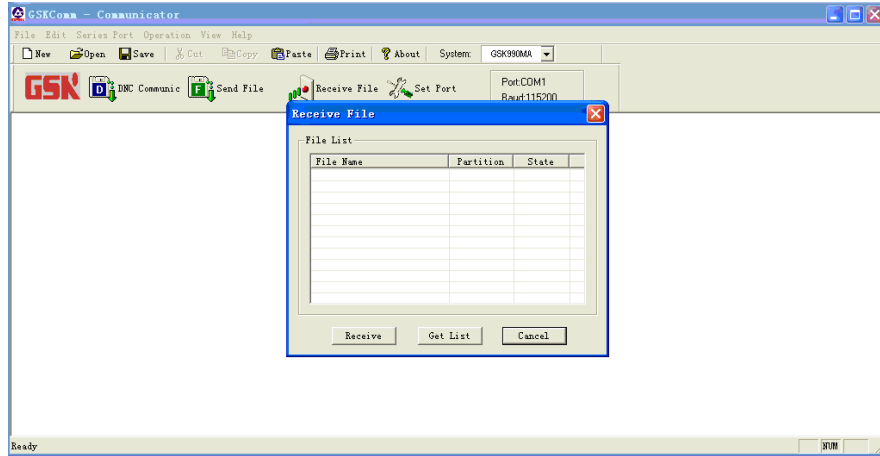



Fig. 11-1-3-1

3. Click button  in Receive File dialog to obtain the CNC file list, as is shown in fig. 11-1-3-2:

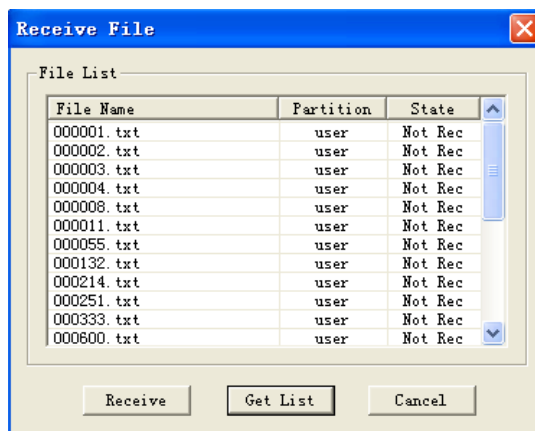



Fig. 11-1-3-2

4. Select the file (or multiple files) to be received, then press button  to start the file receiving, as is shown in fig. 11-1-3-3:

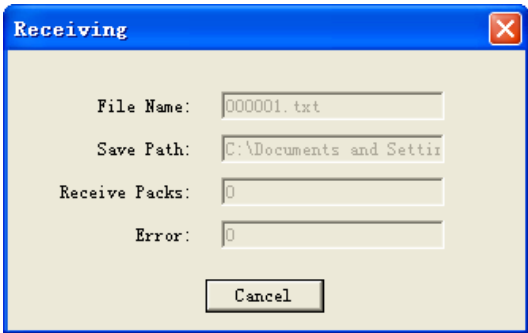


Fig. 11-1-3-3

5. After the file receiving, the status bar of the dialog displays “Received”, as is shown in Fig.11-1-3-4:

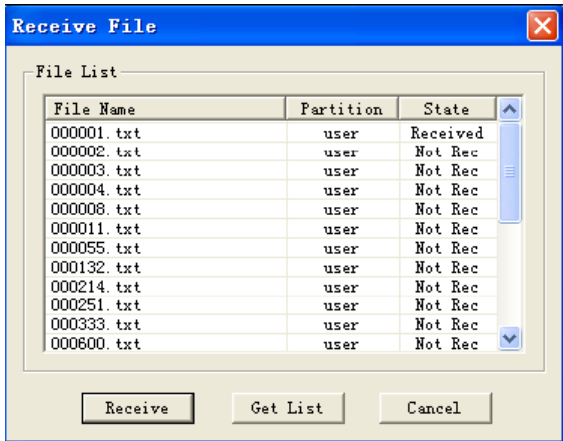
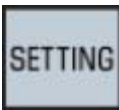
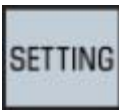


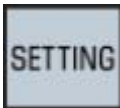
Fig.11-1-3-4


**B. Data input (PC→CNC):**




1. Press  key to enter setting (data processing ) page, press **【INPUT】** key, then the system prompts “Transfer waiting”

**A. Data input (PC→CNC):**



1. Press  key to enter setting (data processing ) page, according to the content being transferred, move red symble to “Data input” dialog box by direction key, and press **【INPUT】** key, then the system prompts “Transfer waiting”

2. Click button  (or press “Send File” in the down menu of “OPERATION”) to

pop up Send File Dialog in the GSK com serial communication software, as is shown in fig. 11-1-3-5.

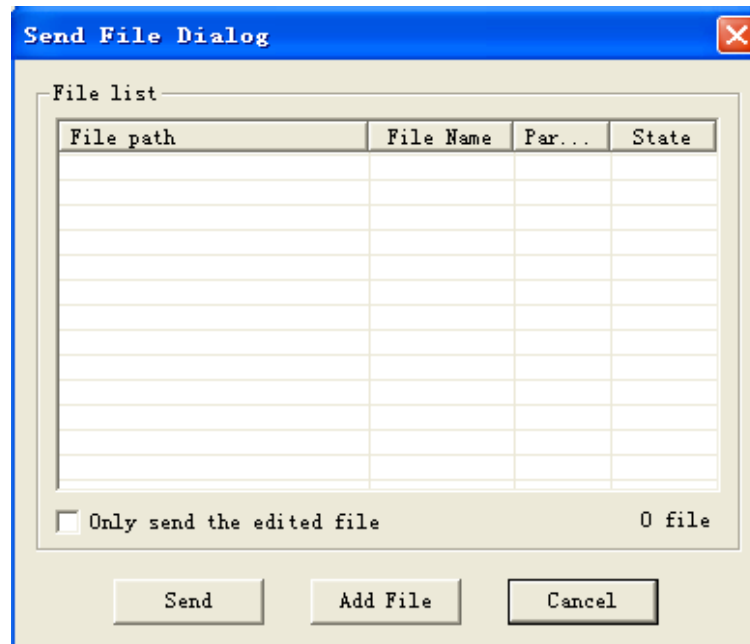



Fig.11-1-3-5

3. Click button  in the “Send File” dialog, then the “Select Part Dialog” pops up as in fig. 11-1-3-6.

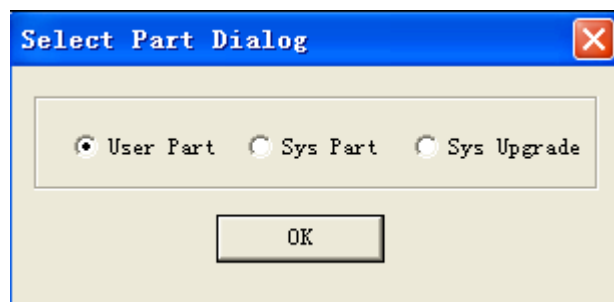



Fig.11-1-3-6

4. In the “Select Part Dialog”:  
Select “User Part” when sending CNC part programs and custom macro programs; select “System Part” when sending files such ladder (PLC), parameters (PLC), system parameter values, tool offset values, pitch offset values and system macro variables.
5. After selecting the partition, select the file (or multiple files) to be sent, and click button  to start the file sending, as is shown in fig. 11-1-3-7.

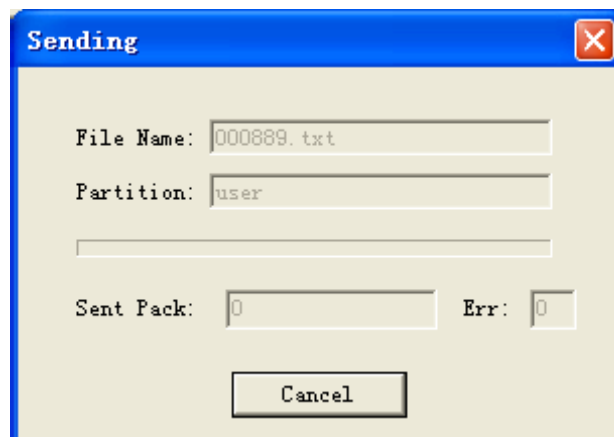


Fig.11-1-3-7

6. After sending the file/files, "Sent" is displayed in the dialog, as is shown in fig. 11-1-3-8.

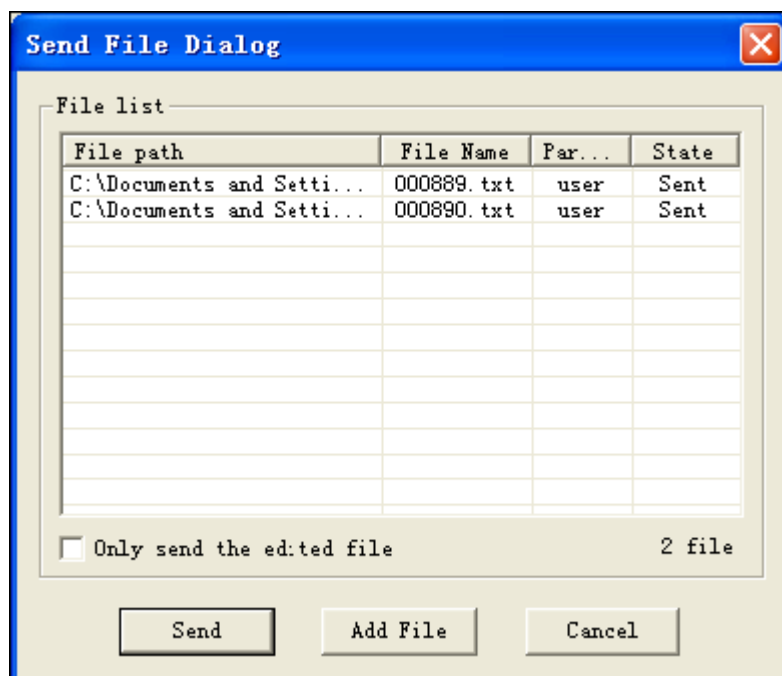


Fig.11-1-3-8

Note 1: Make sure the baudrate is correctly set and the serial line is reliably connected before data transmission,

Note 2: It is forbidden to switch operation modes or pages during data transmission, or critical errors will occur.

Note 3: File LADCHI\*.TXT is ineffective when transferred to the system unless the power is turned off.

### 11.1.4 Serial port DNC on-line machining

#### Operation steps

1. Setting for CNC side:



1) In MDI mode, press  key to enter setting page, and set I/O channel to 0 or 2.

2) Select <DNC> mode; then the system prompts “DNC state ready, press key INPUT after sent by PC.

2. Setting for serial communication software

1) Click menu “Series Port”, set the baudrate to 38400 in Serial Port Setting Dialog.

2) When the system I/O channel is set to 0, select Xon/Xoff in the pull-down menu “DNC Protocol” of Menu “Operation”

When the system I/O channel is set to 1, select XModem in the pull-down menu “DNC Protocol” of Menu “Operation”

3. Open CNC program files. Open the program files by pressing button “Open” in menu “File” or



button in the toolbar, as is shown in fig.11-1-4-1 below (further edit for the program files by serial communication software)

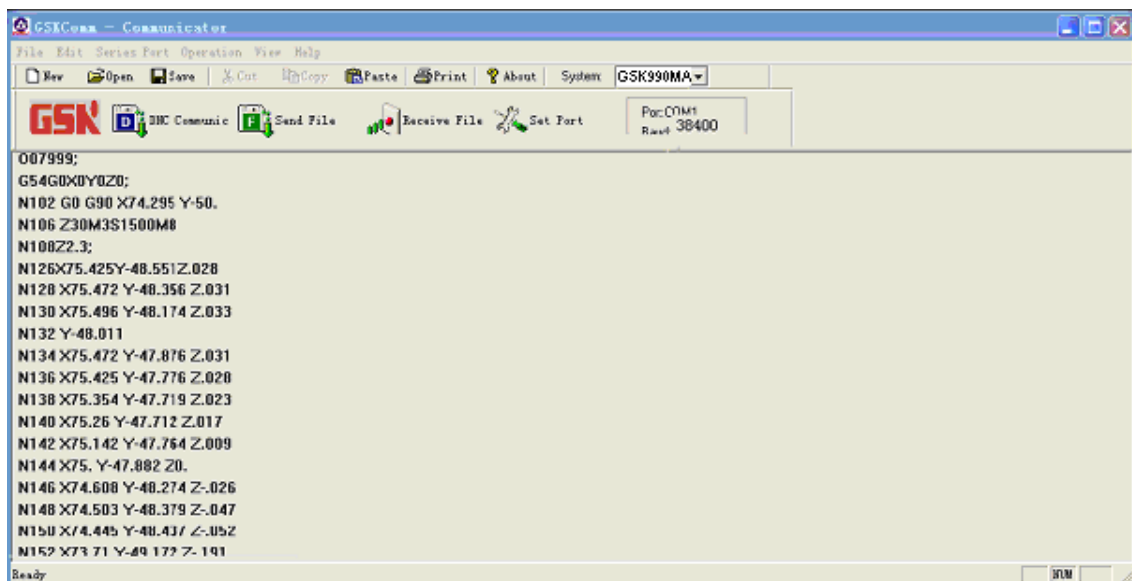
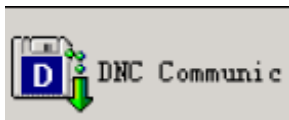


Fig.11-1-4-1



4. DNC transmission. Click **DNC Communic** in the toolbar or pull-down menu “DNC Communic” in menu “Operation” to send the data. When the system I/O channel is set to 0, PC sends the files directly in a common way, then “DNC COMMUNICATION” dialog displays the states of file sending, including the file name, sent bytes, sent lines as well as sent time and speed (byte/s), as is shown in fig. 11-1-4-2. When the system I/O channel is set to 1, PC sends the files by pack, and the dialog displays the states such as sent pack and retransmission times, as is shown in fig. 11-1-4-3:

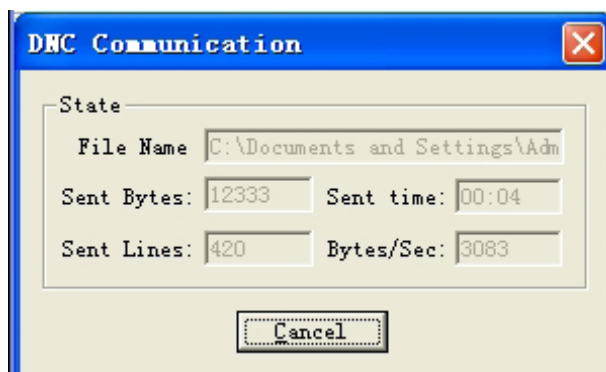


Fig. 11-1-4-2 System I/O channel set to 0

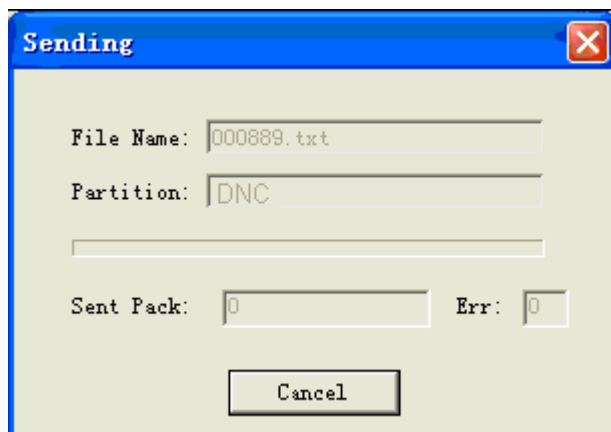
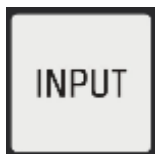


Fig. 11-1-3-4 System I/O channel set to 1



5. Press **INPUT** key on the CNC panel to receive data, press **PROGRAM** key to enter







program page, and then press button  on the panel to start the machining.

- Note:**
1. Do not operate the serial communication software during DNC transmission except for ending the transmission.
  2. M99 is processed as M30 in DNC mode.
  3. Press key to cancel the operation after the machining is completed.

## 11.2 USB communication

### 11.2.1 General and precautions

- 1 The system I/O channel should be set to 1 when USB communication is applied.
- 2 When the USB flash disk operation is finished, pull out the USB flash disk after waiting for a while till the indicator for USB flash disk does not blink, it will avoid the USB flash disk data not fully operated
- 3 The files with suffix txt, nc in the USB flash disk can be displayed.

### 11.2.2 USB part program operation steps

- 1 To copy CNC programs from U flash disk to system disk:
  - a) Press **【USB flash disk】** soft key to switch to USB flash disk file display;
  - b) Press LEFT or RIGHT key to select the CNC program in U disk;
  - c) Press UP or DOWN key to select the CNC program in U disk
  - d) Press **S** soft key, it prompts at the page bottom “Are you sure to copy this file to system disk?” Please key in the new file name ”, key in the new name in the new file box, if the <CANCEL> key is pressed, the copy is cancelled (if the new name is inputted , just cancel the name, not the copy ); press the <INPUT> key start the copy, and the page prompts “Copy ...”. After the copy is finished, it prompts at the page bottom “Copy is finished”.

**Note** note that if there is a file with the same name in CNC storage disk, it will prompt “file exist, please reinput!”

- 2 To delete files from U disk:
  - a) Press **【USB flash disk】** soft key to switch to USB flash disk file display;
  - b) Press UP or DOWN key to select the CNC program in U flash disk;
  - c) Press **【DELETE】** soft key, it prompts at the page bottom “Are you sure to delete the

current file?" , press <CANCEL> key to cancel the deletion; Press <ENTER> key to start the deletion;

d) After the file is deleted, there is no name of this file in U flash disk file display.

3 To copy CNC program from system user disk to U flash disk:

a) Press 【SYSTEM disk】 soft key to switch to system disk file display;

b) Press UP or DOWN key to select the CNC program in system disk;

c) Press 【COPY】 soft key, it prompts at the page bottom "Are you sure to copy this file to USB flash disk?" , rename the new file in the USB flash disk with the original name in the new file box, also it may key in a new name to rename the file; if there is a file with the same name with the copied file in the USB flash disk, it prompts "Are you sure to copy the file to USB flash disk , please input the new name", key in the new name in the new file box, then press <ENTER> key to start the copy, and the page prompts "copy ...". After the copy is finished, it prompts at the page bottom "Copy is finished". Press <CANCEL> key to cancel the copy (if the new file name has been input in the new file name box, it just cancel the name inputted, not the copy).

**Note**      **Note that if there is a file with the same name in U flash disk, it will prompt "file exist, please reinput!".**

4 To delete files from the system user disk:

a) Press 【SYSTEM disk】 soft key to switch to system disk file display;

b) Press UP or DOWN key to select the CNC program in system disk;

c) Press 【DELETE】 soft key, it prompts at the page bottom "Are you sure to delete the current file?" , press <CANCEL> key to cancel the deletion; Press <ENTER> key to start the deletion.

d) After the file is deleted, there is no name of this file in system disk file display.

### 11.2.3 DNC processing operation steps

1 After CNC system starts, set I/O channel value for 1 in <SETTING> page; see details in *OPERATION* Section 3.5.1.

2 Insert the U flash disk.

3 Press 【DNC】 key, it prompts at the page bottom "Please select file in program directory page", press <PROGRAM> key to enter the program page; Press <DIR> soft key to display the USB flash disk programs. Move the cursor to select the processing program, then press <ENTER> key to open this program, and press <CYCLE START> key to execute the DNC processing

#### **11.2.4 U flash disk system exit**

- 1 Pull out U flash disk as the indicator for U flash disk doesn't blink;
- 2 Press **【RETURN】** soft key to return to **【DATA】** soft page in <SETTING> page.

## APPENDIX 1

## GSK990MA PARAMETER LIST

**Explanation:**

The parameters are classified as following patterns according to the data type:

2 data types and data value range

Data type	Effective data range	Remark
Bit	0 or 1	The default value is given by the CNC, and user can modify the setting by requirement.
Number	Specified according to the parameter range	The default value and range are given by the CNC, and user can modify the setting by requirement.

- 1 For bit and axis parameters, the data are comprised by 8 bits with each bit having different meaning.
- 2 The data value range in above table is the common effective range. The specific parameter value range actually differs. See the parameter explanation for details.

**Example**

- (1) Meaning of the bit type parameters

<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>
Data number	BIT7	BIT6	BIT5	BIT4	BIT3	BIT2	BIT1	BIT0	

- (2) Meaning of parameters other than the bit and axis type

<input type="text" value="0"/>	<input type="text" value="2"/>	<input type="text" value="1"/>	<input type="text"/>
Data number	Data		

**Note 1** The blank bits in the parameter explanation and the parameter numbers that are displayed on screen but not in parameter list are reserved for further expansion. They must be set for 0.

- 2 If 0 or 1 of the parameter is not specified with a meaning, it is assumed that: 1 for affirmative, 0 for negative.
- 3 If INI is set for 0, in metric input, the parameter setting unit for linear axis is mm, mm/min; that for rotary axis is deg, deg/min.

If INI is set for 1, in inch input, the parameter setting unit for linear axis is inch, inch/min; that for rotary axis is deg, deg/min.

## 1 Bit parameter

System parameter number

0	0	0			SEQ			INI	ISO	
---	---	---	--	--	-----	--	--	-----	-----	--

**ISO** =1: ISO code

=0: EIA code

**INI** =1: Inch input

=0: Metric input

If INI is set for 0, in metric input, the basic unit for linear axis is mm, mm/min; that for rotary axis is deg, deg/min.

If INI is set for 1, in inch input, the basic unit for linear axis is inch, inch/min; that for rotary axis is deg, deg/min.

**SEQ** =1: Automatic sequence number insertion

=0: Not automatic sequence number insertion

Standard setting: 0 0 0 0 0 0 1 0

System parameter number

0	0	1	SJZ		MIRz	MIRy	MIRx	SPT		
---	---	---	-----	--	------	------	------	-----	--	--

**SPT** =1: Spindle control type: I/O point control

=0: Spindle control type: frequency conversion or others

**MIRx** =1: Mirror setting of X axis: mirror ON

=0: Mirror setting of X axis: mirror OFF

**MIRy** =1: Mirror setting of Y axis: mirror ON

=0: Mirror setting of Y axis: mirror OFF

**MIRz** =1: Mirror setting of Z axis: mirror ON

=0: Mirror setting of Z axis: mirror OFF

**SJZ** =1: Reference point memorizing: yes

=0: Reference point memorizing: no

Standard setting: 0 0 0 0 0 0 0 0

System parameter number

0	0	2	SIOD				DEC4	DECZ	DECY	DECX
---	---	---	------	--	--	--	------	------	------	------

**DECX** =1: X axis deceleration signal high level effective.

=0: X axis deceleration signal low level effective.

- DECY** =1: Y axis deceleration signal high level effective.  
 =0: Y axis deceleration signal low level effective.
- DECZ** =1: Z axis deceleration signal high level effective.  
 =0: Z axis deceleration signal low level effective.
- DEC4** =1: 4TH axis deceleration signal high level effective.  
 =0: 4TH axis deceleration signal low level effective.
- SIOD** =1: Machine zero return deceleration signal operated by PLC logic.  
 =0: Machine zero return deceleration signal not operated by PLC logic.
- Standard setting: 0 0 0 0 0 0 0 0

System parameter number

0	0	3				DIR4	DIRZ	DIRY	DIRX	INM
---	---	---	--	--	--	------	------	------	------	-----

- INM** =1: Min. moving unit of linear axis: Inch  
 =0: Min. moving unit of linear axis: Metric

If INM is set for 0, in metric output, the basic unit for linear axis is mm, mm/min; that for rotary axis is deg, deg/min.

If INM is set for 1, in inch output, the basic unit for linear axis is inch, inch/min; that for rotary axis is deg, deg/min.

- DIRX** =1: X axis feeding direction  
 =0: X axis feeding direction reverse
- DIRY** =1: Y axis feeding direction  
 =0: Y axis feeding direction reverse
- DIRZ** =1: Z axis feeding direction  
 =0: Z axis feeding direction reverse
- DIR4** =1: 4<sup>th</sup> axis feeding direction  
 =0: 4<sup>th</sup> axis feeding direction reverse

Standard setting: 0 0 0 1 1 1 1 0

System parameter number

0	0	4				XIK	AZR	SFD		JAX
---	---	---	--	--	--	-----	-----	-----	--	-----

- JAX** =1: Synch. controlled axes for manual reference point mode: 1 axes (only zero return mode)  
 =0: Synch. controlled axes for manual reference point mode: multiple axes
- SFD** =1: Reference point offset use: yes  
 =0: Reference point offset use: no
- AZR** =1: For G28 when reference point not setup: alarm

=0: For G28 when reference point not setup: use tongue

**XIK** =1: For non-linear positioning axes interlock: all axes stop

=0: For non-linear positioning axes interlock: axes interlock

Standard setting: 0 0 0 1 0 0 0 0

System parameter number

0	0	5	IPR					ISC	
---	---	---	-----	--	--	--	--	-----	--

**ISC** =1: Min. moving unit 0.0001mm,0.0001deg

=0: Min. moving unit 0.001mm,0.001deg

**IPR** =1: Axes min. setting unit is 10 times of min. moving unit: effective

=0: Axes min. setting unit is 10 times of min. moving unit: ineffective

Standard setting: 0 0 0 0 0 0 0 0

System parameter number

0	0	6	MAOB	ZPLS				ZMOD	ZRN
---	---	---	------	------	--	--	--	------	-----

**ZRN** =1: No reference point set up, system alarms if instruction other than G28 is specified during auto running.

=0: No reference point set up, system doesn't alarm if instruction other than G28 is specified during auto running.

**ZMOD** =1: Reference return mode selection: in front of the tongue

=0: Reference return mode selection: behind the tongue

**ZPLS** =1: Zero type selection: one-revolution signal

=0: Zero type selection: non-one-revolution signal

**MAOB** =1: Zero type selection for non-one-revolution signal: B

=0: Zero type selection for non-one-revolution signal: A

Standard setting: 0 1 0 0 0 0 0 0

System parameter number

0	0	7		ZMI4	ZMIZ	ZMIY	ZMIX		A4RT	AXS4
---	---	---	--	------	------	------	------	--	------	------

**AXS4** =1: Set 4<sup>th</sup> axis for linear axis

=0: Set 4<sup>th</sup> axis for rotary axis

**A4RT** =1: Axis rotates with nearest principle

=0: Axis does not rotate with nearest principle

**ZMIx** =1: Direction setting of X axis reference point return: negative

=0: Direction setting of X axis reference point return: positive  
**ZMI<sub>Y</sub>** =1: Direction setting of Y axis reference point return: negative  
 =0: Direction setting of Y axis reference point return: positive  
**ZMI<sub>Z</sub>** =1: Direction setting of Z axis reference point return: negative  
 =0: Direction setting of Z axis reference point return: positive  
**ZMI<sub>4</sub>** =1: Direction setting of 4th axis reference point return: negative  
 =0: Direction setting of 4th axis reference point return: positive  
 Standard setting: 1 0 0 0 0 0 0 0

System parameter number

0	0	8					<b>PLW4</b>	<b>PLWZ</b>	<b>PLWY</b>	<b>PLWX</b>
---	---	---	--	--	--	--	-------------	-------------	-------------	-------------

**PLWX** =1: Pulse width of X-axis is set to 2 microseconds  
 =0: Pulse width of X-axis is set to 1 microsecond  
**PLWY** =1: Pulse width of Y-axis is set to 2 microseconds  
 =0: Pulse width of Y-axis is set to 1 microsecond  
**PLWZ** =1: Pulse width of Z-axis is set to 2 microseconds  
 =0: Pulse width of Z-axis is set to 1 microsecond  
**PLW<sub>4</sub>** =1: Pulse width of the 4<sup>th</sup> axis is set to 2 microseconds  
 =0: Pulse width of the 4<sup>th</sup> axis is set to 1 microsecond  
 Standard setting: 0 0 0 0 0 0 0 0

System parameter number

0	0	9					<b>ZCL</b>		
---	---	---	--	--	--	--	------------	--	--

**ZCL** =1: To cancel relative coordinate system when performing manual reference point return  
 =0: Not cancel relative coordinate system when performing manual reference point return  
 Standard setting: 0 0 0 0 0 0 0 0

System parameter number

0	1	0	<b>RCUR</b>	<b>MSL</b>			<b>RLC</b>		<b>SCBM</b>	
---	---	---	-------------	------------	--	--	------------	--	-------------	--

**SCBM** =1: Check the stroke before moving  
 =0: Not check the stroke before moving  
**RLC** =1: To cancel local coordinate system after resetting  
 =0: Not cancel local coordinate system after resetting  
**MSL** =1: Start from the line where cursor locates on cycle start of multi-section MDI



=0: Start from the first line on cycle start of multi-section MDI

**RCUR** =0: Reset cursor not return to program start point in non-edit mode.

=1: Reset cursor returns to program start point in non-edit mode.

Standard setting: 0 0 0 0 0 0 0 0

System parameter number

0	1	1	BFA	LZR					OUT2
---	---	---	-----	-----	--	--	--	--	------

**OUT2** =1: Outer area entry of 2<sup>nd</sup> stroke limit unallowed

=0: Inner area entry of 2<sup>nd</sup> stroke limit unallowed

**LZR** =1: To perform travel check before manual reference point return after power-on.

=0: Not perform travel check before manual reference point return after power-on.

**BFA** =1: To make alarm after overtravel when overtravel instruction is given.

=0: To make alarm before overtravel when overtravel instruction is given.

Standard setting: 0 0 0 0 0 0 0 1

System parameter number

0	1	2	FDR	RDR	TDR	RFO		LRP	RPD
---	---	---	-----	-----	-----	-----	--	-----	-----

**RPD** =1: Manual rapid effective before reference point return after power-on.

=0: Manual rapid ineffective before reference point return after power-on.

**LRP** =1: The positioning (G00) interpolation type is linear.

=0: The positioning (G00) interpolation type is non-linear.

**RFO** =1: Rapid feed stop when override is F0.

=0: Rapid feed not stop when override is F0.

**TDR** =1: Dry run effective during tapping.

=0: Dry run ineffective during tapping.

**RDR** =1: Dry run effective during cutting feeding.

=0: Dry run ineffective during cutting feeding.

**FDR** =1: Dry run effective during rapid positioning.

=0: Dry run ineffective during rapid positioning.

Standard setting: 0 0 0 0 0 0 0 0

System parameter number

0	1	3						HPC	NPC
---	---	---	--	--	--	--	--	-----	-----

**NPC** =1: Feed per revolution effective with no position encoder

=0: Feed per revolution ineffective with no position encoder

**HPC** =1: Position encoder installed.

=0: Position encoder not installed.

Standard setting: 0 0 0 0 0 0 1 0

System parameter number

0	1	4								<b>DLF</b>	<b>HFC</b>
---	---	---	--	--	--	--	--	--	--	------------	------------

**HFC** =1: Clamp combined by straight line and arc for helical interpolation feedrate

=0: Clamp by straight line and arc separately for helical interpolation feedrate

**DLF** =1: Reference point return by manual feed after reference point is setup and memorized

=0: Reference point return by rapid traverse after reference point is setup and memorized

Standard setting: 0 0 0 0 0 0 0 0

System parameter number

0	1	5			<b>PIIS</b>		<b>PPCK</b>	<b>ASL</b>	<b>PLAC</b>	<b>STL</b>
---	---	---	--	--	-------------	--	-------------	------------	-------------	------------

**STL** =1: To select prereading working type

=0: To select non-prereading working type

**PLAC** =1: Acceleration/deceleration type after forecasting interpolation: exponential

=0: Acceleration /deceleration type after forecasting interpolation: linear

**ASL** =1: Auto corner deceleration function of forecasting: speed difference control

=0: Auto corner deceleration function of forecasting: angular control

**PPCK** =1: To perform in-position check by forecasting.

=0: Not perform in-position check by forecasting.

**PIIS** =1: Overlapping interpolation effective in acceleration/deceleration blocks before forecasting.

=0: Overlapping interpolation ineffective in acceleration /deceleration blocks before forecasting.

Standard setting: 0 0 0 0 0 0 0 1

System parameter number

0	1	6	ALS					FLLS	FBLS	FBOL
---	---	---	-----	--	--	--	--	------	------	------

**FBOL** =1: Rapid traverse type: post acceleration /deceleration

=0: Rapid traverse type: pre-acceleration /deceleration

**FBLS** =1: Pre-acceleration /deceleration type of rapid traverse: S

=0: Pre-acceleration /deceleration type of rapid traverse: linear

**FLLS** =1: Post acceleration /deceleration type of rapid traverse: exponential

=0: Post acceleration /deceleration type of rapid traverse: linear

**ALS** =1: Auto corner feed effective.

=0: Auto corner feed ineffective.

Standard setting: 0 0 0 0 0 0 1 0

System parameter number

0	1	7	CPCT	CALT	WLOE		HLOE	CLLE	CBLS	CBOL
---	---	---	------	------	------	--	------	------	------	------

**CBOL** =1: Cutting feed type: post acceleration /deceleration

=0: Cutting feed type: pre-acceleration /deceleration

**CBLS** =1: Pre-acceleration /deceleration type of cutting feed: S

=0: Pre-acceleration /deceleration type of cutting feed: linear

**CLLE** =1: Post acceleration /deceleration type of cutting feed: exponential

=0: Post acceleration /deceleration type of cutting feed: linear

**HLOE** =1: JOG running type: exponential

=0: JOG running type: linear

**WLOE** =1: MPG running type: exponential

=0: MPG running type: linear

**CALT** =1: Cutting feed acceleration clamping.

=0: Cutting feed acceleration not clamping.

**CPCT** =1: To control the in-position precision in cutting feed.

=0: Not control the in-position precision in cutting feed.

Standard setting: 1 0 1 0 0 1 0 1

System parameter number

0	1	8	RVCS			RBK	FFR			RVIT
---	---	---	------	--	--	-----	-----	--	--	------

**RVIT** =1 To execute next block after compensation as backlash is over value allowable

=0 To execute next block during compensation as backlash is over value allowable

**FFR** =1: Cutting and rapid traverse both effective in feedforward control.

=0: Cutting feed effective in feedforward control.

**RBK** =1: To perform backlash compensation for cutting feed and rapid traverse separately

=0: To perform backlash compensation for cutting feed and rapid traverse together

**RVCS** =1: Backlash compensation type: ascending or decending

=0: Backlash compensation type: fixed frequency

Standard setting: 0 0 0 0 0 0 0 0

System parameter number

0	1	9	IOV		ALMS	ALMS5	ALMS4	ALMSZ	ALMSY	ALMSX
---	---	---	-----	--	------	-------	-------	-------	-------	-------

**ALMX** =1: Driver alarm high level effective

=0: Driver alarm low level effective

**ALMY** =1: Driver alarm high level effective

=0: Driver alarm low level effective

**ALMZ** =1: Driver alarm high level effective

=0: Driver alarm low level effective

**ALM4** =1: Driver alarm high level effective

=0: Driver alarm low level effective

**ALM5** =1: Driver alarm high level effective

=0: Driver alarm low level effective

**IOV** =1: Override signal high level effective

=0: Override signal low level effective

Standard setting: 0 0 0 0 0 0 0 0

System parameter number

0	2	0								ITL
---	---	---	--	--	--	--	--	--	--	-----

**ITL** =1: All axes interlock signal effective

=0: All axes interlock signal ineffective

Standard setting: 0 0 0 0 0 0 0 0

System parameter number

0	2	1	DISL	ENG	CHI					
---	---	---	------	-----	-----	--	--	--	--	--

**CHI** =1: To set the actual language not for Chinese

=0: To set the actual language for Chinese

**ENG** =1: To set the actual language for English

=0: To set the actual language not for English

**DISL** =1: To display company LOGO at start

=0: Not to display company LOGO at start

Standard setting: 0 0 0 0 0 0 0 0

System parameter number

0	2	2		DAL						
---	---	---	--	-----	--	--	--	--	--	--

**DAL** =1: Add tool length compensation in absolute position display.

=0: Not add tool length compensation in absolute position display.

Standard setting: 0 0 0 0 0 0 0 0

System parameter number

0	2	3		POSM						
---	---	---	--	------	--	--	--	--	--	--

**POSM** =1: Mode displayed on program monitoring page.

=0: Mode not displayed on program monitoring page.

Standard setting: 0 1 0 0 0 0 0 0

System parameter number

0	2	4		NPA						
---	---	---	--	-----	--	--	--	--	--	--

**NPA** =1: To switch to alarm page when alarm occurs.

=0: Not switch to alarm page when alarm occurs.

Standard setting: 0 0 0 0 0 0 0 0

System parameter number

0	2	6								PETP
---	---	---	--	--	--	--	--	--	--	------

**PETP** =1: To switch to program page by pressing operator panel Edit key.

=0: Not switch to program page by pressing operator panel Edit key.

Standard setting: 0 0 0 0 0 0 0 1

System parameter number

0	2	7				NE9				NE8
---	---	---	--	--	--	-----	--	--	--	-----

**NE8** =1: Editing of subprogram with the number 80000 – 89999 unallowed

=0: Editing of subprogram with the number 80000 – 89999 allowed

**NE9** =1: Editing of Subprogram with the number 90000 – 99999 unallowed

=0: Editing of Subprogram with the number 90000 – 99999 allowed

Standard setting: 0 0 0 1 0 0 0 1

System parameter number

0	2	8	MCL			MKP				
---	---	---	-----	--	--	-----	--	--	--	--

**MKP** =1: To clear the program edited when M02, M30 or % is executed in MDI mode.

=0: Not clear the program edited when M02, M30 or % is executed in MDI mode.

**MCL** =1: To delete the program edited when pressing RESET key in MDI mode.

=0: Not delete the program edited when pressing RESET key in MDI mode.

Standard setting: 0 0 0 0 0 0 0 0

System parameter number

0	2	9		MCM		IWZ	WZO	MCV	GOF	WOF
---	---	---	--	-----	--	-----	-----	-----	-----	-----

**WOF** =1: Tool wear offset input by MDI disabled.

=0: Tool wear offset input by MDI enabled.

**GOF** =1: Geometric tool offset input by MDI disabled.

=0: Geometric tool offset input by MDI enabled.

**MCV** =1: Macro variables input by MDI disabled.

=0: Macro variables input by MDI enabled.

**WZO** =1: Workpiece origin offset input by MDI disabled.

=0: Workpiece origin offset input by MDI enabled.

**IWZ** =1: Workpiece origin offset input by MDI during dwell disabled.

=0: Workpiece origin offset input by MDI during dwell enabled.

**MCM** =1: Custom macro input by MDI: MDI type

=0: Custom macro input by MDI: any type

Standard setting: 0 0 0 0 0 0 0 0

System parameter number

0	3	0			ABS	MAB				DPI
---	---	---	--	--	-----	-----	--	--	--	-----

**DPI** =1: Decimal point omitted in programming, default: mm,sec

=0: Decimal point omitted in programming, default: minimum unit

**MAB** =1: Absolute or relative setting by parameters in MDI mode.

=0: Absolute or relative setting by G90/G91 in MDI mode.

**ABS** =1: Instructions regarded as absolute in MDI mode.

=0: Instructions regarded as incremental in MDI mode.

Standard setting: 0 0 0 0 0 0 0 0

System parameter number

0	3	1		CLR	G13	G91	G19	G18	G17	G01
---	---	---	--	-----	-----	-----	-----	-----	-----	-----

**G01** =1: G01 at power-on or clearing.

=0: G00 at power-on or clearing.

**G17** =1: G17 plane at power-on or clearing.

=0: Not G17 plane at power-on or clearing.

**G18** =1: G18 plane at power-on or clearing.

=0: Not G18 plane at power-on or clearing.

**G19** =1: G19 plane at power-on or clearing.

=0: Not G19 plane at power-on or clearing.

**G91** =1: To set for G91 mode at power-on or clearing.

=0: To set for G90 mode at power-on or clearing.

**G13** =1: To set for G13 mode at power-on or clearing.

=0: To set for G12 mode at power-on or clearing.

**CLR** =1: MDI reset key, to clear external reset signal, make emergency stop

=0: MDI reset key, to reset external signal, make emergency stop

Standard setting: 0 0 0 0 0 0 1 0

System parameter number

0	3	2		AD2						
---	---	---	--	-----	--	--	--	--	--	--

**AD2** =1: Make alarm if two or more same addresses are specified in a block.

=0: Do not make alarm if two or more same addresses are specified in a block.

Standard setting: 0 1 0 0 0 0 0 0

System parameter number

<b>0</b>	<b>3</b>	<b>3</b>	<b>M3B</b>			<b>M30</b>		<b>M02</b>		
----------	----------	----------	------------	--	--	------------	--	------------	--	--

- M02** =1: To return to block beginning when M02 is being executed.  
 =0: Not return to block beginning when M02 is being executed.
- M30** =1: To return to block beginning when M30 is to be executed.  
 =0: Not return to block beginning when M30 is to be executed.
- M3B** =1: At most three M codes allowable in a section of program.  
 =0: Only one M code allowable in a section of program.

Standard setting: 1 0 0 1 0 0 0 0

System parameter number

<b>0</b>	<b>3</b>	<b>4</b>	<b>CFH</b>							<b>DWL</b>
----------	----------	----------	------------	--	--	--	--	--	--	------------

- DWL** =1: G04 for dwell per revolution in per revolution feed mode.  
 =0: G04 not for dwell per revolution in per revolution feed mode.
- CFH** =1: To clear F,H,D codes at reset or emergency stop.  
 =0: To reserve F,H,D codes at reset or emergency stop.

Standard setting: 0 0 0 0 0 0 0 0

System parameter number

<b>0</b>	<b>3</b>	<b>5</b>	<b>C07</b>	<b>C06</b>	<b>C05</b>	<b>C04</b>	<b>C03</b>	<b>C02</b>	<b>C01</b>	
----------	----------	----------	------------	------------	------------	------------	------------	------------	------------	--

- C01** =1: To clear G codes of 01 group at reset or emergency stop.  
 =0: To reserve G codes of 01 group at reset or emergency stop.
- C02** =1: To clear G codes of 02 group at reset or emergency stop.  
 =0: To reserve G codes of 02 group at reset or emergency stop.
- C03** =1: To clear G codes of 03 group at reset or emergency stop.  
 =0: To reserve G codes of 03 group at reset or emergency stop.
- C04** =1: To clear G codes of 04 group at reset or emergency stop.  
 =0: To reserve G codes of 04 group at reset or emergency stop.
- C05** =1: To clear G codes of 05 group at reset or emergency stop.  
 =0: To reserve G codes of 05 group at reset or emergency stop.
- C06** =1: To clear G codes of 06 group at reset or emergency stop.  
 =0: To reserve G codes of 06 group at reset or emergency stop.
- C07** =1: To clear G codes of 07 group at reset or emergency stop.



=0: To reserve G codes of 07 group at reset or emergency stop.

Standard setting: 0 0 0 0 0 0 0 0

System parameter number

0	3	6		C15	C14	C13	C12	C11	C10	C09	C08
---	---	---	--	-----	-----	-----	-----	-----	-----	-----	-----

**C08** =1: To clear G codes of 08 group at reset or emergency stop.

=0: To reserve G codes of 08 group at reset or emergency stop.

**C09** =1: To clear G codes of 09 group at reset or emergency stop.

=0: To reserve G codes of 09 group at reset or emergency stop.

**C10** =1: To clear G codes of 10 group at reset or emergency stop.

=0: To reserve G codes of 10 group at reset or emergency stop.

**C11** =1: To clear G codes of 11 group at reset or emergency stop.

=0: To reserve G codes of 11 group at reset or emergency stop.

**C12** =1: To clear G codes of 12 group at reset or emergency stop.

=0: To reserve G codes of 12 group at reset or emergency stop.

**C13** =1: To clear G codes of 13 group at reset or emergency stop.

=0: To reserve G codes of 13 group at reset or emergency stop.

**C14** =1: To clear G codes of 14 group at reset or emergency stop.

=0: To reserve G codes of 14 group at reset or emergency stop.

**C15** =1: To clear G codes of 15 group at reset or emergency stop.

=0: To reserve G codes of 15 group at reset or emergency stop.

Standard setting: 0 0 0 0 0 0 0 0

System parameter number

0	3	7					SOC	RSC		SCRW
---	---	---	--	--	--	--	-----	-----	--	------

**SCRW** =1: To perform pitch compensation.

=0: Not perform pitch compensation.

**RSC** =1: To calculate G96 spindle speed according to current coordinate during G0 rapid positioning

=0: To calculate G96 spindle speed according to end point coordinate during G0 rapid positioning

**SOC** =1: G96 spindle speed clamped behind spindle override

=0: G96 spindle speed clamped before spindle override

Standard setting: 0 0 0 0 0 0 0 0

System parameter number

0	3	8	PG2	PG1					SAR
---	---	---	-----	-----	--	--	--	--	-----

**SAR** =1: To detect the spindle speed in-position signal

=0: Not detect the spindle speed in-position signal

**PG2,PG1**: Gear ratio of spindle and position encoder

00 for 1:1; 01 for 2:1; 10 for 4:1; 11 for 8:1

Standard setting: 0 0 0 0 0 0 0 0

System parameter number

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

**TLC** =1: Tool length compensation type: B

=0: Tool length compensation type: A

Standard setting: 0 0 0 0 0 0 0 1

System parameter number

0	4	0	ODI					CCN	SUP
---	---	---	-----	--	--	--	--	-----	-----

**SUP** =1: Start-up type in tool radius compensation: B

=0: Start-up type in tool radius compensation: A

**CCN** =1: To move to the intermediate point by G28 and cancel compensation in tool radius compensation.

=0: To move to the intermediate point by G28 and reserve compensation in tool radius compensation.

**ODI** =1: Tool radius compensation value set by diameter

=0: Tool radius compensation value set by radius

Standard setting: 1 0 0 0 0 1 0 1

System parameter number

0	4	1		CN1	G39			CIM	OIM
---	---	---	--	-----	-----	--	--	-----	-----

**OIM** =1: Metric and inch conversion, automatic tool offset change enabled.

=0: Metric and inch conversion, automatic tool offset change disabled.

**CIM** =1: Metric and inch conversion,for workpiece coordinate system automatic change.

=0: Metric and inch conversion, workpiece coordinate system not automatic change.

**G39** =1: Corner rounding effective in radius compensation.

=0: Corner rounding ineffective in radius compensation.

**CN1** =1: Interference check enabled in radius compensation.

=0: Interference check disabled in radius compensation.

Standard setting: 0 1 1 0 0 0 0 0

System parameter number

0	4	2		M5B	M5T	RD2	RD1			EXC	
---	---	---	--	-----	-----	-----	-----	--	--	-----	--

**EXC** =1: To specify external action by G81.

=0: To specify drilling canned cycle by G81.

**RD1** =1: To set the retraction direction of G76, G87: positive

=0: To set the retraction direction of G76, G87: negative

**RD2**=1: To set the retraction axis of G76, G87: X

=0: To set the retraction axis of G76, G87: Y

**M5T** =1: To output M05 at the spindle CW and CCW shift in tapping cycle.

=0: Not to output M05 at the spindle CW and CCW shift in tapping cycle.

**M5B** =1: To output M05 at the spindle CW and CCW shift in drilling cycle.

=0: Not to output M05 at the spindle CW and CCW shift in drilling cycle.

Standard setting: 0 0 0 0 0 0 0 0

System parameter number

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

**OZA** =1: To make alarm if cut-in depth is not specified in peck drilling cycle (G73,G83).

=0: Not to make alarm if cut-in depth is not specified in peck drilling cycle (G73, G83).

Standard setting: 0 0 0 0 0 0 1 0

System parameter number

0	4	4		FHD	PCP	DOV			VGR	G84
---	---	---	--	-----	-----	-----	--	--	-----	-----

**G84** =1: Use M codes in rigid tapping

=0: Not use M codes in rigid tapping

**VGR** =1: Arbitrary gear ratio of the spindle and position encoder enabled in rigid tapping.

=0: Arbitrary gear ratio of the spindle and position encoder disabled in rigid tapping.

**DOV** =1: Override effective during rigid tapping retraction.

=0: Override ineffective during rigid tapping retraction.

**PCP** =1: To change rigid tapping for high-speed peck drilling cycle.

=0: Not change rigid tapping for high-speed peck drilling cycle.

**FHD** =1: Single block effective for feed dwell during rigid tapping.

=0: Single block ineffective for feed dwell during rigid tapping.

Standard setting: 0 0 0 0 0 0 0 0

System parameter number

0	4	5				OV3	OVU	TDR		NIZ
---	---	---	--	--	--	-----	-----	-----	--	-----

**NIZ** =1: To perform the rigid tapping finishing.

=0: Not perform the rigid tapping finishing.

**TDR** =1: To use the same time constant during the rigid tapping advance and retraction.

=0: Not use the same time constant during the rigid tapping advance and retraction.

**OVU** =1: 10% retraction override for rigid tapping.

=0: 1% retraction override for rigid tapping.

**OV3** =1: Spindle speed effective by program instruction.

=0: Spindle speed ineffective by program instruction.

Standard setting: 0 0 0 0 0 0 0 0

System parameter number

0	4	6			ORI				SSOG	DGN
---	---	---	--	--	-----	--	--	--	------	-----

**DGN** =1: Difference of the spindle and the tapping axis errors

=0: Synch error in rigid tapping.

**SSOG** =1: For servo spindle control at the beginning of rigid tapping.

=0: For following spindle control at the beginning of rigid tapping.

**ORI** =1: To perform spindle dwell when rigid tapping starts.

=0: Not perform spindle dwell when rigid tapping starts.

Standard setting: 0 0 0 0 0 0 0 0

System parameter number

0	4	7		XSC	SCLz	SCLy	SCLx			R1N
---	---	---	--	-----	------	------	------	--	--	-----

**R1N** =1: Rotational angle of coordinate rotation: by G90/G91 instruction

=0: Rotational angle of coordinate rotation: by absolute instruction

**SCLx** =1: X axis scaling effective.

=0: X axis scaling ineffective.

**SCLy** =1: Y axis scaling effective.

=0: Y axis scaling ineffective.

**SCLz** =1: Z axis scaling effective.

=0: Z axis scaling ineffective.

**XSC** =1: Axes scaling override by I, J, K

=0: Axes scaling override by P instruction

Standard setting: 0 1 1 1 1 0 0 1

System parameter number

0	4	8								MDL
---	---	---	--	--	--	--	--	--	--	-----

**MDL** =1: G codes of unidirectional positioning set for modal

=0: G codes of unidirectional positioning not set for modal.

Standard setting: 0 0 0 0 0 0 0 0

System parameter number

0	4	9								RPST
---	---	---	--	--	--	--	--	--	--	------

**RPST** =1: To move Z axis by G01 during program restarting.

=0: To move Z axis by G00 during program restarting.

Standard setting: 0 0 0 0 0 0 0 0

System parameter number

0	5	0		SIM		G90			REL	
---	---	---	--	-----	--	-----	--	--	-----	--

**REL** =1: Relative position display setting of indexing table: within 360°.

=0: Relative position display setting of indexing table: beyond 360°.

**G90** =1: Indexing instruction: absolute instruction.

=0: Indexing instruction: specified by G90/G91.

**SIM** =1: Make alarm if indexing instruction and other axes instructions are in same block.

=0: Do not make alarm if indexing instruction and other axes instructions are in same block.

Standard setting: 0 1 0 0 0 0 0 0

System parameter number

0	5	1	MDLY		SBM					
---	---	---	------	--	-----	--	--	--	--	--

**SBM** =1: Single block allowed in macro statement.  
 =0: Single block unallowed in macro statement.

**MDLY** =1: Delay in macro statement.  
 =0: Not delay in macro statement.

Standard setting: 0 0 0 0 0 0 0 0

System parameter number

0	5	2	CLV	CCV						
---	---	---	-----	-----	--	--	--	--	--	--

**CCV** =1: Macro common variables #100 - #199 clearing after reset.  
 =0: Macro common variables #100 - #199 not clearing after reset.

**CLV** =1: Macro local variables #1 - #50 clearing after reset.  
 =0: Macro local variables #1 - #50 not clearing after reset.

Standard setting: 0 0 0 0 0 0 0 0

System parameter number

0	5	3								
---	---	---	--	--	--	--	--	--	--	--

Standard setting: 0 0 0 0 0 0 0 0

System parameter number

0	5	5								CANT
---	---	---	--	--	--	--	--	--	--	------

**CANT** =1: Automatic clearing for single piece.  
 =0: Not automatic clearing for single piece.

Standard setting: 0 0 0 0 0 0 0 0

System parameter number

0	5	6	HNGD				HISR			HPF
---	---	---	------	--	--	--	------	--	--	-----

**HPF** =1: To select full running for MPG moving.  
 =0: Not select full running for MPG moving.

**HISR** =1: Use MPG/Step interruption function.  
 =0: Not use MPG/Step interruption function.

**HNGD** =1: Axes moving direction are identical with MPG rotation direction.

=0: Axes moving direction are not identical with MPG rotation direction.

Standard setting: 0 0 0 0 0 0 0 1

System parameter number

0	5	8							
---	---	---	--	--	--	--	--	--	--

Standard setting: 0 0 0 0 0 0 0 0

System parameter number

0	5	9	OTOP	LEDT	LOPT		AOV	DEC	RHPG	OHPG
---	---	---	------	------	------	--	-----	-----	------	------

**OHPG** =1: Feed by external handwheel.

=0: Feed not by external handwheel.

**RHPG** =1: Use electronic handwheel drive function。

=0: Not use electronic handwheel drive function。

**DEC** =1: Use external deceleration.

=0: Not use external deceleration.

**AOV** =1: Use automatic corner override.

=0: Not use automatic corner override.

**LOPT** =1: Use external operator panel lock.

=0: Not use external operator panel lock.

**LEDT** =1: Use external editing lock.

=0: Not use external editing lock.

**OTOP** =1: Use external start and stop.

=0: Not use external start and stop.

Standard setting: 0 0 0 0 0 0 0 0

System parameter number

0	6	0			SCL	SPK	IXC			TLF
---	---	---	--	--	-----	-----	-----	--	--	-----

**TLF** =1: Use tool life management.

=0: Not use tool life management.

**IXC** =1: Use indexing table.

=0: Not use indexing table.

**SPK** =1: Use small peck drilling cycle.

=0: Not use small peck drilling cycle.

**SCL** =1: Use scaling.

=0: Not use scaling.

Standard setting: 0 0 1 0 0 0 0 0

System parameter number

0	6	1	<b>FALM</b>	<b>LALM</b>	<b>EALM</b>	<b>SALM</b>	<b>AALM</b>		<b>SYC</b>	<b>SSC</b>
---	---	---	-------------	-------------	-------------	-------------	-------------	--	------------	------------

**SSC** =1: Use constant surface speed control.

=0: Not use constant surface speed control.

**SYC** =1: Use synch spindle.

=0: Not use synch spindle.

**SALM** =1: Spindle driver alarm ignored.

=0: Spindle driver alarm not ignored.

**AALM** =1: External user alarm ignored.

=0: External user alarm not ignored.

**EALM** =1: Emergency stop alarm ignored.

=0: Emergency stop alarm not ignored.

**LALM** =1: Limit alarm ignored.

=0: Limit alarm not ignored.

**FALM** =1: Feed axis driver alarm ignored.

=0: Feed axis driver alarm not ignored.

Standard setting: 0 0 0 0 0 0 0 0



## 2 Number parameter

Parameter number	Definition	Default value
00000	I/O channel, input and output device selection.	0

Setting range: 0~3

It is set to 0 for the communication between CNC and PC via RS232 interface, and set to either 1, 2 or 3 when CNC connecting with U flash disk.

0001	Baudrate of serial port RS232(when I/O channel is set for 0)	38400
------	--	-------

Setting range: 0~115200 (unit: BPS)

0002	File transmission baudrate of serial port U flash disk (when I/O channel are set for 1,2 or 3)	115200
------	--	--------

Setting range: 0~115200 (unit: BPS)

0003	Waiting time of screen protection (minute)	0
------	--	---

Setting range: 0~999

0004	System interpolation period (1, 2, 4, 8ms)	1
------	--	---

Setting range: 1~8

0005	Axes controlled by CNC	3
------	------------------------	---

Setting range: 3~5

0006	Program axis name of rotary axis	0
------	----------------------------------	---

When the CNC controlled axes is set for 4, the program axes names of rotary axes are set for 0, 1, 2, the rotary axis name is displayed for A, B, C respectively.

0007	Axis name setting in primary coordinate system	0
------	--	---

0008	Servo axis number of each axis	0
------	--------------------------------	---

0010	External workpiece origin offset amount along X axis	0.0000
------	--	--------

Setting range: -9999.9999~9999.9999 (mm)

0011	External workpiece origin offset amount along Y axis	0.0000
------	--	--------

Setting range: -9999.9999~9999.9999 (mm)

## Appendix

---

0012	External workpiece origin offset amount along Z axis	0.0000
------	--	--------

Setting range: -9999.9999 ~ 9999.9999 (mm)

0013	External workpiece origin offset amount along 4th axis	0.0000
------	--	--------

Setting range: -9999.9999 ~ 9999.9999 (mm)

0015	Origin offset amount of workpiece coordinate system 1 (G54_X)	0.0000
------	---	--------

Setting range: -9999.9999 ~ 9999.9999 (mm)

0016	Origin offset amount of workpiece coordinate system 1 (G54_Y)	0.0000
------	---	--------

Setting range: -9999.9999 ~ 9999.9999 (mm)

0017	Origin offset amount of workpiece coordinate system 1 (G54_Z)	0.0000
------	---	--------

Setting range: -9999.9999 ~ 9999.9999 (mm)

0018	Origin offset amount of workpiece coordinate system 1 (G54_4TH)	0.0000
------	---	--------

Setting range: -9999.9999 ~ 9999.9999 (mm)

0020	Origin offset amount of workpiece coordinate system 2 (G55_X)	0.0000
------	---	--------

Setting range: -9999.9999 ~ 9999.9999 (mm)

0021	Origin offset amount of workpiece coordinate system 2 (G55_Y)	0.0000
------	---	--------

Setting range: -9999.9999 ~ 9999.9999 (mm)

0022	Origin offset amount of workpiece coordinate system 2 (G55_Z)	0.0000
------	---	--------

Setting range: -9999.9999 ~ 9999.9999 (mm)

0023	Origin offset amount of workpiece coordinate system 2 (G55_4TH)	0.0000
------	---	--------

Setting range: -9999.9999 ~ 9999.9999 (mm)

0025	Origin offset amount of workpiece coordinate system 3 (G56_X)	0.0000
------	--	--------

Setting range: -9999.9999~9999.9999 (mm)

0026	Origin offset amount of workpiece coordinate system 3 (G56_Y)	0.0000
------	--	--------

Setting range: -9999.9999~9999.9999 (mm)

0027	Origin offset amount of workpiece coordinate system 3 (G56_Z)	0.0000
------	--	--------

Setting range: -9999.9999~9999.9999 (mm)

0028	Origin offset amount of workpiece coordinate system 3 (G56_4TH)	0.0000
------	--	--------

Setting range: -9999.9999~9999.9999 (mm)

0030	Origin offset amount of workpiece coordinate system 4 (G57_X)	0.0000
------	--	--------

Setting range: -9999.9999~9999.9999 (mm)

0031	Origin offset amount of workpiece coordinate system 4 (G57_Y)	0.0000
------	--	--------

Setting range: -9999.9999~9999.9999 (mm)

0032	Origin offset amount of workpiece coordinate system 4 (G57_Z)	0.0000
------	--	--------

Setting range: -9999.9999~9999.9999 (mm)

0033	Origin offset amount of workpiece coordinate system 4 (G57_4TH)	0.0000
------	--	--------

Setting range: -9999.9999~9999.9999 (mm)

0035	Origin offset amount of workpiece coordinate system 5 (G58_X)	0.0000
------	--	--------

Setting range: -9999.9999~9999.9999 (mm)

0036	Origin offset amount of workpiece coordinate system 5 (G58_Y)	0.0000
------	--	--------

Setting range: -9999.9999~9999.9999 (mm)

## Appendix

---

0037	Origin offset amount of workpiece coordinate system 5 (G58_Z)	0.0000
------	--	--------

Setting range: -9999.9999 ~ 9999.9999 (mm)

0038	Origin offset amount of workpiece coordinate system 5 (G58_4TH)	0.0000
------	--	--------

Setting range: -9999.9999 ~ 9999.9999 (mm)

0040	Origin offset amount of workpiece coordinate system 6 (G59_X)	0.0000
------	--	--------

Setting range: -9999.9999 ~ 9999.9999 (mm)

0041	Origin offset amount of workpiece coordinate system 6 (G59_Y)	0.0000
------	--	--------

Setting range: -9999.9999 ~ 9999.9999 (mm)

0042	Origin offset amount of workpiece coordinate system 6 (G59_Z)	0.0000
------	--	--------

Setting range: -9999.9999 ~ 9999.9999 (mm)

0043	Origin offset amount of workpiece coordinate system 6 (G59_4TH)	0.0000
------	--	--------

Setting range: -9999.9999 ~ 9999.9999 (mm)

0045	X coordinate of the 1st reference point in machine coordinate system	0.0000
------	---	--------

Setting range: -9999.9999 ~ 9999.9999 (mm)

0046	Y coordinate of the 1st reference point in machine coordinate system	0.0000
------	---	--------

Setting range: -9999.9999 ~ 9999.9999 (mm)

0047	Z coordinate of the 1st reference point in machine coordinate system	0.0000
------	---	--------

Setting range: -9999.9999 ~ 9999.9999 (mm)

0048	4TH coordinate of the 1st reference point in machine coordinate system	0.0000
------	---	--------

Setting range: -9999.9999 ~ 9999.9999 (mm)

0050	X coordinate of the 2nd reference point in machine coordinate system	0.0000
------	--	--------

Setting range: -9999.9999~9999.9999 (mm)

0051	Y coordinate of the 2nd reference point in machine coordinate system	0.0000
------	--	--------

Setting range: -9999.9999~9999.9999 (mm)

0052	Z coordinate of the 2nd reference point in machine coordinate system	0.0000
------	--	--------

Setting range: -9999.9999~9999.9999 (mm)

0053	4TH coordinate of the 2nd reference point in machine coordinate system	0.0000
------	--	--------

Setting range: -9999.9999~9999.9999 (mm)

0055	X coordinate of the 3rd reference point in machine coordinate system	0.0000
------	--	--------

Setting range: -9999.9999~9999.9999 (mm)

0056	Y coordinate of the 3rd reference point in machine coordinate system	0.0000
------	--	--------

Setting range: -9999.9999~9999.9999 (mm)

0057	Z coordinate of the 3rd reference point in machine coordinate system	0.0000
------	--	--------

Setting range: -9999.9999~9999.9999 (mm)

0058	4TH coordinate of the 3rd reference point in machine coordinate system	0.0000
------	--	--------

Setting range: -9999.9999~9999.9999 (mm)

0060	X coordinate of the 4th reference point in machine coordinate system	0.0000
------	--	--------

Setting range: -9999.9999~9999.9999 (mm)

0061	Y coordinate of the 4th reference point in machine coordinate system	0.0000
------	--	--------

Setting range: -9999.9999~9999.9999 (mm)

## Appendix

---

0062	Z coordinate of the 4th reference point in machine coordinate system	0.0000
------	--	--------

Setting range: -9999.9999 ~ 9999.9999 (mm)

0063	4TH coordinate of the 4th reference point in machine coordinate system	0.0000
------	--	--------

Setting range: -9999.9999 ~ 9999.9999 (mm)

0065	Moving amount per revolution of rotary axis	0.0000
------	---	--------

Setting range: 0 ~ 999.9999 (deg)

0066	Negative X axis stroke coordinate of storage travel detection 1	-9999
------	---	-------

Setting range: -9999.9999 ~ 9999.9999 (mm)

0067	Positive X axis stroke coordinate of storage travel detection 1	9999
------	---	------

Setting range: -9999.9999 ~ 9999.9999 (mm)

0068	Negative Y axis stroke coordinate of storage travel detection 1	-9999
------	---	-------

Setting range: -9999.9999 ~ 9999.9999 (mm)

0069	Positive Y axis stroke coordinate of storage travel detection 1	9999
------	---	------

Setting range: -9999.9999 ~ 9999.9999 (mm)

0070	Negative Z axis stroke coordinate of storage travel detection 1	-9999
------	---	-------

Setting range: -9999.9999 ~ 9999.9999 (mm)

0071	Positive Z axis stroke coordinate of storage travel detection 1	9999
------	---	------

Setting range: -9999.9999 ~ 9999.9999 (mm)

0072	Negative 4TH axis stroke coordinate of storage travel detection 1	-9999
------	---	-------

Setting range: -9999.9999 ~ 9999.9999 (mm)

0073	Positive 4TH axis stroke coordinate of storage travel detection 1	9999
------	---	------

Setting range: -9999.9999~9999.9999 (mm)

0076	Negative X axis stroke coordinate of storage travel detection 2	-9999
------	---	-------

Setting range: -9999.9999~9999.9999 (mm)

0077	Positive X axis stroke coordinate of storage travel detection 2	9999
------	---	------

Setting range: -9999.9999~9999.9999 (mm)

0078	Negative Y axis stroke coordinate of storage travel detection 2	-9999
------	---	-------

Setting range: -9999.9999~9999.9999 (mm)

0079	Positive Y axis stroke coordinate of storage travel detection 2	9999
------	---	------

Setting range: -9999.9999~9999.9999 (mm)

0080	Negative Z axis stroke coordinate of storage travel detection 2	-9999
------	---	-------

Setting range: -9999.9999~9999.9999 (mm)

0081	Negative Z axis stroke coordinate of storage travel detection 2	9999
------	---	------

Setting range: -9999.9999~9999.9999 (mm)

0082	Negative 4TH axis stroke coordinate of storage travel detection 2	-9999
------	---	-------

Setting range: -9999.9999~9999.9999 (mm)

0083	Positive 4TH axis stroke coordinate of storage travel detection 2	9999
------	---	------

Setting range: -9999.9999~9999.9999 (mm)

0086	Dry run speed	5000
------	---------------	------

Setting range: 0~9999 (mm/min)

## Appendix

---

0087	Cutting feedrate at power-on	300
------	------------------------------	-----

Setting range: 0~9999 (mm/min)

0088	Rapid traverse speed of X axis	5000
------	--------------------------------	------

Setting range: 0~3000 (mm/min)

0089	Rapid traverse speed of Y axis	5000
------	--------------------------------	------

Setting range: 0~3000 (mm/min)

0090	Rapid traverse speed of Z axis	5000
------	--------------------------------	------

Setting range: 0~3000 (mm/min)

0091	Rapid traverse speed of 4TH axis	5000
------	----------------------------------	------

Setting range: 0~3000 (mm/min)

0093	F0 rapid override of axis (for all axes)	30
------	--	----

Setting range: 0~1000 (mm/min)

0094	Maximum feedrate (for all axes)	8000
------	---------------------------------	------

Setting range: 300~9999(mm/min) Maximum control speed in non-forecast mode

0095	Minimum feedrate (for all axes)	0
------	---------------------------------	---

Setting range: 0~300(mm/min) Minimum control speed in non-forecast mode

0096	Maximum speed in forecasting control mode (for all axes)	6000
------	--	------

Setting range: 300~9999(mm/min)

0097	Minimum speed in forecasting control mode (for all axes)	0
------	--	---

Setting range: 0~300(mm/min)

0098	Feedrate of manual continuous feed for axes (JOG)	2000
------	---	------

Setting range: 0~9999 (mm/min)

0099	Speed(FL) of reference point return (for all axes)	60
------	--	----

Setting range: 0~9999 (mm/min)



0100	X axis reference point return speed	4000
------	-------------------------------------	------

Setting range: 0~9999 (mm/min)

0101	Y axis reference point return speed	4000
------	-------------------------------------	------

Setting range: 0~9999 (mm/min)

0102	Z axis reference point return speed	4000
------	-------------------------------------	------

Setting range: 0~9999 (mm/min)

0103	4 <sup>TH</sup> axis reference point return speed	4000
------	---	------

Setting range: 0~9999 (mm/min)

0105	L type time constant of pre-acceleration /deceleration of rapid X axis	100
------	--	-----

Setting range: 0~400 (ms)

0106	L type time constant of pre-acceleration /deceleration of rapid Y axis	100
------	--	-----

Setting range: 0~400 (ms)

0107	L type time constant of pre-acceleration /deceleration of rapid Z axis	100
------	--	-----

Setting range: 0~400 (ms)

0108	L type time constant of pre-acceleration /deceleration of rapid 4TH axis	100
------	--	-----

Setting range: 0~400 (ms)

0110	S type time constant of pre-acceleration /deceleration of rapid X axis	100
------	--	-----

Setting range: 0~400 (ms)

0111	S type time constant of pre-acceleration /deceleration of rapid Y axis	100
------	--	-----

Setting range: 0~400 (ms)

0112	S type time constant of pre-acceleration /deceleration of rapid Z axis	100
------	--	-----

Setting range: 0~400 (ms)

## Appendix

---

0113	S type time constant of pre-acceleration /deceleration of rapid 4TH axis	100
------	--	-----

Setting range: 0~400 (ms)

0115	L type time constant of post acceleration /deceleration of rapid X axis	80
------	---	----

Setting range: 0~400 (ms)

0116	L type time constant of post acceleration /deceleration of rapid Y axis	80
------	---	----

Setting range: 0~400 (ms)

0117	L type time constant of post acceleration /deceleration of rapid Z axis	80
------	---	----

Setting range: 0~400 (ms)

0118	L type time constant of post acceleration /deceleration of rapid 4TH axis	80
------	---	----

Setting range: 0~400 (ms)

0120	E type time constant of post acceleration /deceleration of rapid X axis	60
------	---	----

Setting range: 0~400 (ms)

0121	E type time constant of post acceleration /deceleration of rapid Y axis	60
------	---	----

Setting range: 0~400 (ms)

0122	E type time constant of post acceleration /deceleration of rapid Z axis	60
------	---	----

Setting range: 0~400 (ms)

0123	E type time constant of post acceleration /deceleration of rapid 4TH axis	60
------	---	----

Setting range: 0~400 (ms)

0125	L type time constant of pre-acceleration /deceleration of cutting feed	100
------	--	-----

Setting range: 0~400 (ms)

0126	S type time constant of pre-acceleration /deceleration of cutting feed	100
------	--	-----

Setting range: 0~400 (ms)

0127	L type time constant of post acceleration /deceleration of cutting feed	80
------	---	----

Setting range: 0~400 (ms)

0128	E type time constant of post acceleration /deceleration of cutting feed	60
------	---	----

Setting range: 0~400 (ms)

0129	FL speed of exponential acceleration /deceleration	10
------	--	----

Setting range: 0~9999 (mm/min)

0130	Maximum blocks merged in pre-interpolation	2
------	--	---

Setting range: 0~10

0131	In-position precision of cutting feed	0.03
------	---------------------------------------	------

Setting range: 0~0.5 (mm)

0132	Control precision of circular interpolation	0.03
------	---	------

Setting range: 0~0.5 (mm)

0133	Contour control precision of pre-interpolation	0.01
------	--	------

Setting range: 0~0.5 (mm)

0134	Acceleration of the fore linear acceleration /deceleration interpolated in forecasting control	250
------	--	-----

Setting range: 0~2000 (mm/s<sup>2</sup>)

0135	Forecasting control, S type pre-acceleration /deceleration time constant	100
------	--	-----

Setting range: 0~400 (ms)

0136	Linear time constant of the post acceleration /deceleration in forecasting control	80
------	--	----

Setting range: 0~400 (ms)

## Appendix

---

0137	Exponential time constant of the post acceleration /deceleration in forecasting control	60
------	---	----

Setting range: 0~400 (ms)

0138	Exponential acceleration/deceleration FL speed of cutting feed in forecasting control	10
------	---	----

Setting range: 0~400 (ms)

0139	Contour control precision in forecasting control	0.01
------	--	------

Setting range: 0~0.5 (mm)

0140	Blocks merged in forecasting control	0
------	--------------------------------------	---

Setting range: 0~10

0141	In-position precision in forecasting control	0.05
------	--	------

Setting range: 0~0.5 (mm)

0142	Length condition of spline formation in forecasting control	5
------	---	---

Setting range: 0~30

0143	Angular condition of spline formation in forecasting control	10
------	--	----

Setting range: 0~30

0144	Critical angle of the two blocks during automatic corner deceleration in forecasting control	5
------	--	---

Setting range: 2~178 (mm/min)

0145	Minimum feedrate of automatic corner deceleration in forecasting control	120
------	--	-----

Setting range: 10~1000 (mm/min)

0146	Axis error allowable for speed difference deceleration in forecasting control	80
------	---	----

Setting range: 60~1000

0147	Cutting precision grade in forecasting control	2
------	--	---

Setting range: 0~8

0148	External acceleration limit of circular interpolation	1000
------	---	------

Setting range: 100~5000 (mm/s<sup>2</sup>)

0149	Lower limit of the external acceleration clamp for circular interpolation	200
------	---	-----

Setting range: 0~2000 (mm/min)

0150	Acceleration clamp time constant of cutting feed	50
------	--	----

Setting range: 0~1000 (ms)

0151	Maximum clamp speed of handwheel incomplete running	2000
------	---	------

Setting range: 0~3000 (mm/min)

0152	Linear acceleration /deceleration time constant of handwheel	120
------	--	-----

Setting range: 0~400 (ms)

0153	Exponential acceleration /deceleration time constant of handwheel	80
------	---	----

Setting range: 0~400 (ms)

0154	Acceleration clamp time constant of handwheel	100
------	---	-----

Setting range: 0~400 (ms)

0155	Maximum clamp speed of step feed	1000
------	----------------------------------	------

Setting range: 0~3000 (mm/min)

0156	Linear acceleration /deceleration time constant of axes JOG feed	100
------	--	-----

Setting range: 0~400 (ms)

0157	Exponential acceleration /deceleration time constant of axes JOG feed	120
------	---	-----

Setting range: 0~400 (ms)

0160	Multiplication coefficient of X axis instruction(CMR)	1
------	---	---

Setting range: 1~65535

## Appendix

---

0161	Multiplication coefficient of Y axis instruction (CMR)	1
------	--	---

Setting range: 1~65535

0162	Multiplication coefficient of Z axis instruction (CMR)	1
------	--	---

Setting range: 1~65535

0163	Multiplication coefficient of 4TH axis instruction (CMR)	1
------	--	---

Setting range: 1~65535

0165	Frequency dividing coefficient of X axis instruction(CMD)	1
------	---	---

Setting range: 1~65535

0166	Frequency dividing coefficient of Y axis instruction(CMD)	1
------	---	---

Setting range: 1~65535

0167	Frequency dividing coefficient of Z axis instruction(CMD)	1
------	---	---

Setting range: 1~65535

0168	Frequency dividing coefficient of 4TH axis instruction(CMD)	1
------	---	---

Setting range: 1-65535

0170	X axis manual rapid positioning speed	5000
------	---------------------------------------	------

Setting range: 0~30000

0171	Y axis manual rapid positioning speed	5000
------	---------------------------------------	------

Setting range: 0~30000

0172	Z axis manual rapid positioning speed	5000
------	---------------------------------------	------

Setting range: 0~30000

0173	4TH axis manual rapid positioning speed	5000
------	---	------

Setting range: 0~30000

0189	Reverse precision by backlash compensation	0.0100
------	--	--------

Setting range: 0.0001~1.0000 (mm)

Set  $\alpha = p(189) \times 0.0001$ , after reverse feeding, if the feeding of single servo period is over  $\alpha$ , the backlash compensation begins.

Therefore, in machining outer circle contour with a larger radius, in order to make the offset position not to exceed the quadrant, it needs to set a smaller precision. While in machining a curve surface, in order to not to perform backlash compensation in a fixed point of the tool path to form a swollen ridge, it needs to set a larger precision to make the clearance compensation to be distributed in a certain width.

0190	Backlash compensation amount of X axis	0.0000
------	--	--------

Setting range: 0~99.9999 (mm)

0191	Backlash compensation amount of Y axis	0.0000
------	--	--------

Setting range: 0~99.9999 (mm)

0192	Backlash compensation amount of Z axis	0.0000
------	--	--------

Setting range: 0~99.9999 (mm)

0193	Backlash compensation amount of 4TH axis	0.0000
------	--	--------

Setting range: 0~99.9999 (mm)

0195	Compensation step of X axis clearance by fixed frequency	0.0030
------	--	--------

Setting range: 0~99.9999 (mm)

0196	Compensation step of Y axis clearance by fixed frequency	0.0030
------	--	--------

Setting range: 0~99.9999 (mm)

0197	Compensation step of Z axis clearance by fixed frequency	0.0030
------	--	--------

Setting range: 0~99.9999 (mm)

0198	Compensation step of 4TH axis clearance by fixed frequency	0.0030
------	--	--------

Setting range: 0~99.9999 (mm)

## Appendix

---

0200	Time constant of backlash compensation by ascending and descending	20
------	--	----

Setting range: 0~400 (ms)

0201	Delay time of strobe signals MF, SF, TF	0
------	---	---

Setting range: 0~9999 (ms)

0202	Width acceptable for M, S, T completion signal	0
------	--	---

Setting range: 0~9999 (ms)

0203	Output time of reset signal	200
------	-----------------------------	-----

Setting range: 50~400 (ms)

0204	Bits allowable for M codes	2
------	----------------------------	---

Setting range: 1~2

0205	Bits allowable for S codes	5
------	----------------------------	---

Setting range: 1~6

0206	Bits allowable for T codes	4
------	----------------------------	---

Setting range: 1~4

0210	Incremental amount for automatic sequence number insertion	10
------	--	----

Setting range: 0~1000

0211	Tool offset heading number input disabled by MDI	0
------	--	---

Setting range: 0~9999

0212	Tool offset numbers input by MDI disabled	0
------	---	---

Setting range: 0~9999

0214	Error limit of arc radius	0.05
------	---------------------------	------

Setting range: -0.1000~0.1000 (mm)

0216	Pitch error compensation number of X axis reference point	0
------	---	---

Setting range: 0~9999



0217	Pitch error compensation number of Y axis reference point	0
------	---	---

Setting range: 0~9999

0218	Pitch error compensation number of Z axis reference point	0
------	---	---

Setting range: 0~9999

0219	Pitch error compensation number of 4TH axis reference point	0
------	---	---

Setting range: 0~9999

0221	Pitch error compensation points of X axis	256
------	---	-----

Setting range: 0~1000

0222	Pitch error compensation points of Y axis	256
------	---	-----

Setting range: 0~1000

0223	Pitch error compensation points of Z axis	256
------	---	-----

Setting range: 0~1000

0224	Pitch error compensation points of 4TH axis	256
------	---	-----

Setting range: 0~1000

0226	Pitch error compensation interval of X axis	5
------	---	---

Setting range: 0~99.9999 (mm)

0227	Pitch error compensation interval of Y axis	5
------	---	---

Setting range: 0~99.9999 (mm)

0228	Pitch error compensation interval of Z axis	5
------	---	---

Setting range: 0~99.9999 (mm)

0229	Pitch error compensation interval of 4TH axis	5
------	---	---

Setting range: 0~99.9999 (mm)

0231	Pitch error compensation override of X axis	0.001
------	---	-------

Setting range: 0~99.9999

## Appendix

---

0232	Pitch error compensation override of Y axis	0.001
------	---	-------

Setting range: 0~99.9999

0233	Pitch error compensation override of Z axis	0.001
------	---	-------

Setting range: 0~99.9999

0234	Pitch error compensation override of 4TH axis	0.001
------	---	-------

Setting range: 0~99.9999

0240	Gain adjustment data for spindle analog output	0
------	--	---

Setting range: 0~9999

0241	Compensation value of offset voltage for spindle analog output	0
------	--	---

Setting range: 0~9999

0242	Spindle speed at spindle orientation, or motor speed at spindle gear shift	50
------	--	----

Setting range: 0~9999(r/min)

0246	Spindle maximum speed to gear 1	5000
------	---------------------------------	------

Setting range: 0~99999 (r/min)

0247	Spindle maximum speed to gear 2	5000
------	---------------------------------	------

Setting range: 0~99999 (r/min)

0248	Spindle maximum speed to gear 3	5000
------	---------------------------------	------

Setting range: 0~99999 (r/min)

0250	Spindle motor speed of gear 1—gear 2 shift	50
------	--	----

Setting range: 0~1000 (r/min)

0251	Maximum spindle motor speed of shifting	50
------	---	----

设定范围: 0~99999 (r/min)

0254	Axis as counting for surface speed control	0
------	--	---

Setting range: 0~5

0255	Spindle minimum speed for constant surface speed control (G96)	100
------	--	-----

Setting range: 0~9999 (r/min)

0257	Spindle upper limit speed in tapping cycle	2000
------	--	------

Setting range: 0~5000 (r/min)

0258	Spindle upper limit speed	5000
------	---------------------------	------

Setting range: 0~99999 (r/min)

0259	Spindle servo loop gain	0
------	-------------------------	---

Setting range: 0~9999

0260	Spindle speed baudrate with no alarm for spindle speed monitoring	0
------	---	---

Setting range: 0~9999

0261	Spindle encoder lines	1024
------	-----------------------	------

Setting range: 0~100000

0262	Spindle override lower limit	0.0000
------	------------------------------	--------

Setting range: 0~1

0266	Limit with vector ignored when moving along outside corner in tool radius compensation C	0
------	--	---

Setting range: 0~9999.9999

0267	Maximum value of tool wear compensation	400.0000
------	---	----------

Setting range: 0~999.9999 (mm)

0268	Maximun error of tool radius compensation C	0.001
------	---	-------

Setting range: 0.0001~0.01

0269	Coefficient of helical feed radius in groove cycle	1.5000
------	--	--------

Setting range: 0.0100~3.0000

0270	Retraction amount of high-speed peck drilling cycle G73	2.0000
------	---	--------

Setting range: 0~999.9999 (mm)

## Appendix

---

0271	Reserved space amount of canned cycle G83	2.0000
------	---	--------

Setting range: 0~999.9999 (mm)

0272	Spindle speed change ratio in tool retraction without overload torque signal	0
------	--	---

Setting range: 0~9999.0000

0273	Spindle speed change ratio in tool retraction with overload torque signal received	0
------	--	---

Setting range: 0~9999.0000

0274	Cutting feedrate change ratio in tool retraction without overload torque signal	0
------	---	---

Setting range: 0~9999.0000

0275	Cutting feedrate change ratio in small peck drilling cycle	0
------	--	---

Setting range: 0~9999.0000

0276	Macro variable number of retraction actions during output cutting	0
------	---	---

Setting range: 0~9999.0000

0277	Macro variable number output of retraction actions due to overload signal	0
------	---	---

Setting range: 0~9999.0000

0278	Traverse speed back to point R with address I not specified	0
------	---	---

Setting range: 0~9999.0000

0279	Traverse speed to the hole bottom with address I not specified	0
------	--	---

Setting range: 0~9999.0000

0280	Clearance of small peck drilling cycle	0
------	--	---

Setting range: 0~9999.0000

0281	Minimum dwell time at the hole bottom	0
------	---------------------------------------	---

Setting range: 0~1000 (ms)

0282	Maximum dwell time at the hole bottom	9999
------	---------------------------------------	------

Setting range: 1000~9999 (ms)

0283	Override for retraction in rigid tapping	1.0000
------	--	--------

Setting range: 0.8000~1.2000

0284	Retraction or spacing amount in peck tapping cycle	0
------	--	---

Setting range: 0~100 (mm)

0285	Synch error range setting for rigid tapping	0
------	---	---

Setting range: 0~100 (mm)

0286	Tooth number of spindle side gear(1st gear)	1
------	---	---

Setting range: 1~999

0287	Tooth number of spindle side gear(2nd gear)	1
------	---	---

Setting range: 1~999

0288	Tooth number of spindle side gear(3rd gear)	1
------	---	---

Setting range: 1~999

0290	Tooth number of position encoder side gear(1st gear)	1
------	--	---

Setting range: 1~999

0291	Tooth number of position encoder side gear(2nd gear)	1
------	--	---

Setting range: 1~999

0292	Tooth number of position encoder side gear (3rd gear)	1
------	---	---

Setting range: 1~999

0294	Maximum spindle speed in rigid tapping(1st gear)	500
------	--	-----

Setting range: 0~9999 (r/min)

## Appendix

---

0295	Maximum spindle speed in rigid tapping(2nd gear)	1000
------	--	------

Setting range: 0~9999 (r/min)

0296	Maximum spindle speed in rigid tapping(3rd gear)	2000
------	--	------

Setting range: 0~9999 (r/min)

0298	Linear acceleration/deceleration time constants of spindle and tapping axis(1st gear)	200
------	---	-----

Setting range: 0~400 (ms)

0299	Linear acceleration/deceleration time constants of spindle and tapping axis(2nd gear)	200
------	---	-----

Setting range: 0~400 (ms)

0300	Linear acceleration/deceleration time constants of spindle and tapping axis(3rd gear)	200
------	---	-----

Setting range: 0~400 (ms)

0302	Time constants of spindle and tapping axis in retraction (1st gear)	200
------	---	-----

Setting range: 0~9999 (ms)

0303	Time constants of spindle and tapping axis in retraction (2nd gear)	200
------	---	-----

Setting range: 0~9999 (ms)

0304	Time constants of spindle and tapping axis in retraction (3rd gear)	200
------	---	-----

Setting range: 0~9999 (ms)

0306	Position control loop gain of spindle and tapping axis in rigid tapping (1st gear)	0
------	--	---

Setting range: 0~9999

0307	Position control loop gain of spindle and tapping axis in rigid tapping (2nd gear)	0
------	--	---

Setting range: 0~9999

0308	Position control loop gain of spindle and tapping axis in rigid tapping(3rd gear)	0
------	---	---

Setting range: 0~9999

0310	Spindle loop gain coefficient in rigid tapping (1st gear)	0
------	---	---

Setting range: 0~9999.9999

0311	Spindle loop gain coefficient in rigid tapping (2nd gear)	0
------	---	---

Setting range: 0~9999.9999

0312	Spindle loop gain coefficient in rigid tapping (3rd gear)	0
------	---	---

Setting range: 0~9999.9999

0314	Spindle in-position width in rigid tapping	0
------	--	---

Setting range: 0~100

0315	Tapping axis in-position width in rigid tapping	0
------	---	---

Setting range: 0~100

0316	Position error limit of tapping axis moving in rigid tapping	0
------	--	---

Setting range: 0~100

0317	Position error limit of spindle moving in rigid tapping	0
------	---	---

Setting range: 0~100

0318	Error limit at tapping axis stopping in rigid tapping	0
------	---	---

Setting range: 0~100

0319	Error limit at spindle stopping in rigid tapping	0
------	--	---

Setting range: 0~100

0320	Spindle clearance in rigid tapping (1st gear)	0
------	---	---

Setting range: 0~99.9999

0321	Spindle clearance in rigid tapping (2nd gear)	0
------	---	---

Setting range: 0~99.9999

## Appendix

---

0322	Spindle clearance in rigid tapping (3rd gear)	0
------	---	---

Setting range: 0~99.9999

0323	Spindle instruction multiplication coefficient (CMR) (1st gear)	512
------	---	-----

Setting range: 1~9999

0324	Spindle instruction multiplication coefficient (CMR) (2nd gear)	512
------	---	-----

Setting range: 1~9999

0325	Spindle instruction multiplication coefficient (CMR) (3rd gear)	512
------	---	-----

Setting range: 1~9999

0326	Spindle instruction frequency dividing coefficient (CMD) (1st gear)	125
------	---	-----

Setting range: 1~9999

0327	Spindle instruction frequency dividing coefficient (CMD) (2nd gear)	125
------	---	-----

Setting range: 1~9999

0328	Spindle instruction frequency dividing coefficient (CMD) (3rd gear)	125
------	---	-----

Setting range: 1~9999

0329	Rotational angle with no rotational angle specified in coordinate rotation	0
------	--	---

Setting range: 0~9999.9999

0330	Scaling with no scaling specified	1
------	-----------------------------------	---

Setting range: 0~9999.9999

0331	Scaling of X axis	1
------	-------------------	---

Setting range: 0~9999.9999

0332	Scaling of Y axis	1
------	-------------------	---

Setting range: 0~9999.9999



0333	Scaling of Z axis	1
------	-------------------	---

Setting range: 0~9999.9999

0334	Dwell time of unidirectional positioning	0
------	--	---

Setting range: 0~10 (s)

0335	Direction and overtravel amount of X axis unidirectional positioning	0
------	--	---

Setting range: -99.9999~99.9999

0336	Direction and overtravel amount of Y axis unidirectional positioning	0
------	--	---

Setting range: -99.9999~99.9999

0337	Direction and overtravel amount of Z axis unidirectional positioning	0
------	--	---

Setting range: -99.9999~99.9999

0338	Direction and overtravel amount of 4TH axis unidirectional positioning	0
------	--	---

Setting range: -99.9999~99.9999

0340	Axis number of controlled axis in normal direction	0
------	--	---

Setting range: 0~9999.9999

0341	Rotation speed of controlled axis in normal direction	0
------	---	---

Setting range: 0~9999.9999

0342	Rotation insertion ineffective limit of controlled axis in normal direction	0
------	---	---

Setting range: 0~9999.9999

0343	Moving limit to be executed by the last program normal angle	0
------	--	---

Setting range: 0~9999.9999

0344	Rotation limit of the controlled axis in normal direction inserted by a single block	0
------	--	---

Setting range: 0~9999.9999

## Appendix

---

0345	Minimum angle of indexing table	0
------	---------------------------------	---

Setting range: 0~9999.9999

0350	Feedrate by tool length measurement	0
------	-------------------------------------	---

Setting range: 0~1000

0351	r value by tool length measurement	0
------	------------------------------------	---

Setting range: 0~9999.9999

0352	e value by tool length measurement	0
------	------------------------------------	---

Setting range: 0~9999.9999

0356	Workpieces machined	0
------	---------------------	---

Setting range: 0~9999

0357	Total workpieces to be machined	0
------	---------------------------------	---

Setting range: 0~9999

0358	Accumulative time of power-on (hour)	0
------	--------------------------------------	---

Setting range: 0~9999

0360	Accumulative time of cutting (hour)	0
------	-------------------------------------	---

Setting range: 0~9999

0361	Tool life management signal ignored	0
------	-------------------------------------	---

Setting range: 0~9999

0362	Tool life left (using times)	0
------	------------------------------	---

Setting range: 0~9999

0363	Tool life left (using time)	0
------	-----------------------------	---

Setting range: 0~9999

0365	Number of MPG used	0
------	--------------------	---

Setting range: 0~9999

0371	Positioning error allowable for reverse X axis	0.0050
------	--	--------

Setting range: 0~99.9999 (mm)

0372	Positioning error allowable for reverse Y axis	0.0050
------	--	--------

Setting range: 0~99.9999 (mm)

0373	Positioning error allowable for reverse Z axis	0.0050
------	--	--------

Setting range: 0~99.9999 (mm)

0374	Positioning error allowable for reverse 4TH axis	0.0050
------	--	--------

Setting range: 0~99.9999 (mm)

0376	Axes moving sequence to program beginning	12345
------	---	-------

Setting range: 0~99999

0380	Referential counter capacity of X axis	0
------	--	---

Setting range: 0~9999

0381	Referential counter capacity of Y axis	0
------	--	---

Setting range: 0~9999

0382	Referential counter capacity of Z axis	0
------	--	---

Setting range: 0~9999

0383	Referential counter capacity of 4TH axis	0
------	--	---

Setting range: 0~9999

## APPENDIX 2

### Alarm List

Alarm No.	Content	Remark
0000	Parameter for cutting off power once is modified	
0001	file open fail	
0002	data input overflow	
0003	program number already in use	
0004	address not found	
0005	no data behind address	
0006	illegal negative sign	
0007	illegal decimal point	
0008	the program file is too large to be loaded completely	
0009	illegal address	
0010	G code wrong	
0011	no feedrate instruction	
0012	disk space is not enough	
0013	the program files are up to the upper limit	
0014	G95 can't be specified, it is not supported by the spindle	
0015	too many axes	
0016	current pitch compensation beyond range	
0017	no authority to modify	
0018	not allowed to modify	
0019	Scaling function is OFF	
0020	beyond radius tolerance	
0021	illegal plane axis	
0022	arc R, I, J, K are all zero	
0023	R, I, J, K of circular interpolation specified together	
0024	Helical interpolation rotation angle is 0	
0025	G12 and other G code can't be in a same block	
0027	no axis instruction in G43/G44	
0028	illegal plane selection	
0029	illegal offset value	

0030	illegal compensation number	
0031	illegal P specified in G10	
0032	illegal compensation value in G10	
0033	no intersecting point in offset C	
0034	start-up disabled or offset cancelled in arc instruction	
0035	the compensation instruction changed when establishing tool offset	
0036	G31 can't be instructed	
0037	plane change disabled in offset C	
0038	interference in arc block	
0039	tool nose positioning error in offset C	
0040	To change the workpiece coordinate system in offset C executing	
0041	interference in offset C	
0042	more than ten nonmovable instructions in offset C	
0044	G27~G30 instruction can't be instructed in canned cycle	
0045	Address Q not found or Q is 0 (G73/G83)	
0046	illegal reference point return	
0047	machine zero should be executed before executing the instruction	
0048	Z level higher than R level	
0049	Z level lower than R level	
0050	position changed when canned cycle mode is changed	
0051	incorrect move after chamfering	
0052	not G01 code after chamfering	
0053	too many address instructions	
0054	DNC transfer setting wrong	
0055	move value wrong in chamfering or corner rounding	
0058	end point not found	
0059	program number not found	
0060	sequence number not found	
0061	X axis not on the reference point	
0062	Y axis not on the reference point	
0063	Z axis not on the reference point	
0064	4th axis not on the reference point	
0066	canned cycle must be cancelled before executing G10	

## Appendix

0067	the setting format is not supported by G10	
0070	storage or memory full	
0071	data end not found	
0072	too many programs	
0073	program number already in use	
0074	illegal program number	
0075	protection	
0076	address P not defined	
0077	subprogram nesting error	
0078	program number not found	
0082	H code specified in G37	
0083	illegal axis instruction in G37	
0084	Overtime or short circuit occurs in key	
0085	communication error	
0086	Planes can not be shifted in canned cycle mode	
0087	X axis reference point return unfinished	
0088	Y axis reference point return unfinished	
0089	Z axis reference point return unfinished	
0090	4th axis reference point return unfinished	
0091	5th axis reference point return unfinished	
0092	axis not on the reference point	
0094	P type not allowed(coordinate)	
0095	P type not allowed(EXT OFS CHG)	
0096	P type not allowed(WRK OFS CHG)	
0097	P type not allowed (auto execution)	
0098	G28 found in sequence return	
0099	MDI not allowed after retrieval	
0100	parameter write effective	
0101	Memory data disordered after power off, please ensure correct location	
0110	data overflow	
0111	operated data overflow	
0112	divided by zero	
0113	improper instruction	
0114	macro format error	

0115	illegal variable	
0116	write protected variable	
0117	G10 online modification is not supported by this parameter. Please modify the program	
0118	parenthesis nesting error	
0119	M00~M02, M06, M98, M99 ,and M30 can't be in a same block with other M codes	
0120	Part of setting is restored	
0122	quadruplicate macro-mode calling	
0123	macro unallowed in DNC	
0124	Illegal program end	
0125	macro format error	
0126	illegal loop number	
0127	NC and macro in a same block	
0128	sequence number by illegal macro	
0129	illegal argument address	
0130	illegal axis operation	
0131	too many external alarm messages	
0132	alarm number not found	
0133	unsupported axis instruction	
0134	Rigid tapping can not be used when CNC controlled axes exceed 3	
0135	illegal angle instruction	
0136	illegal axis instruction	
0139	PLC axis change disabled	
0142	illegal scaling	
0143	scaling motion data overflow	
0144	illegal plane selection	
0148	illegal data setting	
0149	format error in G10L3	
0150	illegal tool group number	
0151	tool group number not found	
0152	no space for tool data	
0153	T code not found	
0154	not using tool in life group	

## Appendix

---

0155	illegal T code in M06	
0156	P/L instruction not found	
0157	too many tool groups	
0158	illegal tool life data	
0159	tool data setting unfinished	
0160	arc programming only by R in polar system	
0161	The instruction can't be executed in polar coordinate mode	
0163	The instruction can't be executed in revolution mode	
0164	The instruction can't be executed in scaling mode	
0165	Please specify the instruction in a single block	
0166	No axis specified in reference point return	
0167	intermediate point coordinate too large	
0168	the min. dwell time at the hole bottom should be shorter than the max. dwell time	
0170	tool radius compensation not cancelled	
0172	P not integer or less than 0 in a block calling subprogram	
0173	Subprogram called beyond 9999 times	
0175	canned cycle can only be executed in G17 plane	
0176	spindle speed not specified before rigid tapping	
0177	spindle orientation not supported	
0178	spindle speed not specified before canned cycle	
0181	illegal M code	
0182	illegal S code	
0183	illegal T code	
0184	tool selected beyond range	
0185	L too small or undefined	
0186	L too large	
0187	Tool radius too large	
0188	U too large	
0189	U less than zero	
0190	V too small or undefined	
0191	W too small or undefined	
0192	Q too small or undefined	
0193	I undefined or I for zero	
0194	J undefined or J for zero	



0195	D undefined or D for zero	
0198	Illegal axis selection	
0199	macro not defined	
0200	illegal S instruction	
0201	feedrate not found in rigid tapping	
0202	position LSI overflow	
0203	program wrong in rigid tapping	
0204	illegal axis operation	
0205	rigid mode DI signal off	
0206	can't change plane(rigid tapping)	
0207	tapping data wrong	
0208	This instruction can not be executed in G10 mode. Please cancel G10 mode first	
0209	Restart of the program is not supported by scaling, revolution, polar coordinate modes	
0210	Program name error after system restarting	
0212	illegal plane selection	
0213	Tool changing macro program does not support G31 skip	
0214	Tool changing macro program does not support skip operation	
0215	Tool changing macro program does not support modifying coordinate system and tool compensation dynamically	
0216	Scaling, revolution and polar coordinate do not support G31 skip	
0217	Scaling, revolution and polar coordinate do not support skip operation	
0218	Scaling, revolution and polar coordinate do not support modifying coordinate system and tool compensation dynamically	
0219	M06 Tool magazine is not used (parameter is not opened). Tool changing instruction can not be used	
0220	Metric/inch switching is not supported by scaling, revolution and polar	

	coordinate mode	
0221	Metric/inch switching is not supported by tool changing macro progra	
0224	reference point return	
0231	illegal format in G10 or L50 or L51	
0232	too many helical interpolation axes specified	
0233	device busy	
0235	end of recording	
0236	program restart parameter error	
0237	no decimal point	
0238	address repetition error	
0239	parameter is 0	
0240	G41/G42 disabled in MDI mode	
0251	emergency stop alarm	
0300	n-axis origin return	
0301	APC alarm: n-axis communication	
0302	APC alarm: n-axis overtime	
0303	APC alarm: n-axis data format	
0304	APC alarm: n-axis parity	
0305	APC alarm: n-axis pulse error	
0306	APC alarm: n-axis battery voltage 0	
0307	APC alarm: n-axis battery voltage low 1	
0308	APC alarm: n-axis battery voltage low 2	
0309	APC alarm: n-axis ZRN impossible	
0350	SPC alarm: n axis pulse encoder	
0351	SPC alarm: n-axis communication	
0401	servo alarm: n-axis VRDY off	
0402	Drive unit alarm 02: power of spindle circuit is too high	
0403	Drive unit alarm 03: main circuit power source is too low	
0404	servo alarm: n-axis VRDY on	
0405	servo alarm: (zero return error)	
0407	servo alarm: superheterodyning	
0408	Drive unit alarm 08: absolute value of value of position deviation counter exceeds 230	
0409	torque alarm: superheterodyning	
0410	servo alarm: n-axis superheterodyning	

0411	servo alarm: n-axis superheterodyning	
0412	Drive unit alarm 12: motor current is too large	
0413	servo alarm: n-axis LSI overflow	
0414	servo alarm: n-axis detection error	
0415	servo alarm: n-axis move too fast	
0420	synch torque: superheterodyning	
0430	Drive unit alarm 30: encoder Z pulse error	
0431	Drive unit alarm 31: encoder UVW signal error or it does not match encoder	
0432	Drive unit alarm 32: UVW with all high level or with all low level	
0433	Drive unit alarm 33: communication interrupted	
0449	n-axis: INV.IPM alarm	
0450	Drive unit is disconnected.	
0451	X axis driver alarm	
0452	Y axis driver alarm	
0453	Z axis driver alarm	
0454	4th axis driver alarm	
0456	spindle driver alarm	
0500	software overtravel: -X	
0501	software overtravel: +X	
0502	software overtravel: -Y	
0503	software overtravel: +Y	
0504	software overtravel: -Z	
0505	software overtravel: +Z	
0506	software overtravel: -4th	
0507	software overtravel: +4th	
0508	Software overtravel: - 5Th	
0509	Software overtravel: +5Th	
0510	hardware overtravel: -X	
0511	hardware overtravel: +X	
0512	hardware overtravel: -Y	
0513	hardware overtravel: +Y	
0514	hardware overtravel: -Z	
0515	hardware overtravel: +Z	
0516	hardware overtravel: -4th	

0517	hardware overtravel: +4th	
0518	Hardware overtravel: - 5Th	
0519	Hardware overtravel: +5Th	
1001	relay or coil address not set	
1002	functional instruction of code input not exist	
1003	incorrect COM / COME instruction use	
1004	User ladder beyond the maximum permissible lineage or step number	
1005	Incorrect END1,END2 functional instruction use	
1006	Illegal output in NET	
1007	PLC communication fail due to hardware failure or sysem interruption	
1008	functional instruction wrongly linked	
1009	network horizontal lines not linked	
1010	editing NET loss due to power-off in ladder editing	
1011	address data wrongly input	
1012	sign input undefined or address input beyond range	
1013	illegal character defined	
1014	CTR address repeated	
1015	functional instruction JMP(LBL10) wrongly processed or beyond the capacity	
1016	incomplete NET constitution	
1017	unsupported NET constitution exists	
1019	TMR address repeated	
1020	no parameter in functional instruction	
1021	PLC stopped automatically by system when executed overtime	
1022	please input functional code	
1023	Address or constant of functional instruction parameter is out of range	
1024	unnecessary relay or coil exists	
1025	Functional instruction output wrongly	
1026	NET link lineage beyond the supported range	
1027	an output address used in another place	
1028	File format wrong	
1029	File losses from ladder diagram being used	
1030	false vertical line in network	

1031	Message data area is full. Please reduce COD instruction data list capacity.	
1032	ladder 1 <sup>st</sup> level too large to be executed on time	
1033	SFT instructions beyond the max. allowed number	
1034	functional instruction DIFU/DIFD wrongly used	
1039	Instruction or network beyond executable area	
1040	Incorrect functional instruction CALL / SP / SPE use	
1041	Level conducting line in parallel with node network	
1042	PLC system parameter file is not loaded	

# **CNCmakers**

---

[www.CNCmakers.com](http://www.CNCmakers.com)

**Declaration!**

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

**Warning !**

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

**Caution !**

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

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

## Safety notes

### ■ Transportation and storage

- Do not pile up the packing boxes over 6 layers.
- Never climb the packing box, neither stand on it, nor place heavy objects on it.
- Do not move or drag the product by the cables connected to it.
- Avoid impact or scratch to the panel and screen.
- Packing box should be protected from dampness, insolation and drench.

### ■ Open-package inspection

- Confirm the product is the one you purchased after opening the package.
- Check whether the product is damaged during transportation.
- Confirm all the elements are complete without damage by referring to the list.
- If there is incorrect product type, incomplete accessories or damage, please contact us in time.

### ■ Connection

- Only qualified personnel can connect and inspect the system.
- The system must be earthed. The earth resistance should not be greater than  $0.1\Omega$ , and a neutral wire (zero wire) cannot be used as an earth wire.
- The connection must be correct and secured. Otherwise, the product may be damaged or unexpected results may occur.
- Connect the surge absorbing diode to the product in the specified direction; otherwise the product may be damaged.
- Turn off the power before inserting or unplugging a plug, or opening the electric cabinet.

### ■ Troubleshooting

- Turn off the power supply before troubleshooting or replacing components.
- Overhaul the system when there is a short circuit or overload, and do not restart it until the trouble is removed.
- Do not turn ON/OFF the product frequently, and the ON/OFF interval should be 1 minute at least.

## **Volume I Programming Description**

Introduce the technical specification, product type series and parameter configuration, command code and program format of 990MA machining center CNC system.

## **Volume II Function Description**

Introduce the main function of the GSK990MA series machining center CNC system.

## **Volume III Operation Description**

Introduce the relative operations of PLC software of the GSK990MA machining center CNC system.

## **Volume IV Installation and Connection**

Introduce the installation, connection and setting methods of the GSK990MA machining center CNC system.

## **Appendix**

Introduce the use explanations of the GSK990MA machining center CNC system and the appendix.



## Contents

<b>I</b>	<b>PROGRAMMING .....</b>	<b>1</b>
<b>1</b>	<b>Sequence Program Creating Process.....</b>	<b>3</b>
1.1	GSK990MA PLC specification .....	3
1.2	What is a sequence program .....	3
1.3	Assignment of interface specifications (step 1) .....	3
1.4	Establishment of ladder diagram (step 2) .....	4
1.5	Sequence program debugging (step 3) .....	4
<b>2</b>	<b>Sequence Program.....</b>	<b>5</b>
2.1	Execution process of sequence program .....	5
2.2	Cycle execution .....	6
2.3	Priority of execution(1st level, and 2nd level).....	6
2.4	Sequence program structure .....	7
2.5	Processing I/O ( input / output ) signals .....	9
2.5.1	Input signal processing .....	10
2.5.2	Output signal processing .....	10
2.5.3	Difference state of signals between 1st level and 2nd level.....	10
2.6	Interlocking .....	11
<b>3</b>	<b>Address .....</b>	<b>12</b>
3.1	Addresses from Machine tool to PLC (X) .....	12
3.1.1	Assignment of IO module X address.....	13
3.1.2	Assignment of MDI panel X address.....	13
3.2	Address (Y) from PLC to machine tool .....	14
3.2.1	Assignment of IO module Y address.....	14
3.2.2	Assignment of IO module Y address.....	14
3.3	Address (G) from PLC to CNC.....	15
3.4	Address (F) from CNC to PLC .....	16
3.5	Internal relay address (R) .....	16
3.6	Address of keep relay (K) .....	17
3.7	Addresses(A) for message selection displayed on CRT .....	17
3.8	Address of meter (C) .....	18
3.9	Meter preset address(DC) .....	18
3.10	Timer addresses (T) .....	18
3.11	Addresses of timer preset value (DT) .....	18
3.12	Address of data table (D) .....	19
3.13	Label address (L) .....	19
3.14	Subprogram numbers (P) .....	19
<b>4</b>	<b>PLC Basic Instruction .....</b>	<b>20</b>
4.1	RD, RD.NOT, WRT, WRT.NOT .....	21
4.2	AND, AND.NOT instructions .....	21

4.3	OR, OR.NOT instructions .....	22
4.4	OR. STK instruction.....	22
4.5	AND.STK instruction.....	23
<b>5</b>	<b>PLC Functional instructions .....</b>	<b>25</b>
5.1	END1 (1 <sup>st</sup> level sequence program end) .....	26
5.2	END2 (2 <sup>nd</sup> level sequence program end) .....	26
5.3	CALL (call subprogram) .....	27
5.4	CALLU (unconditional subprogram call) .....	27
5.5	SP (Subprogram) .....	28
5.6	SPE (subprogram end) .....	28
5.7	SET (set) .....	29
5.8	RST (reset) .....	30
5.9	JMPB (label jump) .....	30
5.10	LBL (Label) .....	31
5.11	TMR (timer) .....	32
5.12	TMRB (fixed timer) .....	33
5.13	TMRC (timer) .....	34
5.14	CTR (binary counter) .....	35
5.15	DEC (binary decode) .....	37
5.16	COD (binary code conversion) .....	38
5.17	COM (common line control) .....	40
5.18	COME (common line control end) .....	40
5.19	ROT (Binary rotation control) .....	41
5.20	SFT (shift register) .....	44
5.21	DIFU (rising edge check) .....	45
5.22	DIFD (falling edge check) .....	46
5.23	COMP (binary comparison) .....	47
5.24	COIN (coincidence check) .....	48
5.25	MOVN (transfer of data) .....	49
5.26	MOVB (transfer of one byte) .....	50
5.27	MOVW (transfer of two bytes) .....	50
5.28	XMOV (Binary index data transfer) .....	51
5.29	DSCH (binary data search) .....	53
5.30	ADD (addition) .....	54
5.31	SUB (binary subtraction) .....	55
5.32	ANDF (functional and) .....	56
5.33	ORF (functional or) .....	58
5.34	NOT (logical not) .....	59
5.35	EOR (exclusive or) .....	60
<b>6</b>	<b>Ladder Writing Limit.....</b>	<b>62</b>
<b>II</b>	<b>FUNCTION .....</b>	<b>65</b>
<b>1</b>	<b>Controlled Axis.....</b>	<b>67</b>

1.1	Output of movement state of an axis .....	67
1.2	Servo ready signal .....	68
<b>2</b>	<b>Preparation for Operation .....</b>	<b>69</b>
2.1	Emergency stop .....	69
2.2	CNC overtravel signal .....	69
2.3	Alarm signal .....	71
2.4	Running mode selection .....	71
2.5	Status output signal .....	72
<b>3</b>	<b>Manual Operation .....</b>	<b>73</b>
3.1	JOG feed/incremental feed .....	73
3.2	MPG/Step feed .....	75
<b>4</b>	<b>Reference Point Return .....</b>	<b>76</b>
4.1	Manual reference point return .....	76
4.2	Reference point return check signal .....	77
4.3	Area check signal .....	79
<b>5</b>	<b>Automatic Operation .....</b>	<b>80</b>
5.1	Cycle start/feed hold .....	80
5.2	Reset .....	82
5.3	Testing a program .....	83
5.3.1	Machine tool lock .....	83
5.3.2	Dry run .....	84
5.3.3	Single block .....	85
5.4	Optional block skip .....	86
5.5	Program restart .....	86
<b>6</b>	<b>Feedrate Control .....</b>	<b>88</b>
6.1	Rapid traverse rate .....	88
6.2	Feedrate override .....	89
6.3	Override cancel .....	89
<b>7</b>	<b>Auxiliary Function .....</b>	<b>90</b>
7.1	Miscellaneous function (M code) .....	90
7.2	Miscellaneous function (S code) .....	91
7.3	Miscellaneous function (T code) .....	92
7.4	Auxiliary function lock .....	96
<b>8</b>	<b>Spindle Speed Function .....</b>	<b>97</b>
8.1	Spindle speed control mode .....	97
8.1.1	Gear spindle .....	97
8.1.2	Analog spindle .....	97
8.2	Rigid tapping .....	100
<b>9</b>	<b>Programming Instrucion .....</b>	<b>101</b>

9.1	Customer macro program.....	101
9.2	Canned cycle.....	103
<b>10</b>	<b>Display/Setting.....</b>	<b>106</b>
10.1	Clock Function.....	106
10.2	Displaying operation history .....	106
10.3	Help function .....	106
<b>11</b>	<b>Measurement .....</b>	<b>107</b>
11.1	Skip function.....	107
<b>12</b>	<b>Panel locked setting.....</b>	<b>108</b>
<b>Appendix</b> .....		<b>109</b>
<b>III</b>	<b>PLC EDITTING.....</b>	<b>113</b>
<b>1</b>	<b>Summary.....</b>	<b>115</b>
<b>2</b>	<b>Software introduction .....</b>	<b>116</b>
2.1	Start software.....	116
2.2	Function introduction .....	116
<b>3</b>	<b>Software operarion .....</b>	<b>118</b>
3.1	Tool bar.....	118
3.1.1	Main tool bar.....	118
3.1.2	Edit tool bar .....	118
3.2	Selecting a graph.....	120
3.3	Editing a graph .....	121
3.3.1	Cutting .....	121
3.3.2	Copying .....	121
3.3.3	Paste .....	122
3.3.4	Deleting .....	122
3.3.5	Inserting one row .....	122
3.3.6	Deleting one row.....	122
3.3.7	Converting .....	122
3.4	Ladder annotation.....	123
3.4.1	Row annotation.....	123
3.4.2	Element annotation.....	123
3.5	Exporting .....	125
<b>IV</b>	<b>CONNECTION .....</b>	<b>127</b>
<b>1</b>	<b>System Structure and Installation.....</b>	<b>129</b>
1.1	System composition .....	129
1.2	System installation & connection .....	129
1.3	CNC system installation dimension .....	130
<b>2</b>	<b>Device Connection.....</b>	<b>134</b>

2.1	CNC external connection .....	134
2.2	Connection between system and driver .....	134
2.2.1	System interface .....	135
2.2.2	Interface signal list .....	135
2.2.3	Signal specification .....	136
2.2.4	Cable connection .....	138
2.3	RS232 standard serial interface .....	139
2.4	MPG ( handwheel ), hand unit connection .....	140
2.4.1	Interface signal list .....	140
2.4.2	Interface signal explanation .....	141
2.5	Spindle unit connection .....	143
2.5.1	Interface signal list .....	143
2.5.2	Interface signal explanation .....	144
2.5.3	DAP01 interface connection.....	144
2.5.4	Connection Circuit of DAP03 Interfaces .....	146
2.6	Power supply interface.....	147
<b>3</b>	<b>Machine Control I/O Interface.....</b>	<b>148</b>
3.1	Interface signal list .....	148
3.2	Input interface .....	148
3.2.1	Input interface principle.....	148
3.2.2	Input signal interface definition.....	149
3.3	Output signal.....	151
3.3.1	Output interface principle .....	151
3.3.2	Output signal interface definition.....	152
<b>4</b>	<b>Debugging Machine .....</b>	<b>154</b>
4.1	Debugging preparation .....	154
4.2	System power on .....	155
4.3	Emergency stop and limit.....	155
4.4	Gear ratio adjustment .....	157
4.5	Backlash compensation .....	159
4.6	Parameter of servo .....	160
4.7	Machine pitch compensation .....	162
4.8	Machine zero return .....	165
4.9	Input/output signal control of spindle CW/CCW .....	169
4.10	Spindle automatic gear change control.....	170
4.11	External cycle start and feed hold .....	172
4.12	External editing lock and external operator panel lock.....	173
4.13	Cooling, lubricant and chip removal control .....	173
4.14	Parameter of feedrate.....	174
4.15	Parameter of tapping .....	176
<b>Appendix 1:</b>	<b>.....</b>	<b>179</b>
	Guide of ladder for GSK990MA matching with turret tool magazine .....	179
	— <b>Notices of GSK990MA matching with turret tool magazine.....</b>	<b>179</b>

二.	Allocation and definition of GSK990MA PLC/IO address and internal auxiliary relay and register .....	179
三.	Usage and maintenance of GSK 990MA CNC System matching with turret tool magazine.....	203
四.	Macro program statement of GSK990MA CNC System matching with turret tool magazine .....	210
<b>APPENDIX 2 .....</b>		<b>211</b>
一.	F signal and meaning for M00-M99 of M code .....	211
二.	“%” that occupies a line exclusively means the end of M code information storage. ....	211
三.	The codes and meaning of X signal X0.0-X6.7 .....	211
四.	The codes and meaning of Y signal Y0.0---Y5.7 .....	212
五.	The codes and meaning of K signal Y6.0---Y63.7 .....	212
六.	The codes and meaning of A signal A0.0-A31.7 .....	212
七.	end//end sign .....	212

# I      **Programming**





## 1

## Sequence Program Creating Process

## 1.1 GSK990MA PLC specification

Specifications of GSK990MA PLC are as follows:

Specification	GSK990MA PLC
Programming method language	Ladder
Number of ladder level	2
1 <sup>st</sup> level execution period	8ms
Mean processing time of basic instruction	10 $\mu$ s
Program capacity	4700 step
Instruction	Basic instruction +function instruction
Internal relay (R)	0~511 byte
PLC alarm detection (A)	0~31 byte
Keep memory	
* Timer (T)	0~127 byte
* Meter (C)	0~127 byte
* Data table (D)	0~255 byte
* Keep relay (K)	0~63 byte
* Meter preset value data register (DC)	0~127 byte
* Timer preset value data register (DT)	0~127 byte
Subprogram (P)	0~99
Label (L)	0~99
I/O module (X)	0~63 byte
(Y)	0~47 byte

## 1.2 What is a sequence program

A sequence program is a program for sequence control of machine tools and other systems.

The program is converted into a format to enable CPU execute encoding and arithmetic processing, and stored into RAM. CPU reads out every instruction stored in the memory at a high-speed and executes the program by arithmetic operation

The sequence program is written firstly from ladder.

## 1.3 Assignment of interface specifications (step 1)

interface may be assigned after control object is determined and the relevant input/output signal points are counted.

For interface assignment, see *input/output interface signal table in Book 4 Connection of this manual*

## 1.4 Establishment of ladder diagram (step 2)

by GSK990MA ladder online edit function, use the ladder to express the machine control actions. As for the timer, counter, etc, which cannot be expressed with the relay symbol, express them with the designated functional instructions symbol.

The edited ladder should be stored and it should be converted into the corresponding PLC instruction namely so-called instruction table before running.

## 1.5 Sequence program debugging (step 3)

The sequence program can be debugged in two ways:

1) Debug by simulator

Instead of the machine, connect a simulator (consisting of lamps and switches). Switch ON/OFF stands for the input signal state of machine, lamp ON/OFF for the output signal state.

2) Actual operation debugging

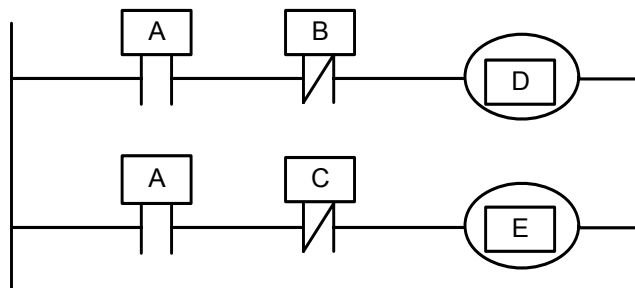
Debug sequence program through operating the machine. Do measures against the unexpected affairs before debugging.

## 2 Sequence Program

Since PLC sequence control handled by ladder online edit function and operates on principle difference from a general relay circuit, the sequence control method must be fully understood in order to design PLC sequence program.

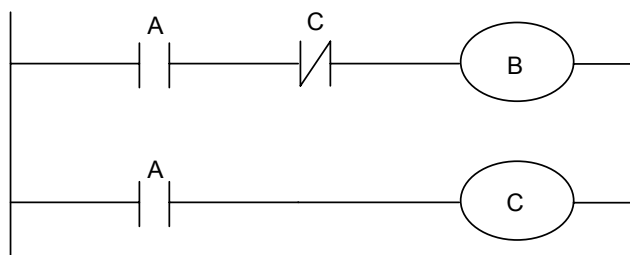
### 2.1 Execution process of sequence program

In general relay control circuit, each relay operates at approximately the same time, in the figure below for example, when relay A operates, the relay D and E operate at approximately the same time (when contacts B and C are off). In PLC sequence control, each relay of circuit operates sequentially. When relay A operates, relay D operates, then relay E operates (see the below figure). Thus each relay operates in sequence which can be written as a ladder diagram. (programmed sequence).



**Fig. 2.1(a) circuit example**

Fig.(b) and (c) illustrate operations varying from the relay circuit to PLC program.



**Fig. 2.1(b)**

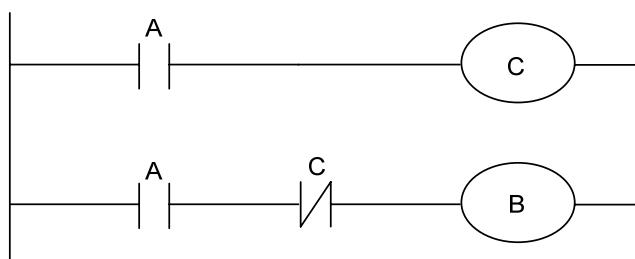


Fig. 2.1(c)

## (1) Relay circuit

In Fig. (A) and (B), the operations are the same. Turning on A turns on B and C. Turning on C turns off B.

## (2) PLC program

In Fig. (B), as in the relay circuit, turning on A turns on B and C, and after one cycle of the PLC sequence, turns off B. But in Fig.(C), turning on A turns on C, but does not turn on B.

## 2.2 Cycle execution

The PLC executes the ladder diagram from the beginning to the end . When the ladder diagram ends, the program starts over from the beginning. This is called cycle execution.

The execution time from the beginning to the end of the ladder diagram is called the cycle processing time. The shorter the process time is, the better the signal response becomes.

## 2.3 Priority of execution(1st level, and 2nd level)

GSK990MA PLC consists of two parts: 1<sup>st</sup> level sequence part, 2<sup>nd</sup> level sequence part. They have different execution period.

The 1<sup>st</sup> level sequence part operates every  $8 \times n$  ms, which can deal with the short pulse signal with high-speed response.

The 2<sup>nd</sup> level sequence part operates every  $8 \times n$  ms. Here n is a dividing number for the 2<sup>nd</sup> level sequence part. The 2<sup>nd</sup> level sequence part is divided into n parts, and every part is executed every 8ms.

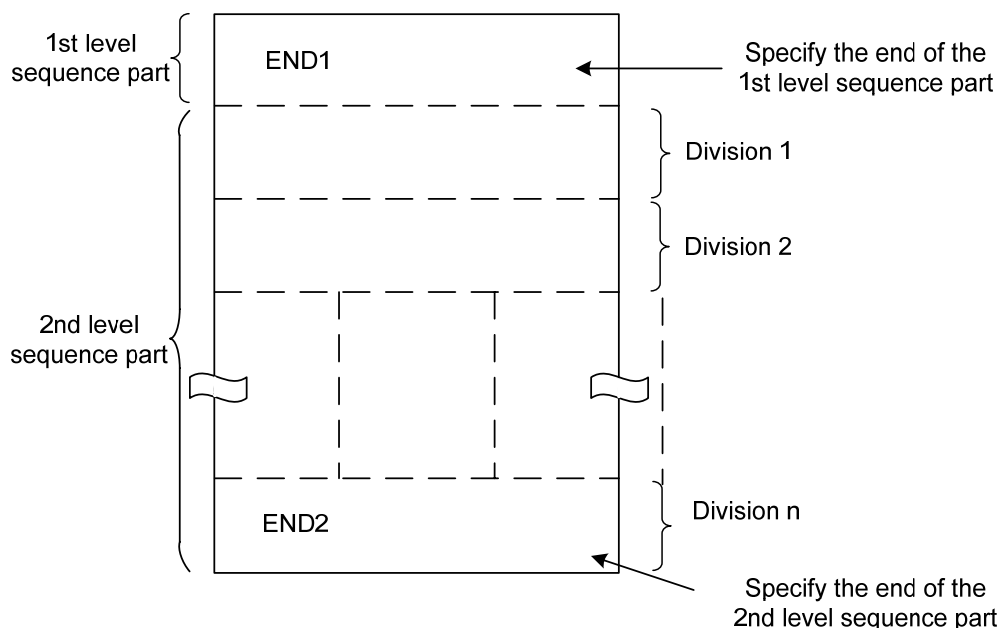


Fig. 2-3-1

990MA PLC is solely executed in PLC-AVR single chip, and the first 1ms of each 8ms is the communication time of CNC reading or writing PLC data. The fifth 1ms is the time that the PLC receives the system control signal (F、X) and uploads the control result data (G、Y parameter) to the external I/O interface (X、Y), except for the time responding the interruption to exchange the data, the PLC executes the ladder operation at the rest time.

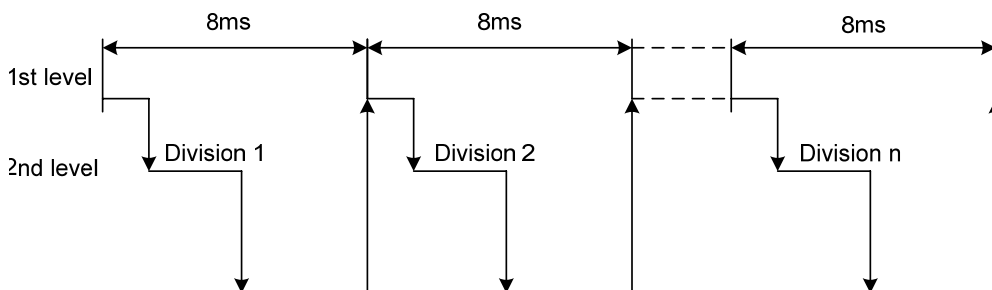


Fig. 2-3-2

After the last 2<sup>nd</sup> level sequence part (division n) is executed, the sequence program is executed again from the beginning. Thus, when the dividing number is n, the cycle of execution is  $8 \times n$  ms. The 1<sup>st</sup> level sequence operates every 8ms, and the 2<sup>nd</sup> level sequence every  $8 \times n$  ms. If the steps of the 1<sup>st</sup> level sequence is increased, the steps of the 2<sup>nd</sup> level sequence operating within 8ms becomes less, thereby increasing the dividing number and making the processing time longer. Therefore, it is desirable to program so as to reduce the 1<sup>st</sup> level sequence to a minimum.

## 2.4 Sequence program structure

With the conventional PLC, a ladder program is created sequentially. By employing a ladder language that allows structured programming, the following benefits are as following :

1. A program can be understood and developed easily
2. A program error can be found easily.
3. When an operation error occurs, the cause can be found easily.

Three major structured programming capabilities are supported:

1) Subprogram

A subprogram can consist of a ladder sequence as the processing unit.

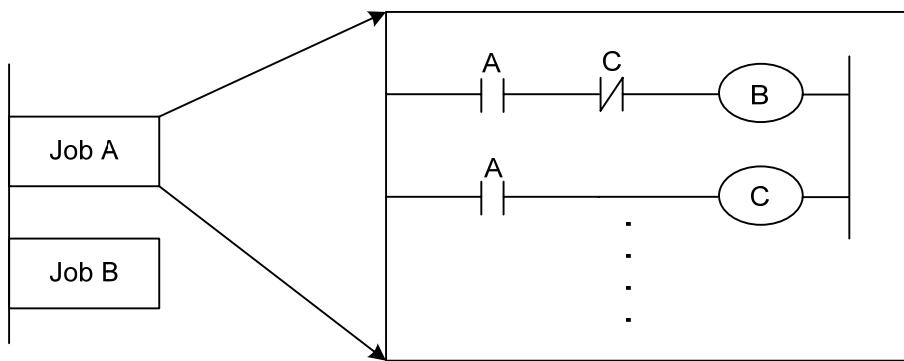


Fig. 2-4-1

2) Nesting

The Ladder subprograms can call the other ladder subprogram to execute the job.

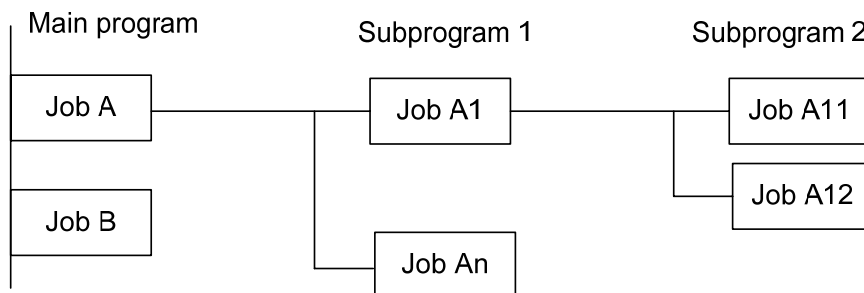


Fig. 2-4-2

3) Conditional branch

The main program loops and checks whether conditions are satisfied. If a condition is satisfied, the corresponding subprogram is executed. If the condition is not satisfied, the subprogram is skipped.

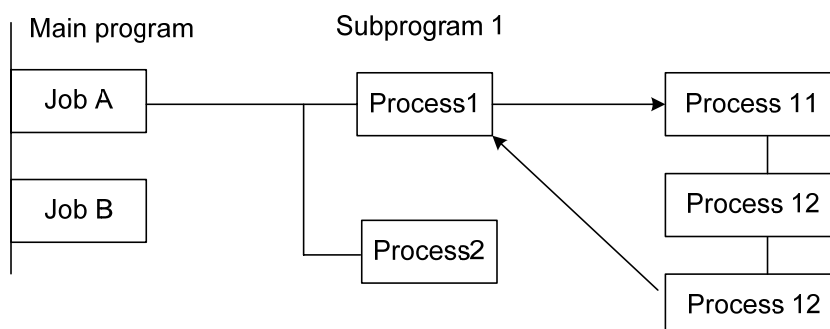


Fig. 2-4-3

## 2.5 Processing I/O ( input / output ) signals

Input signal processing:

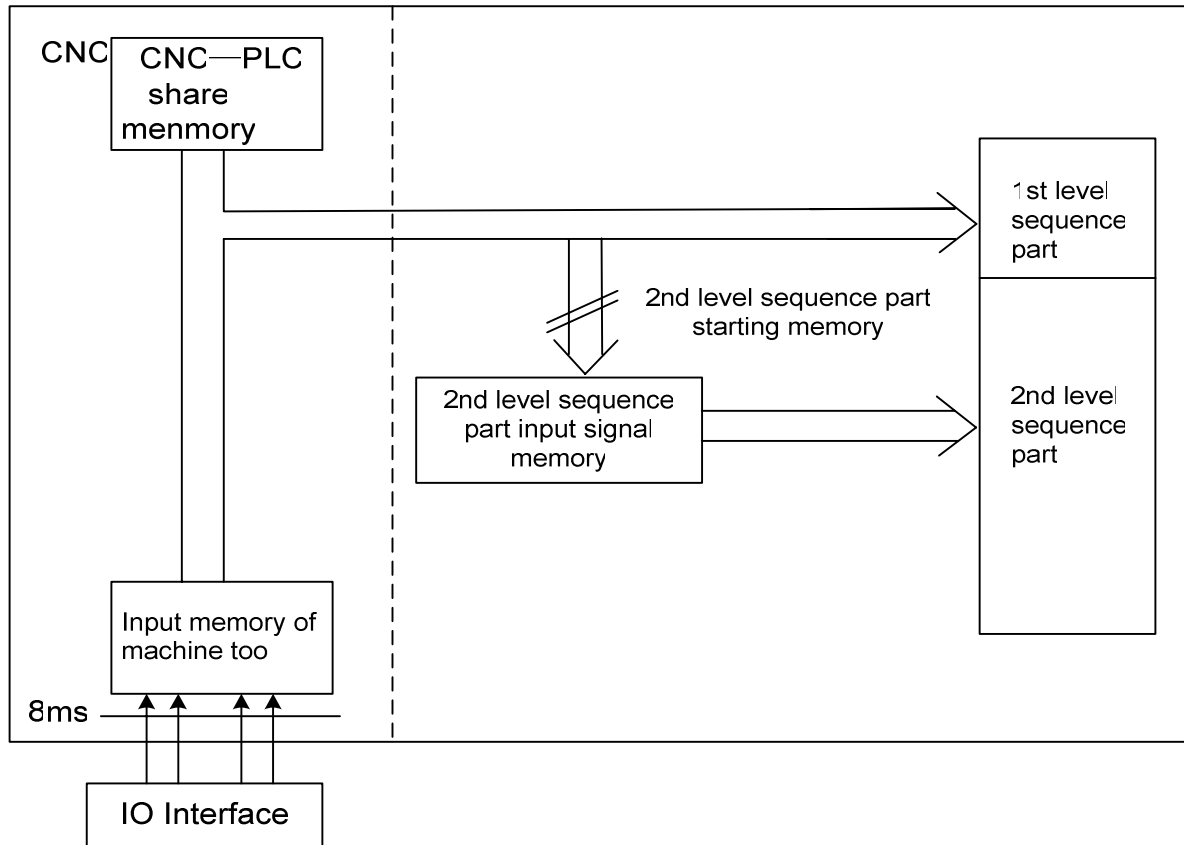


Fig. 2-5-1

Output signal processing:

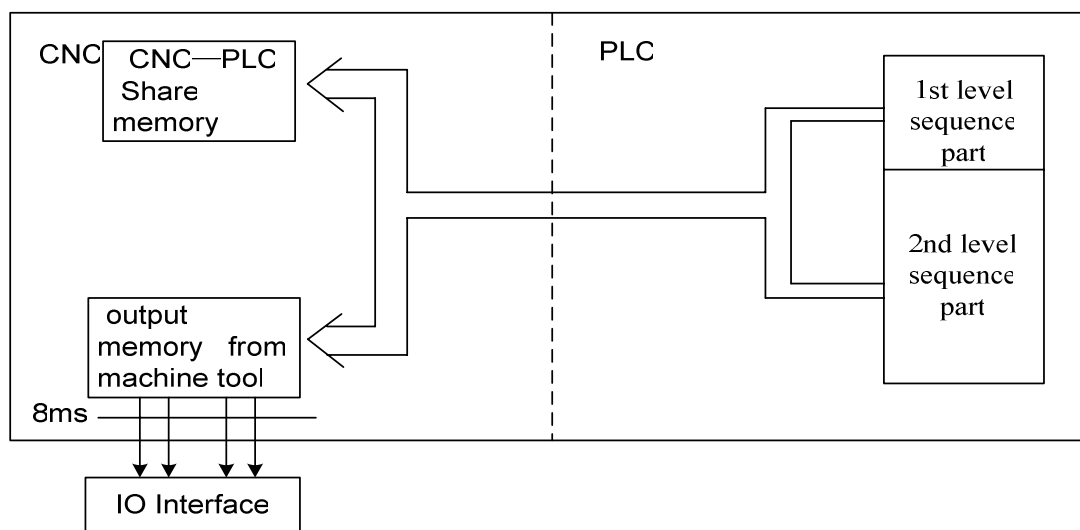


Fig. 2-5-2

### 2.5.1 Input signal processing

#### (1) NC input memory

The input signals from NC are loaded in memory of NC and are transferred to the PLC at intervals of 8ms. Since the 1<sup>st</sup> level sequence part directly refer to these signal and process operations.

#### (2) Input signal memory to machine tool

The input signal memory stores signals transferred from the machine tool at intervals of 8ms period. Since the 1<sup>st</sup> level sequence part directly refer to these signal and process operations.

#### (3) 2<sup>nd</sup> level input signal memory

The 2<sup>nd</sup> level input signal memory is also called 2<sup>nd</sup> level synchronous input signal memory. The stored signals are processed by the 2<sup>nd</sup> level sequence part. State of the signals set this memory synchronizes with that of 2<sup>nd</sup> level sequence part.

Input memory Signals from NC and machine tool are transferred to the 2<sup>nd</sup> level input signal memory only at the beginning of execution of the 2<sup>nd</sup> level sequence part. Therefore, the state of the 2<sup>nd</sup> level synchronous input signal memory does not change from the beginning to end of the execution of the 2<sup>nd</sup> level sequence part.

### 2.5.2 Output signal processing

#### (1) NC output memory

The output signals are transferred form the PLC to the NC output memory at intervals of 8ms.

#### (2) Output signals memory to machine tool

Signals stored in the machine tool output memory are transferred to the machine tool at intervals of 8ms.

Note:

The state of the NC input memory, NC output memory, input signals from machine, input/output memory signals to machine can be checked by using the PC self-diagnosis function. The self-diagnosis number specified is the address number used by the sequence program.

### 2.5.3 Difference state of signals between 1st level and 2nd level

The state of the same input signal may be different in the 1<sup>st</sup> level and 2<sup>nd</sup> level sequences.

Because they use different input memory. That is, at 1<sup>st</sup> level, processing is performed using input signal memory and at 2<sup>nd</sup> level, processing is performed using the 2<sup>nd</sup> level synchronous input signal memory. Therefore, it is possible for a 2<sup>nd</sup> level sequence execution at the worst, compared with a 1<sup>st</sup> level input signal.

This must be kept in mind when writing the sequence program.



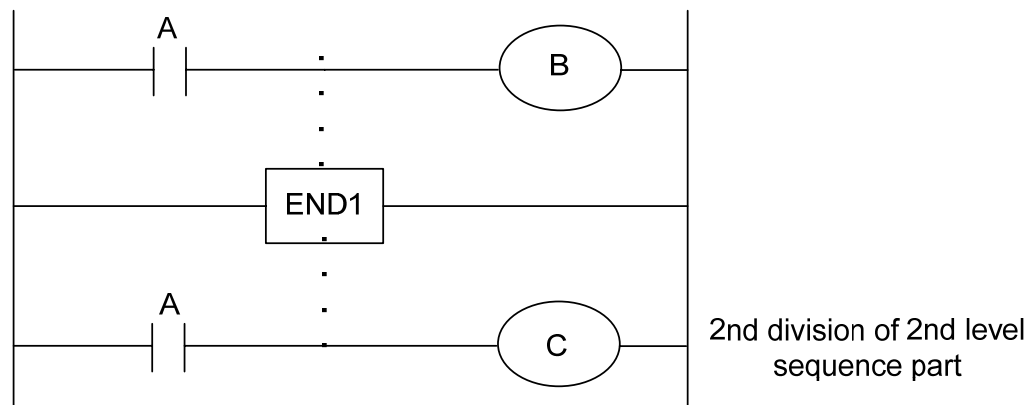


Fig. 2-5-3-1

When the processing is 1st 8ms, A=1, and B=1 after 1st sequence part is executed. At the same time, 2nd sequence part is started to execute, A=1 is stored to the 2nd sequence part and the 1st division of 2nd sequence part is executed.

When the processing is 2nd 8ms, A=0, and B=0 after 1st sequence part is executed. And then 2nd division of 2nd sequence part is executed, at this time, A is still 1. So C=1.

So, B and C are different.

## 2.6 Interlocking

Interlocking is important on sequence control safety.

Interlocking is necessary in the sequence control program. However, hard interlocking for the relay circuit should be applied in the machine strong power cabinet. This is because even interlocking is logically used in the sequence program (software), the interlock will not work when trouble occurs in the hardware used to execute the sequence program. Therefore, provide an interlock inside the machine tool magnetic cabinet to ensure the safety and to protect the machine from damage.

# 3 Address

An address shows a signal location. Addresses include input/output signals with respect to the machine, the input/output signals with respect to the CNC, the internal relays, the meters, the keep relays, and data table. Each address consists of an address number and a bit number. Its serial number regulations are as follows:

Address regulations:

The address comprises the address type, address number and the bit number in the format as shown below:

X    000    .    6  
Type    Address number    Bit number

Type: including X, Y, R, F, G K, A, D ,C, , T

Address number: decimal serial number stands for one byte.

Bit number: octal serial number, 0~7 stands for 0~7 bit of byte of front address number

990MA PLC address type is as follows:

**Table 3-1**

Address	Signal description	Length
X	Machine tool → PLC(64 byte)	INT8U
Y	PLC→ Machine tool (48 byte)	INT8U
F	CNC→PLC(64 byte)	INT8U
G	PLC→CNC(64 byte)	INT8U
R	Auxiliary relay(512 byte)	INT8U
D	Data register (0~255)	INT16U
DC	Counter preset data register	INT16U
C	Meter (0~127)	INT16U
A	PLC message request signal	INT8U
T	Timer (0~127)	INT16U
DT	Timer preset data register	INT16U
K	Keep relay (64 byte)	INT8U

INT8U data type is 8-bit character without signs, INT16U data type is 16-bit integer without signs.

## 3.1 Addresses from Machine tool to PLC (X)

X addresses of GSK990MA PLC are divided into two:

1. X addresses are assigned to IO input interface of XS43, XS44 and XS45.
2. X addresses are assigned to the input press keys on MDI panel.

### 3.1.1 Assignment of IO module X address

The addresses are from X0 to X5. Its type is INT8U, 48 types. They are assigned to three IO input interface of XS 43, XS44 and XS45.

The signal specification of X addresses can be customized by customer according to the actual operation. X addresses are used for machine (tool) connection and the ladder editing. For the initial definition of input address, see **Book 4 Connection of this manual**.

### 3.1.2 Assignment of MDI panel X address

The addresses are from X20 to X30, 11 bytes. They correspond to the press keys on MDI panel, and their signal definitions cannot be changed by user.

Corresponding relationship for addresses and press keys are as follows:

**Table 3-1-2-1**

Input key on operator panel	PLC address	Input key on operator panel	PLC address
Edit mode	X20.0	Feedrate positive override	X24.0
Auto mode	X20.1	Feedrate override cancel	X24.1
MDI mode	X20.2	Feedrate negative override	X24.2
Machine zero return mode	X20.3	Rapid	X24.7
Single step mode	X20.4	Rapid F0 / 0.001	X26.0
Manual mode	X20.5	Rapid 25% / 0.01	X26.1
MPG mode	X20.6	Rapid 50% / 0.1	X26.2
DNC mode	X20.7	Rapid 100% / 1	X26.3
Skip	X21.0	Manual feed axis +X	X27.0
Single block	X21.1	Manual feed axis +Y	X27.1
Dry run	X21.2	Manual feed axis +Z	X27.2
Miscellaneous(M, S, T) lock	X21.3	Manual feed axis +4TH	X27.3
Machine lock	X21.4	USER1	X27.4
Optional stop	X21.5	Manual feed axis -X	X28.0
Program restart	X21.6	Manual feed axis -Y	X28.1
Spindle CCW	X22.0	Manual feed axis -Z	X28.2
Spindle stop	X22.1	Manual feed axis -4TH	X28.3
Spindle CW	X22.2	USER2	X28.4
Spindle negative override	X22.3	USER3	X28.7
Spindle override cancel	X22.4	Spindle orientation	X29.0
Spindle positive override	X22.5	Tool magazine zero return	X29.1
Spindle jog	X22.6	Tool clamp/ release	X29.2
Lubrication	X23.0	Tool magazine CW	X29.3
Cooling	X23.1	Tool magazine CCW	X29.4
Chip removal	X23.2	tool infeed	X29.5

Cycle start	X23.6	tool retraction	X29.6
Feed hold	X23.7	Tool change manipulator	X29.7
		Overtravel release	X30.0

## 3.2 Address (Y) from PLC to machine tool

Y addresses of GSK990MA PLC are divided into two:

1. Y addresses are assigned to IO input interface of XS40, XS41 and XS42.
2. Y addresses are assigned to the indicators on MDI panel.

### 3.2.1 Assignment of IO module Y address

The addresses are from Y0 to Y5. Its type is INT8U, 48 types. They are assigned to three IO output interfaces of XS40, XS41 and XS42.

The signal specification of Y addresses can be customized by customer according to the actual operation. Y addresses are used for machine tool connection and the ladder editing. For the initial definition of input address, see *Appendix one Allocation and definition of PLC IO address, auxiliary relay and register for GSK990MA CNC system*

### 3.2.2 Assignment of IO module Y address

The addresses are from Y12 to Y19, 8 bytes. They correspond to the indicators on MDI panel, and their signal definitions cannot be changed by user.

Corresponding relationship for addresses and indicators are as follows:

**Table 3-2-2-1**

Output key on operator panel	PLC address	Output key on operator panel	PLC address
Edit key indicator	Y12.0	0.01/25% indicator	Y15.4
Auto key indicator	Y12.1	0.1/50% indicator	Y15.5
MDI key indicator	Y12.2	1/100% indicator	Y15.6
Machine zero return indicator	Y12.3	Spindle orientation indicator	Y15.7
Single step key indicator	Y12.4	Tool magazine zero return indicator	Y16.0
Manual key indicator	Y12.5	Tool magazine CCW indicator	Y16.1
MPG key indicator	Y12.6	Tool magazine CW indicator	Y16.2
DNC key indicator	Y12.7	Tool magazine infeed indicator	Y16.3
Spindle CCW indicator	Y13.0	Tool magazine retraction indicator	Y16.4
Spindle CW indicator	Y13.1	Tool magazine clamp indicator	Y16.5
Spindle override cancel indicator	Y13.2	Tool change manipulator indicator	Y16.6

X machine zero return indicator	Y13.3	USER3 (tool change position) indicator	Y16.7
Y machine zero return indicator	Y13.4	+X indicator	Y17.0
Z machine zero return indicator	Y13.5	+Y indicator	Y17.1
4TH machine zero indicator	Y13.6	+Z indicator	Y17.2
Optional stop indicator	Y13.7	+4TH indicator	Y17.3
Skip indicator	Y14.0	USER1 indicator	Y17.4
Single block indicator	Y14.1	-X indicator	Y18.0
Dry run indicator	Y14.2	-Y indicator	Y18.1
Miscellaneous(M, S, T) lock indicator	Y14.3	-A indicator	Y18.2

### 3.3 Address (G) from PLC to CNC

Addresses are from G0 to G63. Type: INT8U, 64 bytes.

Key signals on the operator panel

**Table 3-3-1**

Key signal on operator panel	PLC address	Key signal on operator panel	PLC address
Edit mode	G20.0	Rapid switch	G24.7
Auto mode	G20.1	Rapid F0	G25.0
MDI mode	G20.2	Rapid 25%	G25.1
Machine zero return mode	G20.3	Rapid 50%	G25.2
Single step mode	G20.4	Rapid 100%	G25.3
Manual mode	G20.5	Incremental step 0.001	G26.0
MPG mode	G20.6	Incremental step 0.01	G26.1
DNC mode	G20.7	Incremental step 0.1	G26.2
Skip	G21.0	Incremental step 1	G26.3
Single block	G21.1	MPG step 0.001	G26.4
Dry run	G21.2	MPG step 0.01	G26.5
Miscellaneous (M,S, T) lock	G21.3	MPG step 0.1	G26.6
Machine tool lock	G21.4	Manual feed axis +X	G27.0
Selection stop	G21.5	Manual feed axis +Y	G27.1
Program restart	G21.6	Manual feed axis +Z	G27.2
Spindle CW	G22.0	Manual feed axis +4TH	G27.3
Spindle stop	G22.1	Manual feed axis -X	G28.0
Spindle CCW	G22.2	Manual feed axis -Y	G28.1
Spindle negative override	G22.3	Manual feed axis -Z	G28.2
Spindle override cancel	G22.4	Manual feed axis -4TH	G28.3

Spindle positive override	G22.5	Spindle orientation	G29.0
Spindle jog	G22.6	Tool magazine zero return	G29.1

The bit signals of G63 bytes are internally used by the system, G63.0, G63.1 and G63.2 are separately the system internal response signals for M, S, T code completion.

### 3.4 Address (F) from CNC to PLC

Addresses are from F0 to F63. Type: INT8U, 64 bytes.

For signals, see Volume Function.

### 3.5 Internal relay address (R)

The address area is cleared to zero when the power is turned on. R510 and R511 are used by the system.

Type: INT8U, 512 bytes.

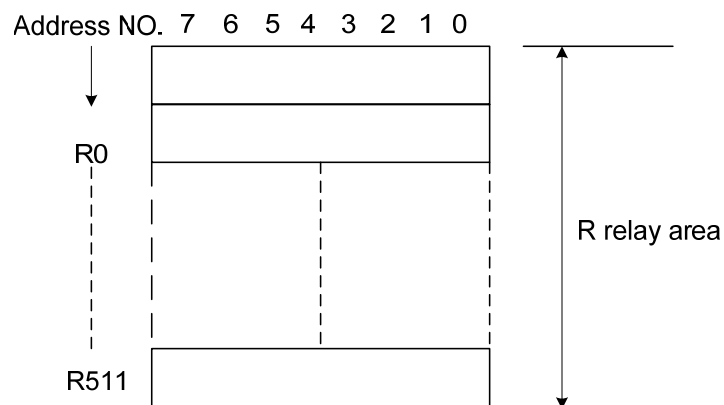


Fig. 3-5-1

#### System program management area:

R510

Address signal of R510.0 is set to 1 when PLC is started or restarted, which is used for the signal set by user in initialization. It is set to 0 after the ladder finishing the first pass.

R511 (timer for system)

The following four signals are used as system timer:

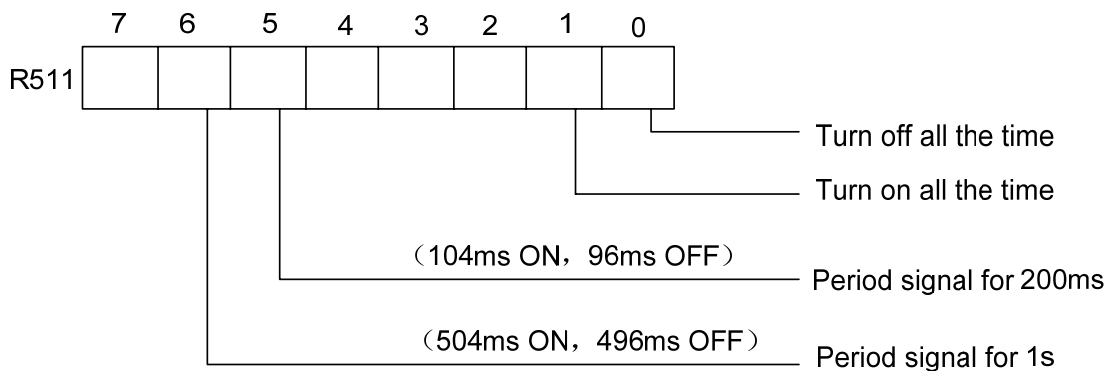


Fig. 3-5-2

### 3.6 Address of keep relay (K)

The area is used as keep relays and PLC parameters. Since this area is nonvolatile, the content of the memory do not disappear even when the power is turned off. K000~~K005 are used by the system, which are used for storing the PLC system parameter. It is convenient for the user to control the PLC by CNC.

Type: INT8U, with 64 bytes.

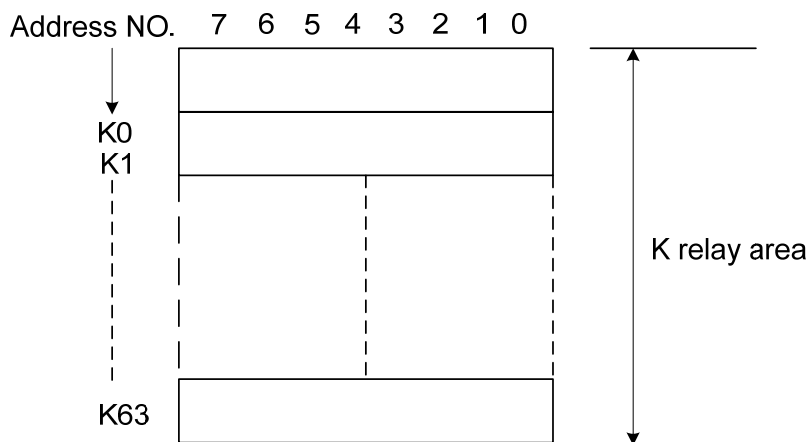


Fig. 3-6-1

**Note:** When PLC address K K005.2=1, PLC enters the debug mode, cancel all the external alarm signals and machine interlock signals, the tool change instruction can't be executed. The parameter can be modified under fully understanding it to avoid the machine damage or the person accident.

### 3.7 Addresses(A) for message selection displayed on CRT

The address area is cleared to zero when the power is turned on.

Type: INT8U, with 32 bytes.

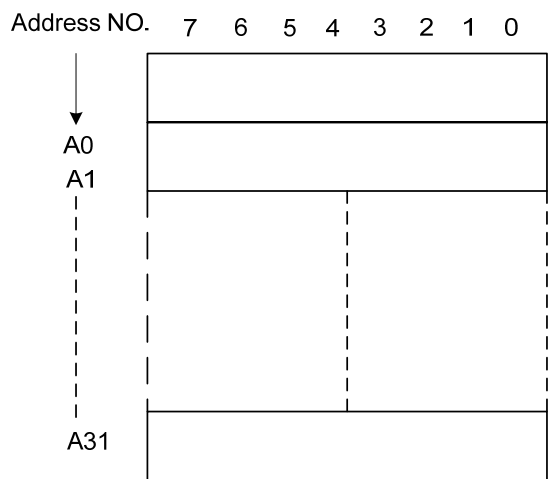


Fig. 3-7-1

**3.8 Address of meter (C)**

The area is used as storing current counting value in meter. The address area is cleared to zero when the power is turned on.  
Type: 128 addresses.

**3.9 Meter preset address(DC)**

The address area is used to store the meter preset value. Since this area is nonvolatile, the content of the memory do not disappear even when the power is turned off.  
Type: 128 addresses.

**3.10 Timer addresses (T)**

The area is used as storing current counting value in timer. The initial data is the preset value when the system is turned off. When preset value is 0, the current data is preset value.  
Type: 128 addresses.

**3.11 Addresses of timer preset value (DT)**

The address area is used as storing preset value. Since this area is nonvolatile, the content of the memory do not disappear even when the power is turned off.  
Type: 128 addresses.



### **3.12 Address of data table (D)**

The content of the memory do not disappear even when the power is turned off.

Type: 256 addresses.D240~247 are used by the system and cannot be defined by the user.

### **3.13 Label address (L)**

Label addresses are used to specify jump destination labels and LBL labels in JMPB instructions.

Range: 0~99

### **3.14 Subprogram numbers (P)**

Subprogram numbers are used to specify jump destination subprogram labels and SP instruction subprogram labels in CALL instruction.

Range: 0~99

# 4 PLC Basic Instruction

Designing a sequence program begins with creating a ladder diagram. The ladder diagram is written using relay contact symbols and functional instruction code. Logic written in the ladder diagram is entered as a sequence program in the Programmer. There are two sequence program entry methods. One is the entry method with the mnemonic language (PLC instructions such as RD, AND, OR but currently the system does not support it). The other is the relay symbols of the ladder diagram. When the relay symbol method is used, the ladder diagram format can be used and programming can be performed without understanding the PLC instruction format.

Actually, however, the sequence program entered by the relay symbol method is also internally converted into the instruction corresponding to the PLC instruction.

The basic instructions are often used when the sequence program is designed, and they execute one-bit operation.

GSK990MA basic instructions are as follows:

**Table 4-1**

Instruction	Function
RD	Shifts left the content by one bit in register and sets the state of a specified signal in ST0.
RD.NOT	Shifts left the content by one bit in register and sets the logic state of a specified signal in ST0.
WRT	Outputs the results of logic operation to a specified address.
WRT.NOT	Inverts the results of logical operations and output it to a specified address.
AND	Induces a logical product.
AND.NOT	Inverts the state of a specified signal and induces a logical product.
OR	Induces a logical sum.
OR.NOT	Inverts the state of a specified signal and induces a logical sum.
OR.STK	Sets the logical sum of ST0 and ST1, and shifts the stack register right by one bit.
AND.STK	Sets the logical product of ST0 and ST1, and shifts the stack register right by one bit.

## 4.1 RD, RD.NOT, WRT, WRT.NOT

### Instructions and functions

Table 4-1-1

Instruction	Function
RD	Shifts left the content by one bit in register and sets the state of a specified signal in ST0.
RD.NOT	Shifts left the content by one bit in register and sets the logic state of a specified signal in ST0.
WRT	Outputs the results of logic operation to a specified address.
WRT.NOT	Inverts the results of logical operations and output it to a specified address.

### Instruction explanation:

- WRT, WRT. NOT are the output relays, internal relay instructions. They cannot be used for input relay.
- The parallel WRT instruction can be continuously used many times, but double-coil output is disabled.

### Programming:

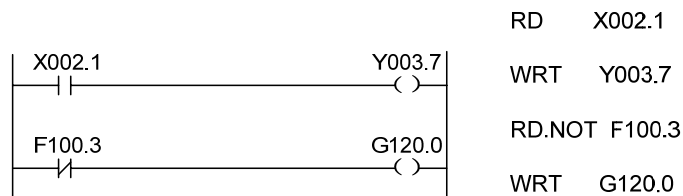


Fig. 4-1

## 4.2 AND, AND.NOT instructions

### Instructions and functions

Table 4-2-1

Instruction	Function
AND	Induces a logical product.
AND.NOT	Inverts the state of a specified signal and induces a logical product.

### Instruction explanation:

- AND, AND NOT can connect with one contact in serial. The serial contact numbers are not limited and they can be used many times.

### Programming

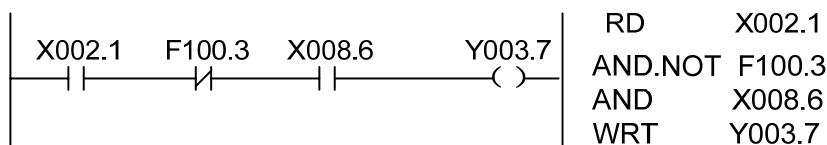


Fig. 4-2-1

## 4.3 OR, OR.NOT instructions

### Instructions and functions

Table 4-3-1

Instruction	Function
OR	Induces a logical sum.
OR.NOT	Inverts the state of a specified signal and induces a logical sum.

### Instruction specification:

- OR, OR\_NOT can connect with one contact in parallel.
- OR, OR.NOT begins from their step, which can connect with the RD, RD.NOT instructions step mentioned before in parallel.

### Programming:

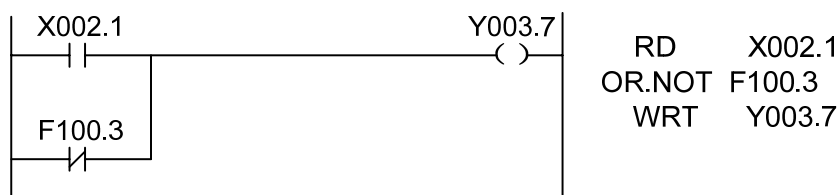


Fig. 4-3-1

## 4.4 OR. STK instruction

### Instruction and function:

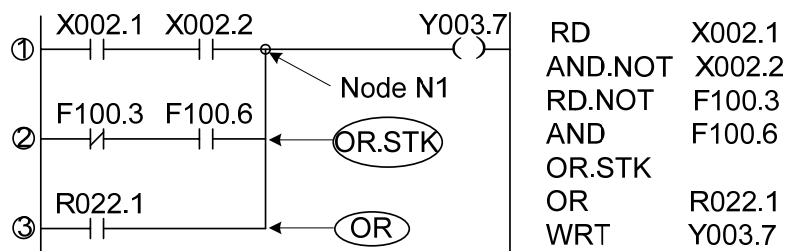
### Table 4-4-1

Instruction	Function
OR. STK	Sets the logical sum of ST0 and ST1, and shifts the stack register right by one bit.

**Instruction specification:**

- OR.STK a sole instruction without other address.

## Programming



**Fig. 4-4-1**

As the above figure, there are three branch circuit ①, ②, ③ from left bus to the node N1, among which ①, ② is circuit block in series; when there is the serial circuit block in the parallel from the bus to node or between nodes, the following branch end uses RD instruction except for the first branch. The branch ③ is not serial circuit block to use OR instruction.

OR.STK and AND.STK are instructions without operation components, indicating the OR, AND relationship between circuit blocks.

## 4.5 AND.STK instruction

## Instruction and function

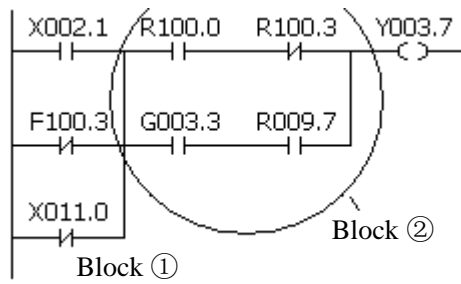
### Table 4-4-1

Instruction	Function
AND.STK	Sets the logical product of ST0 and ST1, and shifts the stack register right by one bit.

## Instruction specification

- When the branch loop (parallel loop block) is connected to the previous loop in series, use AND.STK instruction. The starting point of branch uses RD, RD.NOT instruction, after the parallel loop block ends, AND,STK instruction is connected to previous loop in series.
- AND.STK a sole instruction without other address.

Programming



```

RD  X002.1
OR.NOT  F100.3
OR.NOT  X011.0
RD  R100.0
AND.NOT  R100.3
RD  G003.3
AND  R009.7
OR.STK                                     ← (1)
AND.STK                                   ← (2)
WRT  Y003.7
    
```

Fig. 4-5-1

As the above figure and instruction list, (1) RD reports the circuit block in series is connected parallel (2)AND.STK reports the block ① and ② are connected in series.

## 5 PLC Functional instructions

If basic instructions such as controlling operations of machine tool are difficult to program, functional instructions are available to facilitate programming.

**Table 5-1 (990MA PLC functional instruction code)**

No.	Instruction	Processing
1	END1	End of a first-level ladder program
2	END2	End of a second-level ladder program
3	CALL	Calling subprogram
4	CALLU	
5	SP	Subprogram
6	SPE	End of subprogram
7	SET	Set
8	RST	Reset
9	JMPB	Label jump
10	LBL	Label
11	TMR	Timer
12	TMRB	Fixed Timer
13	TMRC	Timer
14	CTR	Binary meter
15	DEC	Binary decoding
16	COD	Binary code conversion
17	COM	Common line control
18	COME	End of common line control
19	ROT	Binary rotation control
20	SFT	Register shift
21	DIFU	Rising edge check
22	DIFD	Falling edge check
23	COMP	Binary comparison
24	COIN	Coincidence check
25	MOVN	Data transfer

26	MOVB	Transfer of an arbitrary number of bytes
27	MOVW	Transfer of two arbitrary number of bytes
28	XMOV	Binary Indexed data transfer
29	DSCH	Binary data search
30	ADD	Binary addition
31	SUB	Binary subtraction
32	ANDF	Functional AND
33	ORF	Functional OR
34	NOT	Logical Negation
35	EOR	Exclusive OR

## 5.1 END1 (1<sup>st</sup> level sequence program end)

### Function:

It must be specified once in a sequence program, either at the end of the 1<sup>st</sup> level sequence, or at the beginning of the 2<sup>nd</sup> level sequence when there is no 1<sup>st</sup> level sequence. It can write 500 steps at most.

### Format:

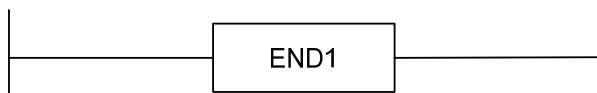


Fig. 5-1-1

## 5.2 END2 (2<sup>nd</sup> level sequence program end)

### Function

It is specified at the end of 2nd level sequence.

### Format:

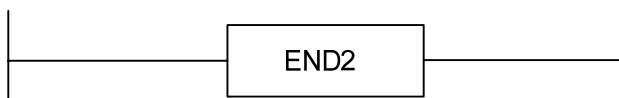


Fig. 5-2-1



### 5.3 CALL (call subprogram)

#### Function

Call a specified subprogram.

CALL has the following features and restrictions:

The subprogram may be nested up to 18 levels by other subprograms, but if a dead cycle is made by the closed loop calling, an alarm will be issued by system. Therefore to execute the data volume under the control, the allowable subprogram calling times are 100, and the subprogram calling in the 1st level is disabled. Alarm will be issued for the instructions or network between SP and END2, SPE and SP which can't be executed by system.

#### Format:

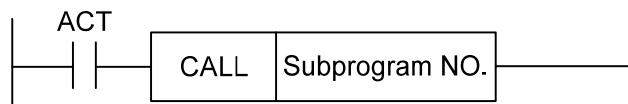


Fig. 5-3-1

#### Control condition:

ACT=0, execute the next instruction behind CALL.

ACT=1, call subprogram which number is specified.

#### Parameter:

Subprogram number.: specifies the number of a subprogram to be called.

Range: 0~99.

### 5.4 CALLU (unconditional subprogram call)

#### Function:

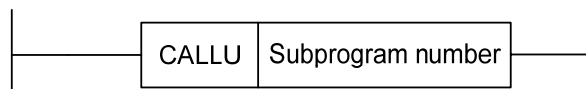
Call a specified subprogram without condition.

CALL has the following features and restrictions:

The subprogram may be nested up to 18 levels by other subprograms, but if a dead cycle is made by the closed loop calling, an alarm will be issued by system. Therefore to execute the data volume under the control, the allowable subprogram calling times are 100, and the subprogram calling in the 1st level is disabled. Alarm will be issued for the instructions or network between SP

and END2, SPE and SP which can't be executed by system

**Format:**



**Fig. 5-4-1**

**Parameter:**

Subprogram number: specifies the number of a subprogram to be called. Range:0~99.

## 5.5 SP (Subprogram)

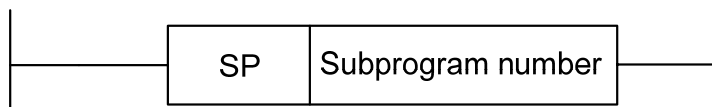
**Function:**

The SP functional instruction is used to create a subprogram. A subprogram number is specified as a subprogram name. SP is used with the SPE functional instruction to specify the subprogram range.

**Note:**

1. A subprogram must be written after END2.
2. Another subprogram cannot be nested into a subprogram.

**Format:**



**Fig. 5-5-1**

**Parameter:**

Subprogram number: specifies the subprogram number of a subprogram to be called.

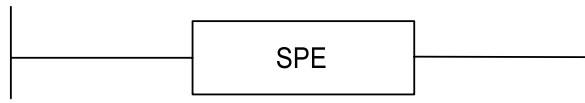
Range: 0~99.

## 5.6 SPE (subprogram end)

**Function:**

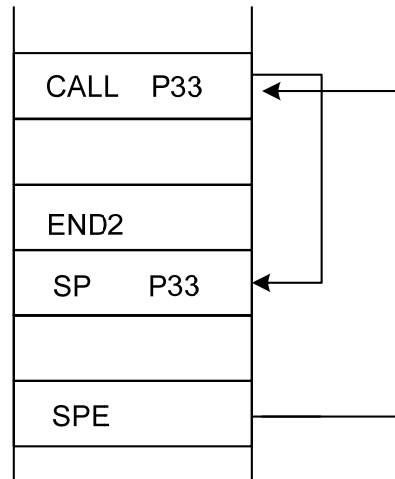
- \* it is used to specify the range of subprogram when SPE is used with the S P.
- \* the control will return to the main program which called the subprogram when the instruction is executed.
- \* the subprogram must be written after END2.

**Format:**



**Fig. 5-6-1**

**Example:**



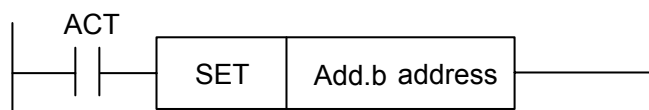
**Fig. 5-6-1**

## 5.7 SET (set)

**Function:**

Set to 1 for the specified address.

**Format:**



**Fig. 5-7-1**

**Control condition:**

ACT=0, add.b keep invariably.

ACT=1, add.b set to1.

**Parameter:**

Add.b: set element address bit can be the output coil, Add= Y, G, R, K, A.

## 5.8 RST (reset)

### Function:

Set to 0 for the specified address.

### Format:

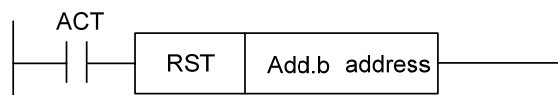


Fig.5-8-1

### Control condition:

ACT=0, add.b keep invariably.

ACT=1, add.b set to1.

### Parameter:

Add.b: reset element address bit can be the output coil, Add= Y, G, R, K, A.

## 5.9 JMPB (label jump)

### Function:

The JUMP functional instruction transfer control to a Ladder immediately after the lable set in a Ladder program.

JMPB has the following features and restrictions:

- \* More than one jump instruction can be coded for the same label.
- \* Jump between 1<sup>st</sup> level program and 2<sup>nd</sup> level is forbidden.
- \* Jump between subprograms is forbidden.
- \* Jump back is permitted, but the user should handle the endless loop may be caused by it .
- \* Jump between main program and subprogram is forbidden.

### Format:

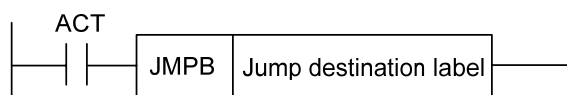


Fig. 5-9-1

### Control conditions:

ACT=0: The next instruction after the JMPB instruction is executed.

ACT=1: Control is transferred to the Ladder immediately after the specified label.

**Parameter:**

Lx: specifies the label of the jump destination. A value from 0 to 99 can be specified.

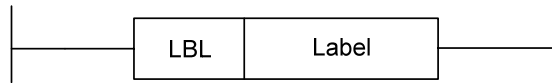
## 5.10 LBL (Label)

**Function:**

The LBL functional instruction specifies a label in a ladder program. It specifies the jump destination for JMPB functional instruction.

**Note:** one Lx label is only specified one time with LBL. Otherwise, the system alarms.

**Format:**

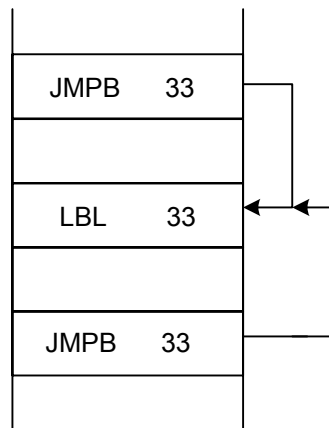


**Fig. 5-10-1**

**Parameter:**

Lx: specifies the label of the jump destination. Label number range: 0~99

**Example:**



**Fig. 5-10-2**

## 5.11 TMR (timer)

### Function:

This is an on-delay timer.

### Format:

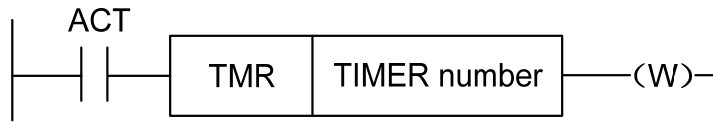


Fig. 5-11-1

### Control condition:

ACT=0: turns off the timer relay.

ACT=1: initiates the timer.

### Detailed functions:

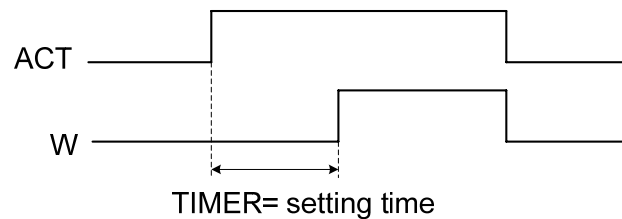


Fig. 5-11-2

### Parameter:

TIMER : timer serial number is named with xxx which are numbers (0~127).

### Output:

W : output coil. W=1 when the output reaches the preset value. W=0 when the output does not reach the preset value.

### Note:

Timer is executed each 8ms, take ms as its setting unit, and 8ms is taken as the execution base. Those time less than 8ms are taken as 8ms. i.e. it is set for 54ms,  $54=6*8+6$ , 2ms is needed to be added, so the actual execution time is 56ms.

The time of the timer is set under the 【TMR】 of 【PLCPAR】 in PRG interface.

The system will automatically detect the range of the sequence number of the timer, alarm will be issued for those duplicate or beyond range sequence numbers.

## 5.12 TMRB (fixed timer)

### Function:

This is an on-delay timer.

### Format:

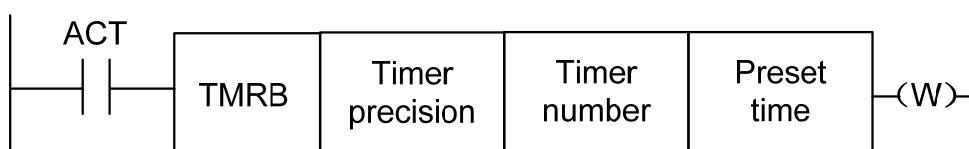


Fig. 5-12-1

### Control condition:

ACT=0: turns off the timer.

ACT=1: initiates the timer.

### Detailed functions:

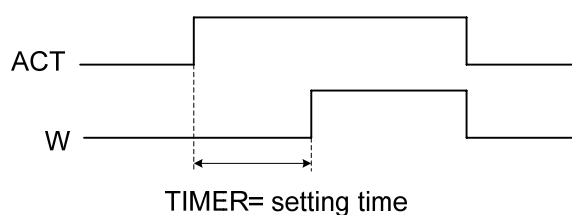


Fig. 5-12-2

### Parameter:

TIMER : fixed timer serial number is named with xxx which are numbers (0~127).

Table 5-12-1 (timer precision)

Timer precision	Setting number	Setting time range	Error range
8msec	0	From 8msec to 524.280sec	From 0 to scan period for 1st level program
48msec	1	From 48msec to 31.456 min	From 0 to scan period for 1st level program
1sec	2	From 1sec to 546 min	From 0 to scan period for 1st level program
10sec	3	From 10sec to 182 h	From 0 to scan period for 1st level program
1min	4	From 1min to 1092 h	From 0 to 1sec
1msec	5	From 1msec to 65.4sec	From 0 to scan period for 1st level program

**Preset time:**

Fixed timer timing setting, range (0~65535).

**Output:**

W : output coil. W=1 when the output reaches the preset value. W=0 when the output does not reach the preset value.

**Note:**

The system will automatically detect the range of the sequence number of the timer, alarm will be issued for those duplicate or beyond range sequence numbers.

The preset time of the timer will be fixed in ROM with the ladder, therefore, the time can be changed only by altering the ladder.

## 5.13 TMRC (timer)

**Function:**

This is an on-delay timer.

**Format:**

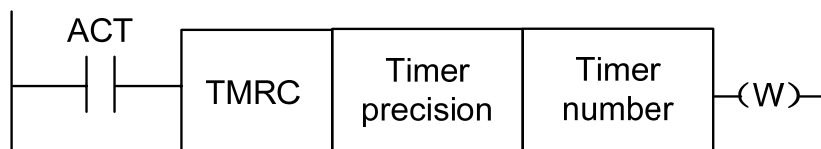


Fig. 5-13-1

**Control condition:**

ACT=0: turns off the timer.

ACT=1: initiates the timer.

**Detailed functions:**

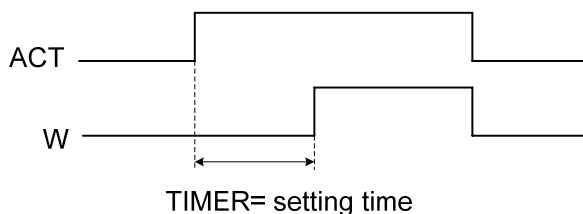


Fig. 5-13-2



**Parameter:**

TIMER : timer serial number is named with xxx which are numbers (0~127).

**Table 5-12-1 (timer precision)**

Timer precision	Setting number	Setting time range	Error range
8msec	0	From 8msec to 524.280sec	From 0 to scan period for 1st level program
48msec	1	From 48msec to 31.456 min	From 0 to scan period for 1st level program
1sec	2	From 1sec to 546 min	From 0 to scan period for 1st level program
10sec	3	From 10sec to 182 h	From 0 to scan period for 1st level program
1min	4	From 1min to 1092 h	From 0 to 1sec
1msec	5	From 1msec to 65.4sec	From 0 to scan period for 1st level program

**Output:**

W : output coil. W=1 when the output reaches the preset value. W=0 when the output does not reach the preset value.

**Note:**

The time can be set via 【TMR】 in 【PLCPAR】 of PLC windows.

TMRC timer and TMR timer use a same address, therefore, their sequence number cannot be the same.

The system will automatically detect the range of the sequence number of the timer, alarm will be issued for those duplicate or beyond range sequence numbers.

## 5.14 CTR (binary counter)

**Function:**

The data in the counter are binary and their functions are as follows:

1) Preset counter

Preset the count. It outputs a signal when the preset count is reached.

2) Ring counter

If the counter reaches the preset value, input the count signal to restore the initial value and recount.

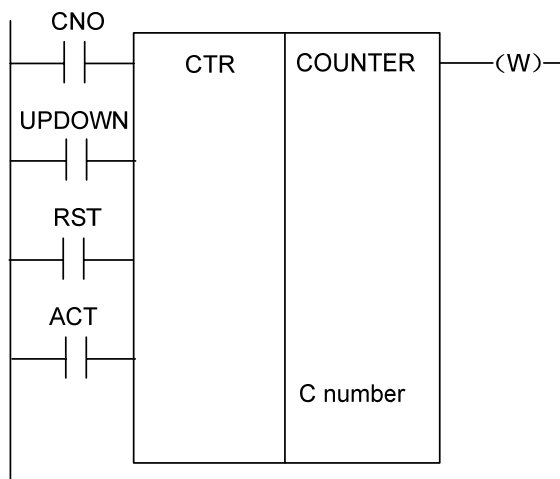
3) Up/down counter

The count can be either up or down.

4) Selection of initial value

Its initial value is 0 or 1.

**Format:**



**Fig. 5-14-1**

**Control condition:**

Specifies the initial value(CN0):

CN0=0: begins the value of the counter with 0.

CN0=1 begins the value of the counter with 1.

Specify up or down counter (UPDOWN):

UPDOWN=1: Up counter

UPDOWN=0: Down counter

Reset (RST):

RST=0: release reset.

RST=1: enable reset. When W=0, the integrated value is reset to the initial value.

RST is set to 1 only when reset is required.

Count signal(ACT):

ACT=1: count is made by catching the rise of ACT.

ACT=0: counter does not operate. W does not change.

**Parameter:**

METER: specifies the counter serial number with xxx which are numbers (0~127).

**Output:**

W: coil output. W=1 when the counter reaches the preset value.

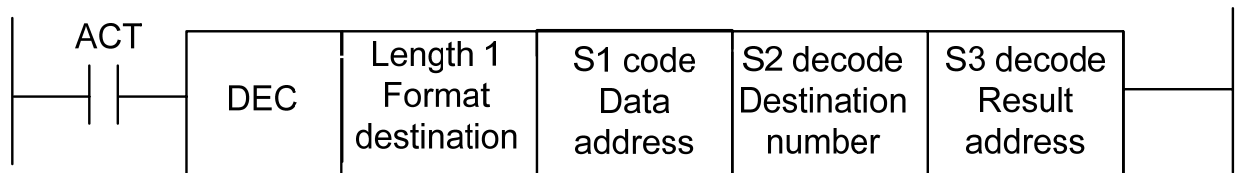
Note: The system will automatically detect the range of the sequence number of the counter, alarm will be issued for those duplicate or beyond range sequence numbers.

It needs to reset again after setting the preset value again to ensure the instruction reliability.

**5.15 DEC (binary decode)****Function:**

DEC can decode binary code data. Outputs 1 when the eight-digit BCD signal is equal to a specified number, and 0 when not.

It is mainly used to decode M or T function.

**Format:**

**Fig. 5-15-1**

**Control condition:**

ACT=0 : resets all the output data bit.

ACT=1 : decode data. Results of processing is set in the output data address.

**Parameter:**

length : Set the size of code data to the 1<sup>st</sup> digit of the parameter.

0001: code data is in binary format of 1 byte length.

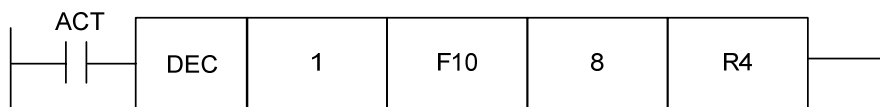
0002: code data is in binary format of 2 byte length.

S1 : code data address. Specifies an address at which code data is stored.

S2 : number specification decode designation. Specifies the first of the 8 continuous numbers to be decoded.

S3 : decode result address. Specifies an address where the decoded result shall be output. A one-byte area is necessary in the memory for the output.

Example:



When ACT=1 and F10=8, R4=0000,0001;

When ACT=1 and F10=9, R4=0000,0010;

.....

When ACT=1 and F10=15, R4=1000,0000;

## 5.16 COD (binary code conversion)

Function:

COD instruction automatically creates a table with corresponding size used for user inputting conversion table data when it inputs the data capacity. Each table has 10 lattices and if it is not divided by 10, count the lattices by its quotient adding 1, but its capacity data does not change and the address of table number will not be displayed.

Format:

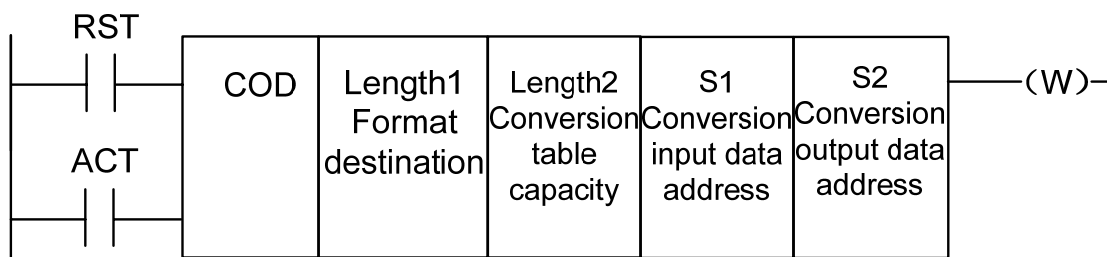


Fig. 5-16-1

S1	0	1	2	.....	9
S2	XXX	YYY	AAA	.....	.....
S1	10	11	12	.....	N-1
S2	.....	.....	.....	.....	UUU

Fig. 5-16-2

Control conditions:

Reset (RST):

RST=0: do not reset.

RST=1: reset error output W.

Activate instruction (ACT):

ACT=0 : do not execute COD.

ACT=1 : execute COD. Take value of "Conversion input data address(S1)" as the table number of conversion table, take out of 1 conversion data which corresponds to the table number from the conversion table, output the output address used for the conversion data (S2).

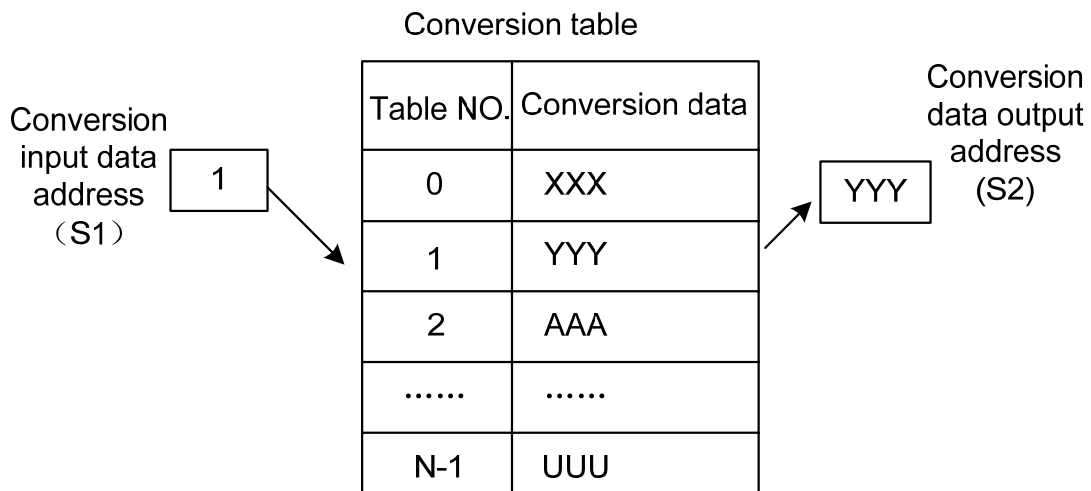


Fig. 5-16-2

#### Parameter:

Length1 : Designates binary numerical size in the conversion table.

1: Numerical data is binary 1-byte data.

2: Numerical data is binary 2-byte data.

length2 : Number of conversion table data. 100 data can be made at most. 100 bytes when designating 1 byte format, and 100 words when 2 byte format. All number is at most 512 bytes in COD conversion table.

S1 : Data in the conversion data table can be taken out by specifying the table number. The address specifying the table number is called conversion input data address, and 1-byte memory is required from the specified address.

S2 : Conversion data output address. Memory of the byte length specified in the format designation is necessary from the specified address.

#### Output:

If there are any abnormality when executing the CODB instruction, W=1, and error will be output.

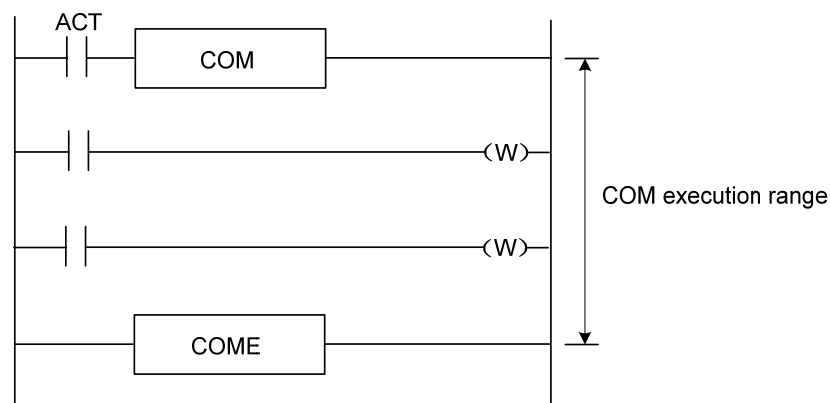
**Note:** Size of the conversion data table is maximum 256. This conversion data table is programmed between the parameter conversion data output address of this instruction and the error output (W).

### 5.17 COM (common line control)

**Function:**

This function can be used for specifying the number of coil only on the PLC-SB/SC. If the common line end instruction is not specified, the system will alarm.

**Format:**



**Control condition:**

ACT=0: The specified number of coils or the coils within the region specified are unconditionally turned off (W=0).

ACT=1: No processing is performed.

**Note:**

1. In the range specified with a COM instruction, no additional COM instruction can be specified.
2. The coil for WRT.NOT in the range specified with a COM instruction is singly set to 1 when COM ACT=0.
3. do not use the function instructions such as JMPB、END1、END2、CALL、CALLU、LBL、SP、SPE、COM、COME between COM and COME, otherwise, the system will alarm.

### 5.18 COME (common line control end)

**Function:**

The instruction specifies the control range of the common line control instruction (COM). It cannot be used alone. It must be used together with the COM instruction.

Format:



Fig. 5-18-1

## 5.19 ROT (Binary rotation control)

Function:

Controls rotors, such as the toolpost, rotary table, etc., and is used for the following functions.

1. Selection of the rotation direction via the shorter path.
2. Calculation of the number of steps between the current position and the goal position; calculation of the position on position before the goal to the number of steps up to one position before the goal.
3. Calculation of the position one position before the goal or of the number of steps up to one position before the goal.

Format:

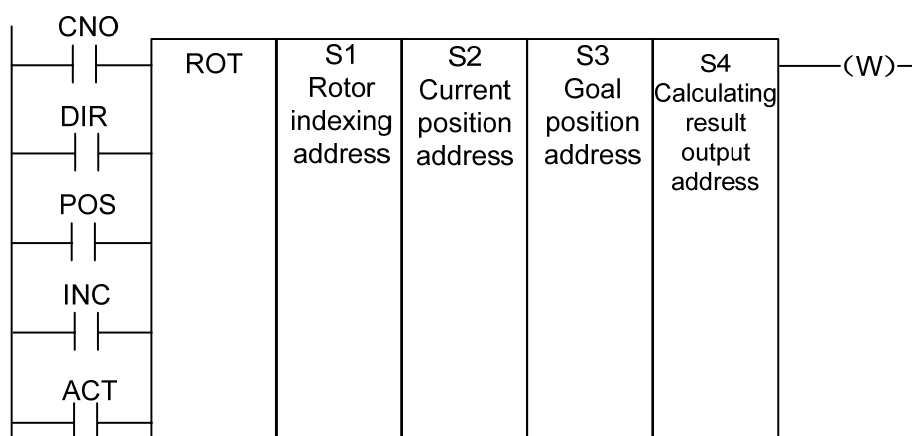


Fig. 5-19-1

Control conditions:

Specify the starting number of the rotor(CN0):

CNO=0: begins the number of the position of the rotor with 0.

CNO=1: begins the number of the position of the rotor with 1.

Select the rotation direction via the shorter path or not: (DIR):

DIR=0: no direction is selected. The direction of rotation is only forward.

DIR=1: selected. The direction of rotation is forward/backward.

Specify the operation conditions (POS):

POS=0: calculate the goal position.

POS=1: calculates the position one position before the goal position.

Specify the position or the number of steps (INC):

INC=0: calculates the number of the position. If the position one position before the goal position is to be calculated, specify INC=0 and POS=1.

INC=1: calculates the number of steps. If the difference between the current position and the goal position is to be calculated, specify INC=1 and POS=0.

Execution instruction (ACT):

ACT= 0: the ROT instruction is not executed. W does not change.

ACT= 1: executed. Normally, set ACT=0. If the operation results are required, set ACT=1.

**Parameter:**

- S1 : specify the rotor indexing number.
- S2 : specify the address storing the current position.
- S3 : specify the address storing the goal position(or instruction value), for example the address storing the CNC output T code.
- S4 : calculate the number of steps for the rotor to rotate, the number of steps up to the position one position before, or the position before the goal. When the calculating result is to be used, always check that ACT=1.

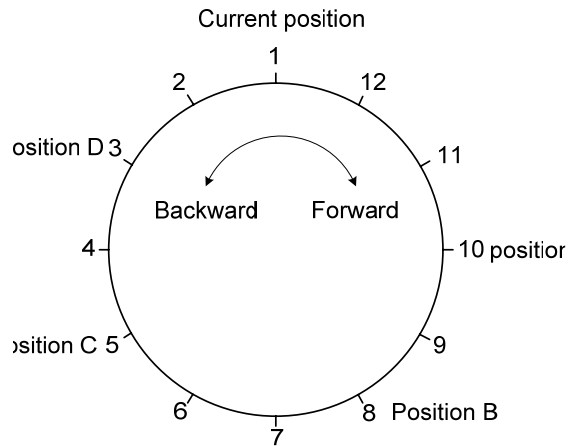
**Output:**

W: The direction of rotation for control of rotation via the shorter path is output to W. When W=0, the direction is forward (FOR) , when 1, reverse (REV). The definition of FOR and REV is shown in the following figure. If the number given to the rotor is ascending, the rotation is FOR; if descending, REV. The address of W can be determined arbitrarily. When, however, the result of W is to be used, always check that ACT=1.

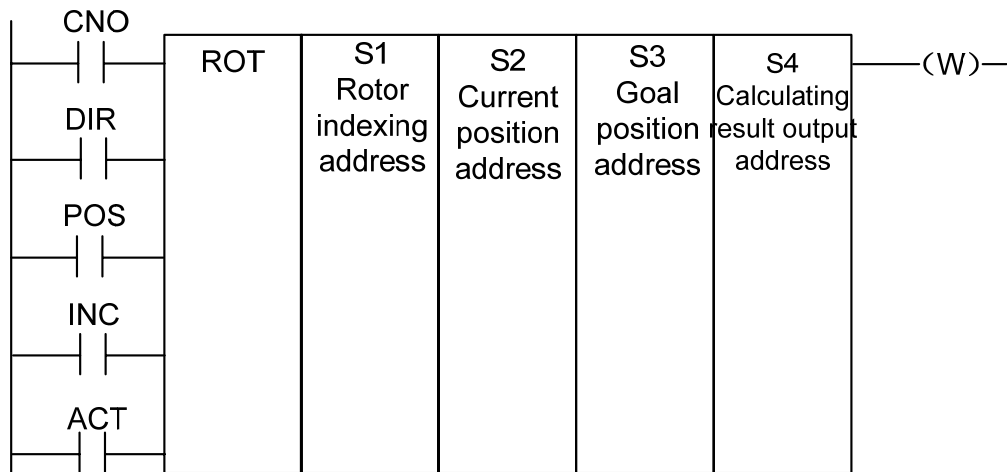
**Example:**

Rotor rotation direction:





**Fig. 5-19-2**



**Fig. 5-19-3**

Perform the short path rotation, and calculate the position number of previous one position of goal position.

Current position number S2=1, position number of rotation graduation S1=12, CNO=1, DIR=1

POS=1, INC=0:

When S3=10 goal position is A, and ACT=1, S4=11, W=1.

When S3=8 goal position is B, and ACT=1, S4=9, W=1.

When S3=5 goal position is C, and ACT=1, S4=4, W=0.

When S3=3 goal position is D, and ACT=1, S4=2, W=0.

## 5.20 SFT (shift register)

### Function:

This instruction can each time shift a byte data (8 bits) by a bit number set by a Parameter, For the circular shifting, each overflowing "1" will be added reversely, i.e. If the highest bit "1" is overflowed by the left shifting, the lowest bit will be filled by "1", vice versa.

### Format:

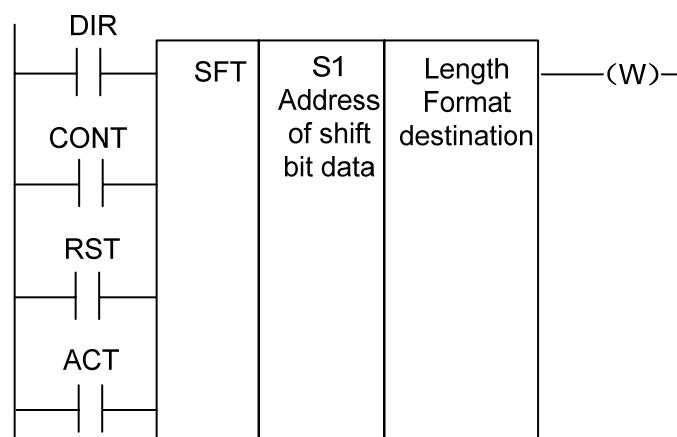


Fig. 5-20-1

### Control conditions:

Shift direction specification (DIR)

DIR=0: Left shift

DIR=1: Right shift

Condition specification (CONT)

CONT=0: do not cycle shift

CONT=1: cycle shift

Reset (RST)

The shifted out data(W=1) is reset (W=0).

RST=0: W is not reset.

RST=1: W is reset (W=0).

Actuation condition (ACT)

ACT=0: do not execute SFT instruction.

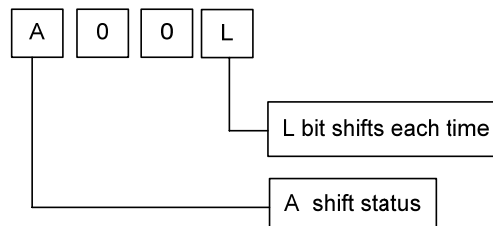
ACT=1: execute shifting. If shifting one bit only, set ACT to 0 after finishing the

execution.

**Parameters:**

**S1** : sets shift data addresses. These designated addresses require a continuous 1-byte memory for shift data.

**Length** : a 4-bit number, and its definition is as follows:



**Fig. 5-20-2**

**L** : range: 0~8.

**A** : bit parameter. A=0: When ACT=1 is shifting, the shift period is one bit.

A=1: ACT is taken as pulse signal, it is 1 from 0, shift one bit.

**Output:**

**W** : W=0: "1" was not shifted out because of the shift operation.

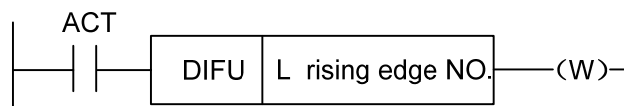
W=1: "1" was shifted out because of the shift operation.

## 5.21 DIFU (rising edge check)

**Function:**

The DIFU instruction sets the output signal to 1 for one scanning cycle on a rising edge of the input signal.

**Format:**



**Fig. 5-21-1**

**Control condition:**

**Input signal:** On a rising edge (0→1) of the input signal, the output signal is set to 1.

**Output signal:** The output signal level remains at 1 for one scanning cycle of the ladder when this functional instruction is operating.

**Parameter:**

L : rising edge number, range 0~255. If another DIFU instruction or DIFD instruction in the ladder uses the same number, the system will alarm.

**Operation:**

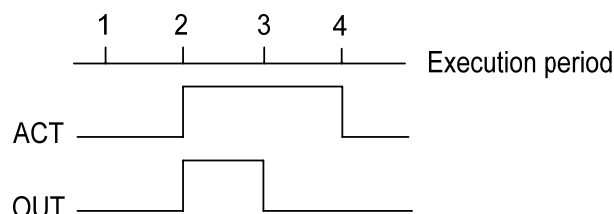


Fig. 5-21-1

## 5.22 DIFD (falling edge check)

**Function:**

The DIFD instruction set the output signal to 1 for one scanning period on a falling edge of the input signal.

**Format:**

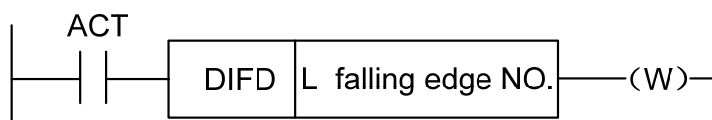


Fig. 5-22-1

**Control conditions:**

Input signal: on a falling edge (1→0) of the input signal, the output signal is set to 1.

Output signal: the output signal level remains at 1 for one scanning period of the ladder when this functional instruction is operating.

**Parameter:**

L : rising edge number, range 0~255. If another DIFU instruction or DIFD instruction in the ladder uses the same number, the system will alarm.

**Operation:**

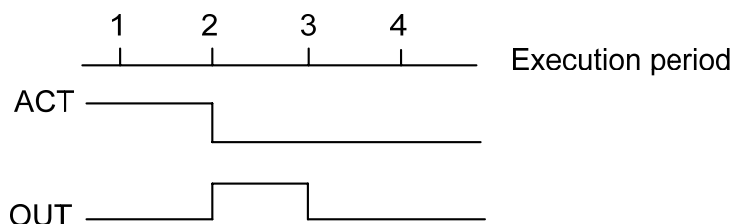


Fig. 5-22-2

The system will check the sequence number of the falling edge automatically, when the number exceeds the range or the number is duplicated, alarm occurs.

## 5.23 COMP (binary comparison)

### Function:

Compares two binary values. It needs to specify enough byte to store the input data and the comparison data in the memory.

### Format:

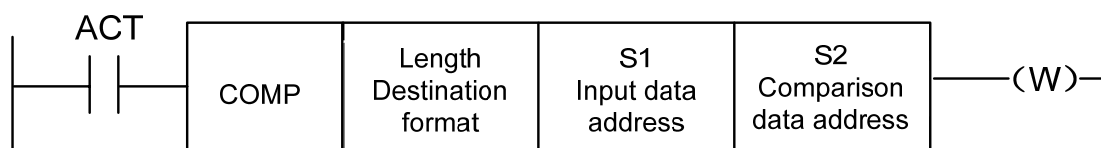


Fig. 5-23-1

### Control conditions:

ACT=0: The COMP instruction is not executed. W does not change.

ACT=1: The COMP instruction is executed.

### Parameter:

Length: specification format( constant or address) and data length(1 or 2 bytes) for the input data.

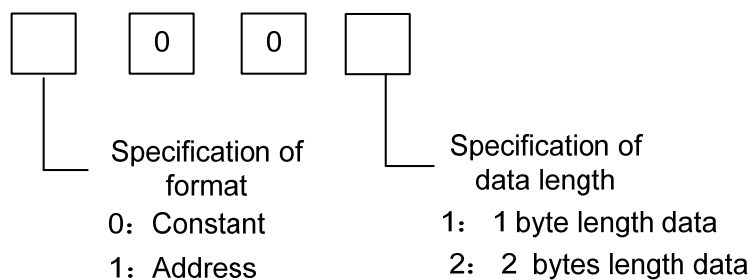


Fig. 5-23-2

S1, S2: content of comparison source 1 and comparison source 2. It can be constant

and also be address number.

Address number: R, X, Y, F, G, K, A, D, C.

#### Output:

W=0: input data > comparison data

W=1: input data ≤ comparison data

## 5.24 COIN (coincidence check)

#### Function:

Checks whether the input value and comparison value coincide.

#### Format:

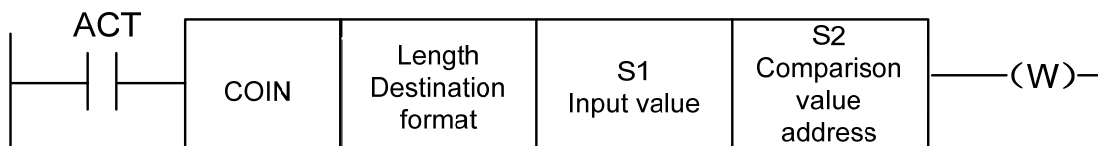


Fig. 5-24-1

#### Control conditions:

ACT=0, the COIN instruction is not executed. W does not change.

ACT=1, the COIN instruction is executed.

#### Parameter:

Length: specification format ( constant or address) and data length(1 or 2 bytes) for the input data.

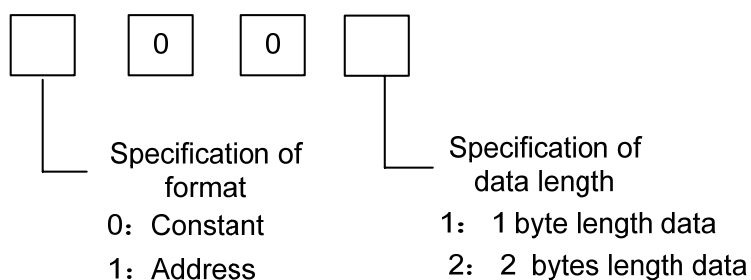


Fig. 5-24-2

S1 : The input data can be specified by either a constant or an address storing it.

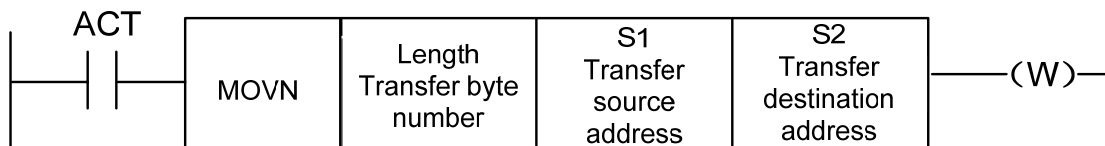
S2 : Address storing of comparison data

**Output:**

W : W=0: Input value ≠ comparison value  
 W=1: Input value = comparison value

**5.25 MOVN (transfer of data)****Function:**

It transfers data from source address or a specified binary data to a destination address.

**Format:****Fig. 5-25-1****Control condition:**

ACT=0: No data is transferred.

ACT=1: The byte of specified number is transferred.

**Parameter:**

Length : The byte number or data number to be transferred.

S1 : Source address or constant.

Selecting transfer format according to S1:

1. S1 is constant: S1 is constant: if S2 is a single byte address, S1 value is copied by byte to the address corresponding Length bytes headed with S2 ; if S2 is a word address, S1 value is copied by word to the address corresponding Length words headed with S2;
2. S1 is address: it is transferred by byte address with no regard to the address type match of S1 and S2.

S2 : destination address.

**Example:**

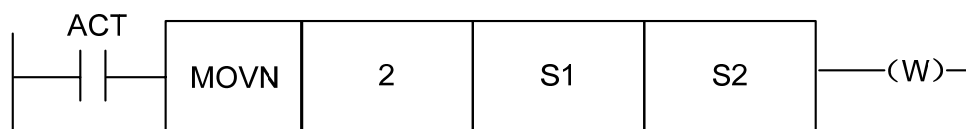


Fig. 5-25-2

1. When S1 is the constant 5 and S2 is R60, R60=00000101, R61=00000101

2. When S1 is the constant 5 and S2 is D60, D60=1285

3. When S1 is the constant D50 and S2 is D60, D60=D50

W=1, the specified number byte is delivered.

W=0, no data be delivered.

If it detects that it exceeds the range of parameter type in transferring, the alarm will be issued

## 5.26 MOVB (transfer of one byte)

### Function:

It transfers one byte data from specified source address to a specified destination address.

### Format:

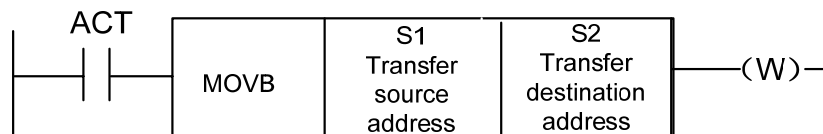


Fig. 5-26-1

### Control condition:

ACT=0: No data is transferred.

ACT=1: one byte length data is transferred.

### Parameter:

S1 : source address or constant.

if S2 is a single byte address, S1 is copied to S2 address by byte unit; if S2 is word unit, it is copied by word address;

S2 : destination address.

## 5.27 MOVW (transfer of two bytes)

### Function:

It transfers two bytes data from specified source address to a specified destination address.



**Format:**

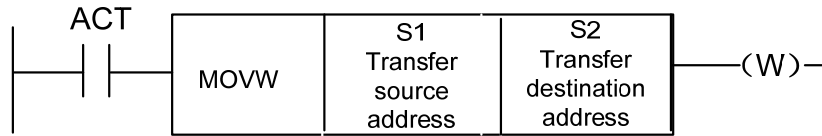


Fig. 5-27-1

**Control condition:**

ACT=0: No data is transferred.

ACT=1: one byte length data is transferred.

**Parameter:**

S1 : source address or constant.

S2 : destination address.

## 5.28 XMOV (Binary index data transfer)

**Function:**

This function instruction is used for reading and rewriting the data in the data table. The number of data (table capacity) in the data table can be specified by specifying the address, the PLC operates the data table according to the user setting.

**Format:**

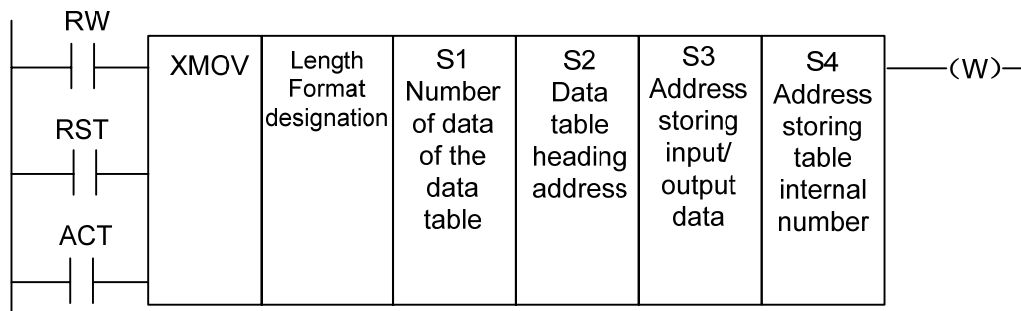


Fig. 5-28-1

**Control condition:**

Specify the reading or rewriting (RW)

RW=0: data is read from the data table.

RW=1: data is written in the data table.

Reset (RST)

RST=0: release reset.

RST=1: reset W=0.

Execution instruction (ACT):

ACT=0 : The XMOV instruction is not executed. W does not change.

ACT=1 : The XMOV instruction is executed.

**Parameter:**

Length : Specify the data long.

1: 1-byte long data.

2: 2-byte long data.

S1 : Storage address of data number of data table . The address is used to store the data number of data table, its byte should correspond to the length specified in Length format specification, and the effective range of data number of data table is set by the byte length specified by Length1 format.

1 byte length: 1 to 512.

2 byte length: 1to 256, i.e.  $256 \times 2=512$  byte, which is the capacity of the PLC data table.

S2 : Sets the head address of the data table. The memory of data table is (byte length )X (number of data table elements). The head address must be the value set in the D data table.

S3 : Input/output data storage address. In case of the reading, set the address of the memory which stores a reading result ; in case of writing, set the address of the memory which stores a writing result. its byte should correspond to the length specified in Length format specification.

S4 : index storage address. Set the address of the memory in which an index value is stored. its byte should correspond to the length specified in Length format specification.

If setting an index value above the value to set in S1 storage address of number of data table elements, it causes an error and output W=1. Actual transfer address=head address+ index which is  $0 \sim (S1-1)$ , actual transfer address cannot exceed the number in the data table.

**Output:**

If the index value set in Index storage address exceeds the value set in S1, W=1, the reading or writing of the data table isn't executed.

W=0, No error.

W=1, Error found.

## 5.29 DSCH (binary data search)

### Function:

The DSCH instruction is used to search the binary data in data table. The number of data (table capacity) in the data table can be specified by specifying the address. Thus allowing change in table capacity even after writing the sequence program in the ROM.

### Format:

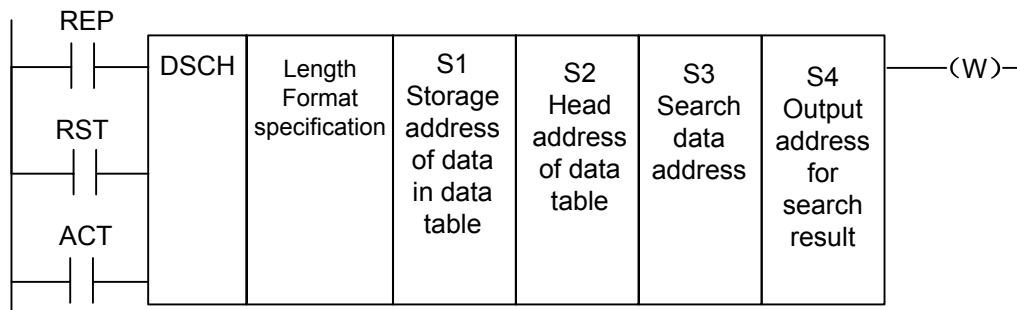


Fig. 5-29-1

### Control conditions:

Check repetition (REP)

REP=0: Execute the DSCH instruction, search begins from the head address, and the repetition will be omitted, the search stops when the target data is found in the first time, and output its address. If the searched data is not found, W=1.

REP=1: Execute the DSCH instruction, if the searched data is not found or is two (or more than two), W=1.

Reset (RST)

RST=0: Release reset.

RST=1: Reset. W=0.

Activation instruction (ACT):

ACT=0 : Do not execute DSCH instruction, W does not change.

ACT=1 : Execute DSCH instruction. If the search data is found, table number where the data is stored will be output. If the search data is not found, W becomes 1.

### Parameter:

Length : Specifies data length

1: 1-byte long data.

2: 2-byte long data.

- S1 : Storage address of number of data in data table. This address requires memory of number of byte according to the format designation. Number of data in the table is n+1(head number in the table is 0 and the last number is n).
- S2 : Data table head address.
- S3 : Search data input address.
- S4 : Search result output address. Actual transfer address=head address+ index which is 0~(S1-1), actual transfer address cannot exceed the number in the data table. After searching, if search data is found, the table number where the data is stored will be output. the searched table number is output in this search result output address. This address requires memory of number of byte according to the format designation.

#### Output:

W=0, Search data found.

W=1, Search data not found.

### 5.30 ADD (addition)

#### Function:

This instruction performs binary addition between 1-byte, 2-byte data. The required number of bytes is necessary to store each augend, the added, and the operation output data.

#### Format:

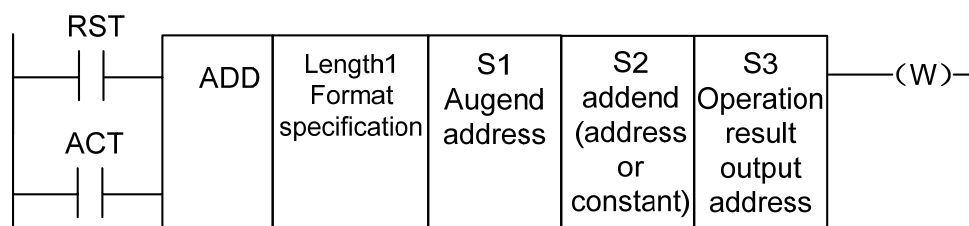


Fig. 5-30-1

#### Control conditions:

Reset (RST):

RST=0: Release reset.

RST=1: Reset. W=0.

Activation instruction (ACT):

ACT=0 : Do not execute ADD instruction. W does not change.

ACT=1 : Execute ADD instruction.

**Parameter:**

Length : Specifies data length(1 or 2 bytes) and the format for the addend(constant or address).

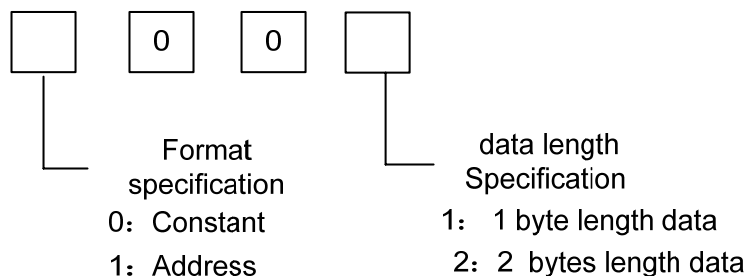


Fig. 5-30-2

S1 : Address storing the Augends.

S2 : Addend data specification defined by the Length specification.

S3 : Specify the output address of operation result

**Output:**

W=0: Operation normal.

W=1: Operation abnormal.

If the result of addition exceeds the specified data length, W=1.

### 5.31 SUB (binary subtraction)

**Function:**

This instruction executes the subtraction operation in the binary format of 1 or 2 bytes. In the A required number of bytes is necessary to store the subtrahend, and the result.

**Format:**

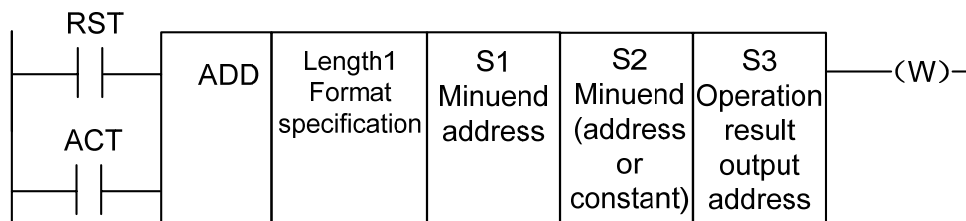


Fig. 5-31-1

**Control conditions:**

Reset (RST):

RST=0: Release reset.

RST=1: Reset. W=0.

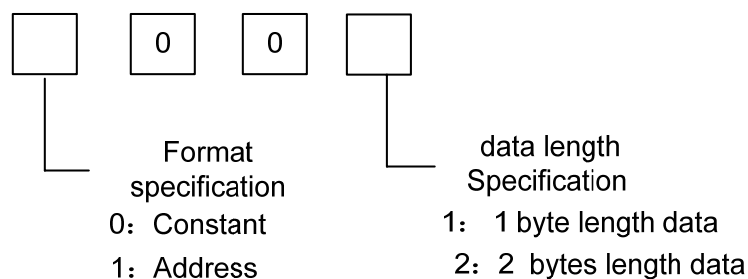
Activation instruction (ACT):

ACT=0 : Do not execute SUB instruction. W does not change.

ACT=1 : Execute SUB instruction.

**Parameter:**

Length : Specifies data length(1 or 2 bytes) and the format for the subtrahend(constant or address).



**Fig. 5-31-2**

S1 : Address storing the minuend.

S2 : Subtrahend data specification defined by the Length specification.

S3 : Specify the output address of operation result.

**Output:**

W=0: Operation normal.

W=1: Operation abnormal.

If the result of subtraction exceeds the specified data length, W=1.

## 5.32 ANDF (functional and)

**Function:**

The ANDF instruction ANDFs the contents of address A with a constant( or the contents of address B), and stores the result at address C.

**Format:**

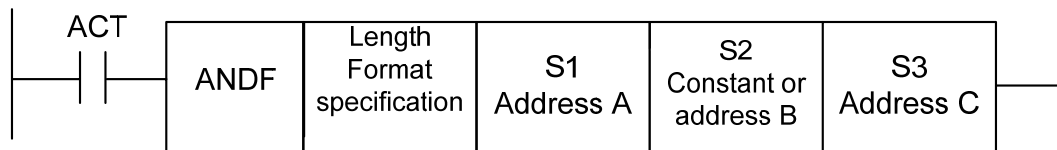


Fig.5-32-1

**Control conditions:**

ACT=0 : The ANDF instruction is not executed.

ACT=1 : The ANDF instruction is executed.

**Parameter:**

Length : Specify a data length (1 or 2 bytes), and an input data format(constant or address specification).

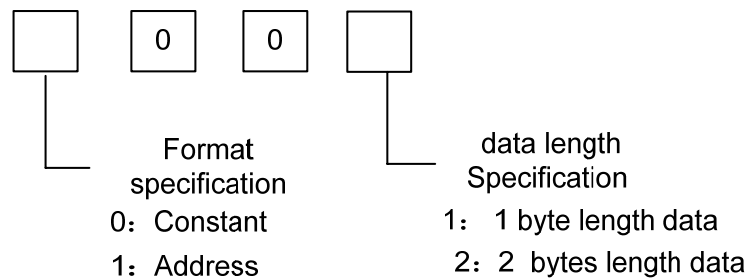


Fig. 5-31-2

- S1 : specifies the input data to be ANDed. The data that is held starting at this address and has the data length specified in Length format specification is treated as input data.
- S2 : Input data to be ANDed with. When address specification is selected in format specification, the data that is held starting at this address and has the data length specified in Length format specification is treated as input data.
- S3 : Address used to store the result of an ANDF operation. The result of an ANDF operation is stored starting at this address, and has the data length specified in Length format specification.

**Example:.**

When address A and address B hold the following data:

Address A	1	1	1	0	0	0	1	1
Address B	0	1	0	1	0	1	0	1

The result of the ANDF operation is as follows:

Address C

0	1	0	0	0	0	0	1
---	---	---	---	---	---	---	---

### 5.33 ORF (functional or)

#### Function::

The ORF instruction ORFs the contents of address A with a constant (or the contents of address B),and stores the result at address C.

#### Format:

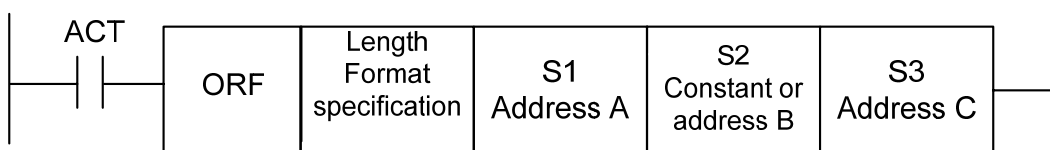


Fig. 5-33-1

#### Control conditions:

ACT=0 : The ORF instruction is not executed.

ACT=1 : The ORF instruction is executed.

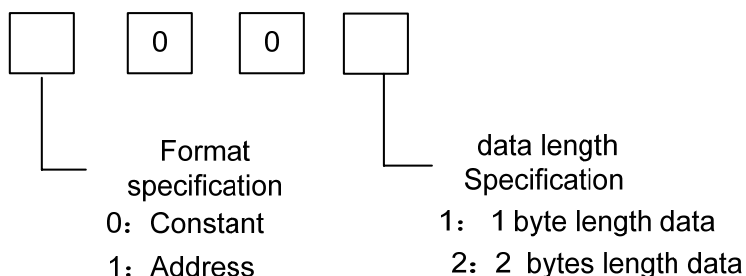


Fig. 5-33-2

#### Parameter:

Length: Specify a data length(1 or 2 bytes), and an input data format(constant or address specification).

S1 : Specify the input data to ORed. The data that is held starting at this address and has the data length specified in Length format specification is treated as input data.

S2 : Input data to be ORed with. When address specification is selected in format specification, the data that is held starting at this address and has the data length specified in Length format specification is treated as input data.

S3 : Address used to store the result of an ORF operation. The result of an ORF



operation is stored starting at this address, and has the data length specified in format specification.

**Example:**

When address A and address B hold the following data:

Address A	1	1	1	0	0	0	1	1
-----------	---	---	---	---	---	---	---	---

Address B	0	1	0	1	0	1	0	1
-----------	---	---	---	---	---	---	---	---

ORF operation results are as follows:

Address C	1	1	1	1	0	1	1	1
-----------	---	---	---	---	---	---	---	---

## 5.34 NOT (logical not)

**Function:**

The NOT instruction inverts each bit of the contents of address A, and stores the result at address B.

**Format:**

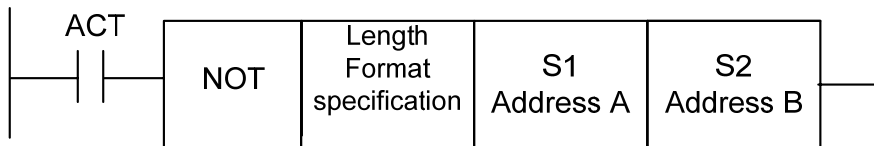


Fig. 5-34-1

**Control condition:**

ACT=0: The NOT instruction is not executed.

ACT=1: The NOT instruction is executed.

**Parameter:**

Length : Specifies a data length(1 or 2 bytes).

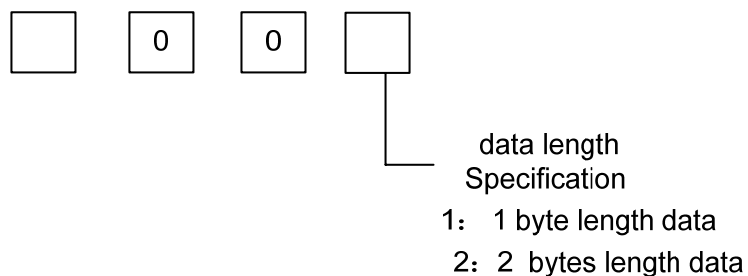


Fig. 5-34-2

- S1 : Input data to be inverted bit by bit. The data that is held starting at this address and has the data length specified in Length format specification is treated as input data.
- S2 : Address used to output the result of a NOT operation. The result of a NOT operation is stored starting at this address. And has the data length specified in Length format specification.

**Example:**

When address A holds the following data:

Address A	1	1	1	0	0	0	1	1
-----------	---	---	---	---	---	---	---	---

The result of the NOT operation is as follows:

Address B	0	0	0	1	1	1	0	0
-----------	---	---	---	---	---	---	---	---

**5.35 EOR (exclusive or)****Function:**

The EOR instruction exclusive-Ors the contents of address A with a constant( or the contents of address B), and stores the result at address C.

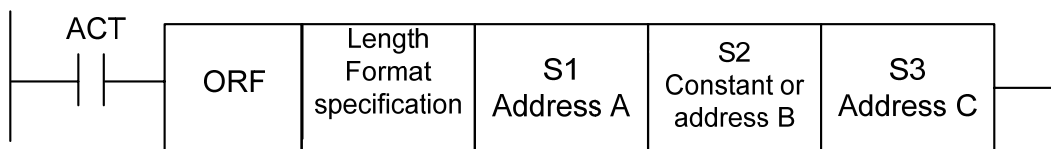
**Format:**

Fig. 5-35-1

**Control conditions:**

ACT=0 : The EOR instruction is not executed.

ACT=1 : The EOR instruction is executed.

**Parameter:**

Length : Specify a data length (1 or 2 bytes) and an input data format(constant or address specification).

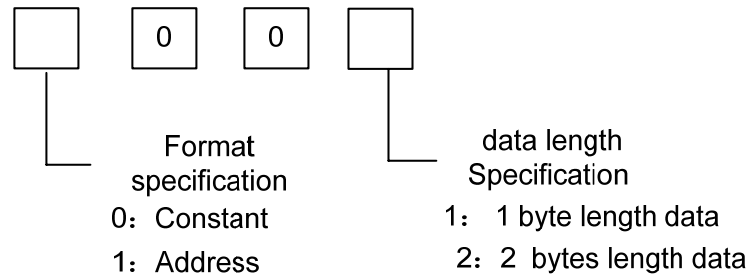


Fig. 5-35-2

- S1 : Input data to be exclusive-ORed. The data that is held starting at this address and has the data length specified in Length format specification is treated as input data.
- S2 : Input data to be exclusive-ORed with. When address specification is selected report that specification, the data that is held starting at this address and has the data length specified in Length format specification is treated as input data.
- S3 : Address used to store the result of an exclusive EOR operation. The result of an exclusive EOR operation is stored starting at this address, and has the data length specified in Length format specification.

**Example:**

When address A and B hold the following data:

Address A	1	1	1	0	0	0	1	1
Address B	0	1	0	1	0	1	0	1

The result of the exclusive EOR operation is as follows:

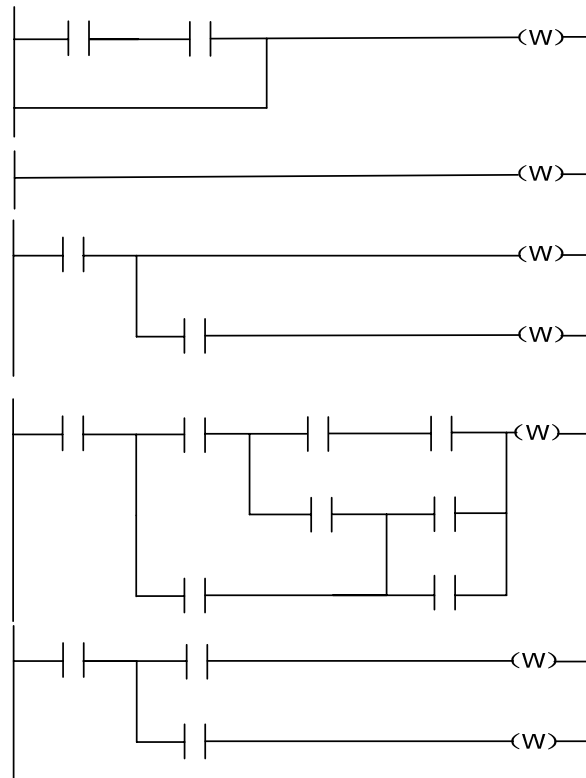
Address C	1	0	1	1	0	1	1	0

# 6

## Ladder Writing Limit

1. Sequence program must have END1 and END2 which are taken as the end marks of 1<sup>st</sup> level and 2<sup>nd</sup> level sequence part, and END1 must be before END2.
2. They only support the parallel output and do not support the multi-level output.
3. The result output address in all basic instructions and output function instruction are not set the following addresses:
  - 1) Counter preset address DC, timer preset address DT.
  - 2) K0~K5 address is occupied by the system, and the user can't define them.
  - 3) G63, R255 address are occupied by the system. and the user can't define them.
  - 4) X address and F address on IO input interface.
4. Such case like vertical line overhanging, node disconnected, horizontal through line paralleling to the node network will result in the nodes or network that can't be executed, so alarm will be issued by the system.
5. Star network, in which there is no direct connection between the vertical lines of different lines in a column, and a line in the middle isn't jointed with a vertical line. So alarm will be issued because the case can't be processed by the system.
6. The upward convex is not allowed in the network. That is there is a parallel network above the nodes of a line, and no line can be connected to this network. So alarm will be issued

The followings are the phrasing error, and the system will alarm.



**Fig. 6-1**

## II      **Function**



# 1

## Controlled Axis

### 1.1 Output of movement state of an axis

General The movement state of each axis can be output to the PLC.

#### Signal Axis moving signals

**MV1~MV4 (F017#0~F017#3)**

[Classification] Output signal

[Function] These signals indicate that a controlled axis is moving.

MV1: X is moving.

MV2: Y is moving.

MV3: Z is moving.

MV4: A is moving.

[Output conditions]

The signals become 1 when:

- The corresponding axis has started moving.

The signals become 0 when:

- The corresponding axis has stopped moving.

#### Axis moving direction signals

**MVD1~MVD4 (F019#0~F019#3)**

[Classification] Output signal

[Function] These signals indicate the movement direction of controlled axis.

MV1: movement direction of X.

MV2: movement direction of Y.

MV3: movement direction of Z.

MV4: movement direction of A.

[Output conditions]

“0” indicates the corresponding axes are negatively moving,

“1” indicates the corresponding axes are positively moving.



**Caution:**

These signals maintain their condition during a stop, indicating the direction of the axes' movement before stopping.

**Signal address**

	#7	#6	#5	#4	#3	#2	#1	#0
F017					MV4	MV3	MV2	MV1
F019					MVD4	MVD3	MVD2	MVD1

**1.2 Servo ready signal**

**Signal**      **servo ready signal**

**SA (F000#6)**

[Classification]      Output signal

[Function]      After the servo is ready, SA signal becomes 1. For the axis with absorption brake, release the brake when outputting the signal, execute the brake when the system does not output the signal.

**Signal address**

	#7	#6	#5	#4	#3	#2	#1	#0
F000		SA						

## 2 Preparation for Operation

### 2.1 Emergency stop

**General** If you press Emergency Stop button on the machine operator panel, the machine movement stops immediately.

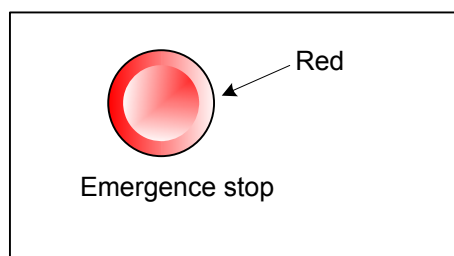


Fig. 2-1

The button is locked when it is pressed, Although its release method varies with the machine tool builder, the button can usually be unlocked by twisting it.

**Signal** **Emergency stop signal**

**\*ESP (G001.0 )**

[Classification] Input signal

[Function] input an emergency stop signal to stop the machine instantly.

[Operation] When the emergency stop \*ESP becomes 1, the emergency stop is applied to the machine and the CNC is reset.

**Signal address**

	#7	#6	#5	#4	#3	#2	#1	#0
G001								*ESP

### 2.2 CNC overtravel signal

**General** When the tool tries to move beyond the stroke end set by the machine tool limit switch, the tool decelerates and stops as a result of tripping the limit switch, and an Over TRAVEL is displayed. The signal can be output with an alarm.

**Signal****Overtravel signal**

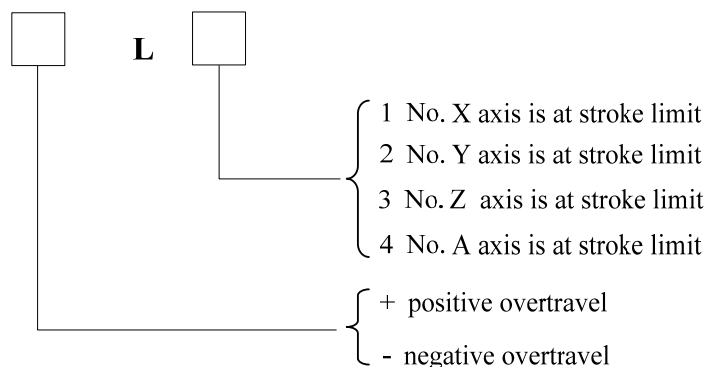
+L1~+L4(G12#0~G12#3)

−L1~−L4(G13#0~G13#3)

[Classification] Input signal

[Function] Indicates that the control axis has reached its stroke limit. There are individual signals for each direction in every control axis.

The + /- in the signal name indicate the direction and the number corresponds to the control axis.



[Operations] “0”: the controlled unit operates as follows:

Automatic operation: If even one axis overtravel signal becomes, all axes are decelerated to stop, an alarm is given and operation is halted.

Manual operation: Only the axis whose overtravel signal has become 0 is decelerated to a stop, and the axis can be moved in the opposite direction.

Once the axis overtravel signal has become 0, the axis direction is registered.

Even if the signal returns to 1, it is not possible to move that axis in that direction until the alarm is cleared.

**Signal address**

	#7	#6	#5	#4	#3	#2	#1	#0
G012					+L4	+L3	+L2	+L1
G013					−L4	−L3	−L2	−L1

## 2.3 Alarm signal

**General** When an alarm is triggered in the CNC, the alarm is displayed on the screen, and the alarm signal is set to 1. If the voltage level of the memory backup battery falls to below a specified level while the CNC is turned off, the battery alarm signal is set to 1.

**Signal** **Alarm signal**

**AL (F001 #0)**

[Classification] Output signal

[Function] Alarm signal reports CNC is in an alarm state as follows:

- a) P/S alarm
- b) Overtravel alarm
- c) Servo alarm

[Output conditions] The alarm signal is set to 1 when:

——The CNC is placed in the alarm state.

The alarm signal is set to 0 when:

——The alarm has been released by resetting the CNC.

**Signal address**

	#7	#6	#5	#4	#3	#2	#1	#0
F001								AL

## 2.4 Running mode selection

**Signal** **mode check signal**

**F003#0~F003#7**

[Classification] Output signal

[Function] Report the current selected operation mode

**Signal address**

	#7	#6	#5	#4	#3	#2	#1	#0
F003	MZRO	MEDT	MMEM	MRMT	MMDI	MJ	MH	MINC

## 2.5    Status output signal

### Cutting feed signal

**CUT (F002 #6)**

[Classification]    Output signal

[Function]    These signals indicate that the cutting feed is being performed by automatic operation.

[Output conditions]    These signals are 1 when:

Cutting feed is being performed by automatic operation (cutting feed for linear interpolation, circular interpolation, helical interpolation, thread cutting, skip cutting, or cutting in canned cycle).

**Note:**

Do not output the signal in the state of feed hold.

Output the signal during the interlock or the feedrate override is set to 0.

### Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
F002		CUT						

# 3

## Manual Operation

### 3.1 JOG feed/incremental feed

#### General

- JOG feed** In JOG mode, setting a feed axis and direction selection bit to 1 on the machine operator's panel moves the machine along the selected axis in the selected direction.
- Incremental feed** In incremental feed mode, setting a feed axis and direction selection bit to 1 on the machine operator's panel moves the machine one step along the selected axis in the selected direction. The minimum distance the machine moves, is the least input increment. The step can be 10, 100, or 1000 times the least input increment.

The only difference between JOG feed and incremental feed is the method of selecting the feed distance. In JOG feed, the machine continues to be fed while the following signals selecting the feed axis and direction are 1: +J1, -J1, +J2, -J2, +J3, -J3, etc. In incremental feed, the machine is fed by one step. Using JOG feedrate override dial can regulate JOG feedrate. The step distance can be selected by MPG feed movement distance G026 #0~G026 #3.

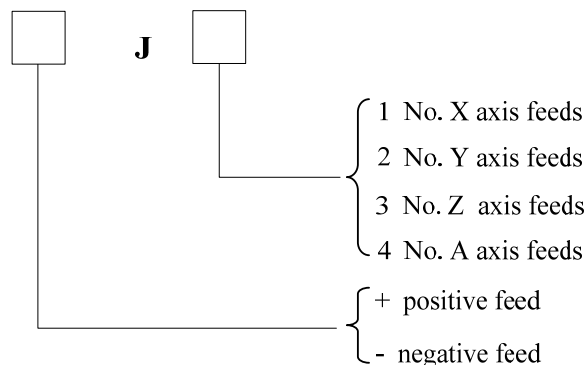
#### Signal Feed axis and direction selection signal

+J1~+J4 (G27 #0~G27 #3)

-J1~-J4 (G28 #0~G28 #3)

[Classification] Input signal

[Function] In JOG feed or Incremental feed mode, select the required feed axis and direction. +/- in the signal name indicates the feed direction, the number corresponds to the controlled axis.



[Operation] When the signal is set to 1, the control unit operate as follows:

When JOG feed or incremental feed is enabled, the control unit moves the specified axis in the specified direction.

When the signal is set to 1 in JOG feed, the control unit continues to feed that axis while the signal is set to 1.

In incremental feed, the control unit feeds the requested axis by the step distance which is specified by the manual handle feed move distance selection signal, then the axis stops. Even if the signal is set to 0 while the axis is being fed, the control unit does not stop moving.

To feed the axis again set the signal to 0, then to 1 again.

Manual rapid traverse selection signal

#### RT (G024#7)

[Classification] Input signal

[Function] Select the rapid traverse rate in JOG feed or incremental feed mode.

[Use] When the signal becomes 1, the control unit operates as follows:

- The control unit executes the jog feed or incremental feed at a rapid traverse rate. The rapid traverse override is validated.
- When the signal is switched from 1 to 0 or vice versa in jog feed or incremental feed, the feedrate is decelerated until it reaches zero, then increased to the specified value. During acceleration and deceleration, the feed axis and direction selection signal can be kept 1.

#### Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G024	RT							
G027					+J4	+J3	+J2	+J1
G028					−J4	−J3	−J2	−J1

### **3.2 MPG/Step feed**

**General** In MPG/Step feed mode, the machine moves by rotating the manual pulse generator(MPG)/Step. Select the axis along which the machine moves with the MPG feed axis selection signal/axis move signal.

**Signal** MPG/Incremental select signal

(G026#0~G026#3)

[Classification] Input signal

[Function] When the signal selects the step feed, MPG generates a movement distance of a pulse by pressing axis moving key every time.

which also can select the movement distance per step of incremental feed .



# 4 Reference Point Return

## 4.1 Manual reference point return

**General** In manual reference point return mode, the machine tool move in the specified direction by setting the position parameter N0: 7#3~#7 to execute the reference point return. The selected axis on the panel reports the axis to execute the machine zero return, which is not related to the move direction of axis.

**The following signals are related to the manual reference point return:**

Table 4-1-1

	Manual reference point return
Reference point return deceleration signal	DECX,DECY,DECZ,DECA
Reference point return completion signal	ZP1,P2, ,ZP3,ZP4

**signal**

**Reference point return completion signals**

**ZP1~ZP4(F016#0~F016#3)**

[Classification] Output signal

[Function] These signals report that the machine tool is at the reference point on a controlled axis.

These signals correspond separately to all axes.

Table 4-1-2

ZP1	X axis reference point return completion signal.
ZP2	Y axis reference point return completion signal.
ZP3	Z axis reference point return completion signal.
ZP4	A axis reference point return completion signal.

[Output conditions] When these signals becomes 1:

- Manual reference point return is completed and the current position is in the in-position area.
- The automatic reference point return(G28) is completed and the current position is in the in-position area.
- The reference point return check is completed and the current

position is in the in-position area.

When the signal becomes 0:

- The machine tool moves from the reference point.
- The emergency stop signal appears.
- The servo alarm appears.

#### Reference point return deceleration signal check

**DECX (X017#0) DECY (X017#1) DECZ (X017#2) DECA (X017#3)**

[Classification] Input signal

[Function] These signals decelerate the feedrate for manual reference point return to a low feedrate.

## 4.2 Reference point return check signal

### 1<sup>st</sup> reference point check permission signal

**PREF20---PREF23 (G056#0---#3)**

### 2<sup>nd</sup> reference point permission signal

**PREF20---PREF23 (G057#0---#3)**

### 3<sup>rd</sup> reference point permission signal

**PREF30---PREF33 (G058#0---#3)**

### 4<sup>th</sup> reference point permission signal

**PREF40---PREF43 (G059#0---#3)**

[Type] Input signal

[Function] When the signal is set to 1, the reference point return completion signals(F42, F43, F44) are enabled. Respectively

These signals correspond respectively to all axes.

**Table 4-2-1**

PREF*0	X axis reference point check permission signal
PREF*1	Y axis reference point check permission signal
PREF*2	Z axis reference point check permission signal
PREF*3	A axis reference point check permission signal

### 1<sup>st</sup> reference point return completion signal

**ZP21---ZP24 (F016#0---#3)**

**2<sup>nd</sup> reference point return completion signal**

**ZP21---ZP24 (F042#0---#3)**

**3<sup>rd</sup> reference point return completion signal**

**ZP31---ZP34 (F043#0---#3)**

**4<sup>th</sup> reference point return completion signal**

**ZP41---ZP44 (F044#0---#3)**

[Type] output signal

[Function] These signals report that the machine tool is at the reference point on a controlled axis.

These signals correspond separately to all axes

**Table 4-2-2**

ZP*1	X axis reference point return completion signal
ZP*2	Y axis reference point return completion signal
ZP*3	Z axis reference point return completion signal
ZP*4	A axis reference point return completion signal

[Output conditions] the signal is enabled when it is the reference point check permission signals(G57, G58, G59) become 1.

When these signals becomes 1:

- Manual reference point return is completed and the current position is in the in-position area.
- The automatic reference point return(G30) is completed and the current position is in the in-position area.
- The reference point return check is completed and the current position is in the in-position area.

When the signal becomes 0:

- The reference point check permission signal (G57, G58,G59 become 0,
- The machine tool moves from the reference point.
- The emergency stop signal appears.
- The servo alarm appears.

## 4.3 Area check signal

### Area check signal

#### AQ1—AQ3 (F045#0---#2)

[Type] Output signal

[Function] These signals report that the machine tool is at the reference point on a controlled axis.

These signals correspond respectively to all axes.

**Table 4-3-1**

AQ1	X axis area check signal
AQ2	Y axis area check signal
AQ3	Z axis area check signal

[Output conditions]

When the machine is in the stored travel check 1( the data parameter set **P66~P75** the limit, and outside the stored travel check 2 (the data parameter **P76~P8** or program command can set the limit of this side), the signal becomes 1, otherwise becomes 0.

Signal addresses

	# 7	# 6	# 5	# 4	# 3	# 2	# 1	# 0
F016					ZP4	ZP3	ZP2	ZP1
F042					ZP24	ZP23	ZP22	ZP21
F043					ZP234	ZP33	ZP32	ZP31
F044					ZP44	ZP43	ZP42	ZP41
F045						AQ3	AQ2	AQ1
G017					DECA	DECZ	DECY	DECX
G057					PREF23	PREF22	PREF21	PREF20
G058					PREF43	PREF32	PREF31	PREF30
G059					PREF43	PREF42	PREF41	PREF40

# 5

## Automatic Operation

### 5.1 Cycle start/feed hold

#### General

**Start of automatic Operation (cycle start)** When automatic operation start signal ST is set to 1 then 0 while the CNC is in memory mode, DNC operation mode or MDI mode, the CNC enters the automatic operation start state then starts operating.

The signal ST is ignored as follows:

1. When the CNC is in other modes except for AUTO, DNC, or MDI mode.
2. When the feed hold signal (SP) is set to 1.
3. The emergency stop signal (ESP) is set to 1.
4. When <RESET> on MDI panel is pressed.
5. When CNC is in the state of alarm.
6. When the automatic operation is started.
7. When the program restart signal (SRN) is set to 1.
8. When CNC is searching one sequence number.

The CNC enters the feed hold state and stops operation in automatic operation as follows:

1. When the feed hold signal (SP) is set to 1.
2. The single block instruction is end when the single block is running.
3. MDI operation is completed.
4. CNC alarms.
5. The single block instruction is end after the mode is changed to others or Edit mode.

In automatic operation, the CNC enters the feed hold and stops running as follows:

1. When the emergency stop signal (ESP) is set to 1.
2. When <RESET> on MDI panel is pressed.

\* Halt of automatic operation

(feed hold) When the feed hold signal SP is set to 1 in automatic operation, the CNC enters the feed hold state and stops operation. At the same time, cycle start lamp signal STL is set to 0 and feed hold lamp signal SPL is set to 1. Re-setting signal SP to 0 in itself will not restart automatic operation. To restart automatic operation, first set signal SP to 0, then set signal ST to 1 and to 0.

**Signal Cycle start signal**

**ST (G023#6)**

[Type] Input signal

[Function] Start the automatic operation.

[Operation] When signal ST is set to 1 then 0 in automatic operation(Auto), DNC and MDI mode, the CNC enters the cycle start state and starts operations.

**Feed hold signal**

**SP (G023#7)**

[Classification] Input signal

[Function] Halt the automatic operation

[Operation] In Auto mode, SP signal is set to 1, CNC enters the feed hold and stops running. When SP signal is set to 0, the automatic operation does not start.

**Cycle start lamp signal**

**STL (F000#5)**

[Classification] Output signal

[Function] The signal reports PLC that the automatic operation start is entered.

[Output conditions] The signal is set to 1 or 0, which is determined by CNC state as Fig. 5.1.

**Feed hold lamp signal**

**SPL (F000#4)**

[Classification] Output signal

[Function] The signal reports PLC that the feed hold is entered.

[Output conditions] The signal is set to 1 or 0, which is determined by CNC state as Fig. 5.1.

**Automatic operation signal**

**OP (F000#7)**

[Classification] Output signal

[Function] The signal reports PLC that the automatic operation is entered.

[Output conditions] The signal is set to 1 or 0, which is defined by CNC state as Fig. 5.1.

**Table 5.1**

	Cycle start lamp STL	Feed hold lamp SPL	Automatic operation lamp OP
Cycle start	1	0	1
Feed hold	0	1	1
Automatic operation stopping	0	0	0
Reset	0	0	0

### Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G023	SP	ST						
F000	OP		STL	SPL				

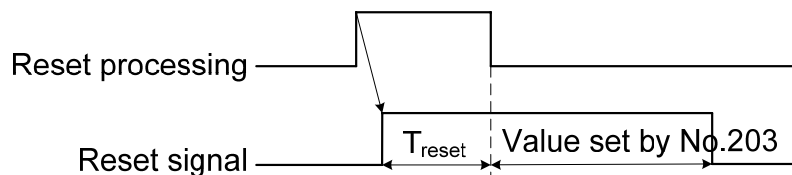
## 5.2 Reset

**General** CNC is reset and enters the reset state as follows:

1. When the emergency signal (ESP) is set to 1.
2. When <RESET> on MDI panel is pressed.

When the CNC is reset, the resetting signal RST is output to the PLC. The resetting signal RST is set to 0 when the resetting signal output time, set by No. 203, has elapsed after the above conditions have been released.

$$RST = T_{\text{reset}} (\text{Reset processing time}) + \text{parameter setting value by No. 203}$$



**Fig.5-2**

When the CNC is reset in automatic operation, the automatic operation is stopped and movement axis is decelerated and stopped. When the CNC is reset during the execution of the M, S, T

function, signal MF, SF or TF is set to 0 within 16ms.

#### **RST (F001 #1)**

[ Classification ]      Output signal

[ Function ]              The signal reports PLC that CNC is reset, it is used for PLC reset processing.

[ Output conditions ]    The signal is set to 1 when:

1: When the emergency stop signal (ESP) is set to 1.

2: When <RESET> on MDI panel is pressed.

The signal is set to 0 when:

When the reset signal output time set by No. 203# is completed after the above are released and CNC is reset.

#### **Signal address**

	#7	#6	#5	#4	#3	#2	#1	#0
F001							RST	

## **5.3 Testing a program**

**General**      Before machining is started, the automatic running check can be executed. It checks whether the established program can operate the machine as desired. This check can be accomplished by running the machine or view the position display change without running the machine.

### **5.3.1 Machine tool lock**

**General**      The change of the position display can be monitored without moving the machine.

When all-axis machine lock signal MMLK is set to 1, output pulses to the servo motors are stopped in manual or automatic operation. The instructions are distributed, however, updating the absolute and relative coordinates. The operator can therefore check if the instructions are correct by monitoring the position display.

#### **all-axis machine lock check signal**

#### **MMLK (F004 #1)**



[ Classification ] Output signal

[ Function ] The signal reports PLC of the state of all-axis machine tool lock signal.

[ Output condition ] When the signal is set to 1, all-axis machine tool lock signal is set to 1.

When the signal is set to 0, all axes machine tool lock signals are set to 0.

#### Signal address

	# 7	# 6	# 5	# 4	# 3	# 2	# 1	# 0
F004							MMLK	

### 5.3.2 Dry run

**General** Dry run is valid only for automatic operation. The machine moves at a constant feedrate regardless of the feedrate specified in the program. The feedrate is set by data parameter P86.

This function is used to check the movement of the machine without a workpiece.

**Signal** **Dry run signal**

**DRN (G021 #2)**

[ Classification ] Input signal

[ Function ] Enables dry run.

[ Operation ] When the signal is set to 1, the machine tool moves at the feedrate specified for dry run.

When the signal is 0, the machine tool normally moves.

#### Caution:

When the dry run signal is changed from 0 to 1 or 1 to 0 during the movement of the machine, the feedrate of the machine is first decelerated to 0 before being accelerated to the specified feedrate.

#### Signal address

	# 7	# 6	# 5	# 4	# 3	# 2	# 1	# 0
G021						DRN		

### 5.3.3 Single block

**General** The single block operation is valid only for automatic operation. (Auto mode).

When the single block signal (SBK) is set to 1 during automatic operation, the CNC enters the automatic operation stop state after executing the current block. In sequence automatic operation, the CNC enters the automatic operation stop state after executing each block in the program. When the single block signal (SBK) is set to 0, execute the automatic operation over again

**Signal**      **Single block signal**

**SBK (G021 #1)**

[Classification]      Input signal

[Function]      Enables single block operation.

[Operation]      Execute the single block when the signal is set to 1.

Execute the normal operation when the signal is set to 0.

**Single block check signal**

**MSBK (F004 #3)**

[Classification]      Output signal

[Function]      The signal reports PLC of the state of single block signal.

[Operation]      The signal is set to 1 as follows:

——When the single block signal SBK is set to 1.

The signal is set to 0 as follows:

——When the single block signal SBK is set to 0.

**Note:**

1. Operations in thread cutting

When the SBK signal becomes 1 in thread cutting, the operation stops after the first non-thread cutting signal after thread cutting instruction.

2. Operation in canned cycle

When the SBK signal becomes 1 during canned cycle operation, the operation stops at each positioning, approach, drilling and retraction instead of the end of the block. The SPL signal becomes 1 while the STL signal becomes 0, showing that the end of the block has not been reached. When the execution of one block is completed, the STL and SPL signals become 0 and the operation is stopped.

## Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G021							SBK	
F004					MSBK			

## 5.4 Optional block skip

**General** When a slash followed by a number is specified at the head of a block, and optional block skip signal BDT is set to 1 during automatic operation, the block is ignored.

**Signal** Optional block skip signal

**BDT (G021#0)**

[Classification] Input signal

[Function] Select whether a block with “/” is neglected.

[Operation] In automatic operation, when BDT is 1, the block with “/” is ignored.  
The program is normally executed when BDT is 0.

**Optional block skip check signal**

**MBDT (F004#0)**

[Classification] Output signal

[Function] The signal reports PLC of the state of skip optional block BDT.

## Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G021								BDT
F004								MBDT

## 5.5 Program restart

**General** A program may be restarted at a block by specifying the sequence number of the block, after automatic operation is stopped because of a broken tool or for holidays.

**Signal** Program restart signal

SRN<G021#6>

[Classification] Input signal

[Function] Select the program restart

[Operation] When the program restart signal is set to 1 to search for the sequence number of the block to be restarted, the CRT screen changed to the program restart screen. When the program restart signal is set to 0, and automatic operation is activated, the machine moves back to the machining restart point at dry run speed along the axes one by one. When the machine moves to the restart point, machining restarts.

#### Signal during program restart

SRNMV<F002#4>

[Classification] Output signal

[Function] Report the program is started.

[Output conditions] The signal becomes 1 when:

—When G21#6 is 1 in automatic mode, the program restarting signal is set to 1.

The signal becomes 0 when ::

—The program restart sequence ends(all controlled axes of machine tool moves to the restart point).

#### Signal address

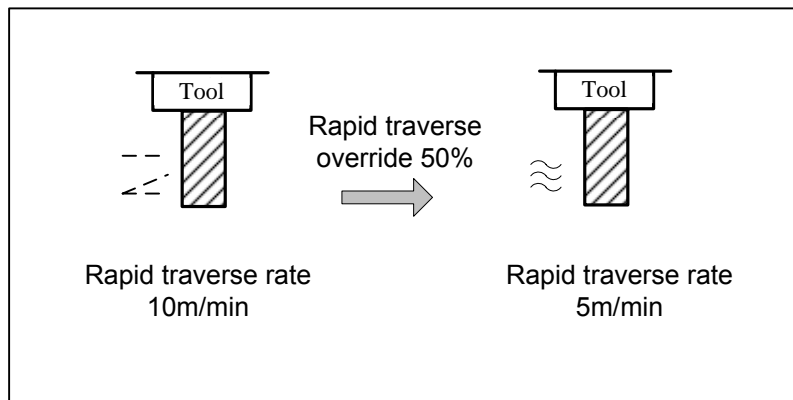
	#7	#6	#5	#4	#3	#2	#1	#0
G021		SNR						
F002				SRNM				

# 6

## Feedrate Control

### 6.1 Rapid traverse rate

**General** The 4 gears ( F0, 25%, 50%, 100%) can be used for rapid traverse rate.



**Fig. 6-1**

**Feedrate:** Actual moving speed is obtained by multiplying the override value by the value set by the data parameter (P088-092) either in the auto mode or manual operation mode (including manual reference point return and program zero return).

**F0** : it is set by the data parameter P093.

**Signal rapid traverse override signal**

**Rapid F0** (G025#0)

**Rapid F25%** (G025#1)

**Rapid F50%** (G025#2)

**Rapid F100%** (G025#3)

[Classification] Input signal

[Function] it is the rapid traverse override signal

**The code detection signal of the rapid traverse override (G11#0~G11#1)**

[Type] Input signal

[Function] It is the code detection signal of the rapid traverse override

[Operation] The code signal corresponding with the following override

Table 6-1-1

Code signal at the rapid traverse rate		Override value
RV0	RV1	
0	0	100%
0	1	50%
1	0	25%
1	1	0%

## 6.2 Feedrate override

**General** A programmed feedrate can be reduced or increased by a percentage selected by the override dial. This feature is used for a program check. For example, when a feedrate of 100 mm/minute is specified in the program, set the override dial to 50% move to the tool at 50 mm/min.

**Signal** Feedrate positive override signal (G24#0)  
 Feedrate negative override signal (G24#2)  
 [Classification] Input signal  
 [Function] Cutting feedrate override signal. 16 gears (0%~150%).  
 [Operation] Actual feedrate is obtained by multiplying the specified speed by the override value selected by this signal.

## 6.3 Override cancel

**General** The override cancel signal fixes the feedrate override to 100%.

**Signal** Override cancel signal

**OVC (G024#1)**

[Classification] Input signal

[Function] fixes the feedrate override to 100%.

[Operation] When the signal is 1, CNC operates as follows:

- The feedrate override is fixed to 100% irrespective of the feedrate override signal.
- Rapid traverse override and spindle speed override are not affected.

**Signal address**

	#7	#6	#5	#4	#3	#2	#1	#0
G024							OVC	

# 7

## Auxiliary Function

### 7.1 Miscellaneous function (M code)

**General** \* miscellaneous function (M code) When the M code is specified, the code signal (F26~F33) and strobe signal are transmitted to PLC which uses these signals to start or cut off its relative functions.

#### Basic procedure

The following signals are used for the following functions.

**Table 7-1-1**

Function	Program address	Output signal			Completion signal
		Code signal	Strobe signal	Distribution completion signal	
MF	M	M**		DEN	FIN

- (1) Suppose that MXXX is specified in a program ( suppose that code signal is FYYY.Y)  
If MXX is not specified by the user, CNC alarms. When the user registers the M code in the system, the F signal is specified to be the only one, i.e. the code signal for F26~F33.
- (2) If a non-M, S, T, instructions such as move, dwell are specified with MST together, they are ran at the same time. When more than one code of the MST is specified in a same block, they will be executed by sequence.
- (3) Set the register signal FYYY.Y and strobe signal F007#0 to 1 in executing MXX, and ensure the reply signal RESP (G63#0) is set to 0 by PLC.
- (4) Upon completion of the operation, the PLC sets completion signal MFIN (G00#1) and FIN (G00#0) to 1. The completion signal is used by the miscellaneous function, spindle speed function, tool function. If any of these functions are executed simultaneously, the completion signal must be set to 1 upon completion of all the function.
- (5) Ensure that the PLC sets the reply signal MRESP (G63#0) to 0 upon completion of MXX.
- (6) The M, S, T instruction in a same block can be executed simultaneously, and CNC executes the next block when the completion signal FIN is confirmed to be 1

## 7.2 Miscellaneous function (S code)

General Miscellaneous function (S code) When the M code is to be executed, the I/O point or analog control is set by bit parameter No.1#2.

### Basic procedure for spindle S code I/O point control:

Table 7-2-1

Function	Program address	Output signal			Completion signal
		F address signal	Strobe signal	Response signal	
S code function	S	S* (F22)	TF (F007 # 2)	SRESP (G63#1)	FIN(G00#0) SFIN(G00#4)

These S code functions are explained as the following:

- (1) Suppose that SX is specified in a program:  
X range is 0~7, alarm is issued if it is beyond this range, and S0~S7 correspond to F address signals F22#0~#7 respectively. (For S1, it corresponds to the F address signal F22#1.)
- (2) If a non-M, S, T instruction such as move, dwell is specified in the same block with the miscellaneous function, these instructions will be executed simultaneously.
- (3) When S1 is executed, the F address signal F22#1 and the strobe signal should be set to 1, and the response signal SRESP(G63#1) should be set to 0 by PLC.
- (4) As the operation is finished, the completion signals SFIN (G00#4) and FIN (G00#0) are set to 1 by PLC. If M, S, T functions are to be executed simultaneously, the completion signal FIN (G00#0) can't be set to 1 till they are finished.
- (5) When S1 is executed, the response signal SRESP(G63#1) should be set to 0 by PLC.
- (6) M, S, T codes in a same block will be executed simultaneously, and only the completion signal FIN is set to 1 could next block be executed.

### Basic procedure for spindle S code analog control

Table 7-2-1

Function	Program address	Output signal		Gear shift finish signal	Response signal	Completion signal
		F address signal	Strobe signal			
S code function	S****	S**** (F34#0~#2)	TF (F007 # 2)	GRAR (G02#4)	SRESP (G63#1)	FIN(G00#0) SFIN(G00#4)

These S code functions are explained as the following:



- (1) Suppose that SXXX is specified in a program, F34#0~#2 is defined by data parameter P246~248, this signal can be used for gear exchange by PLC. (etc. S500 by 1000 set by data parameter P246.)
- (2) If a non-M, S, T instruction such as move, dwell is specified in the same block with the miscellaneous function, these instructions will be executed simultaneously.
- (3) When S500 is to be executed, the F address signal F34#0 and the strobe signal should be set to 1, and the response signal SRESP(G63#1) should be set to 0 by PLC.
- (4) When the gear shift is over, the gear shift completion signal GRAR (G02#4) is set to 1 by PLC.
- (5) As the S code is finished, the completion signals SFIN(G00#4) and FIN(G00#0) are set to 1 by PLC. If M, S, T functions are to be executed simultaneously, the completion signal FIN(G00#0) can't be set to 1 till they are finished.
- (6) When S500 is executed, the response signal SRESP(G63#1) should be set to 1 by PLC.
- (7) M, S, T codes in a same block will be executed simultaneously, and only the completion signal FIN is set to 1 could next block be executed.

### 7.3 Miscellaneous function (T code)

**General** Miscellaneous function (T code) T code uses with M code together, such as T06M06; When the T code is to be executed, the code signal (D241) and the strobe signal are sent to PLC, by these signals PLC switch on or off its functions.

#### Basic processing

The signals are used for the following functions:

**Table 7-3-1**

Function	Program address	Output signal			Completion signal
		Code signal	Strobe signal	Response signal	
T code function	T**	T** (D241)	BF (F007 #3)	SRESP (G63#2)	FIN(G00#0) TFIN(G00#5)

- (1) Suppose that TXX is specified in a program (XX is sent to data signal D241) :
- (2) If a non-M, S, T instruction such as move, dwell is specified in the same block with the miscellaneous function, these instructions will be executed simultaneously. If multiple MST codes are specified in the same block, these codes will be executed by sequence.

- (3) If TXX is to be executed, the strobe signal F007#3 is set to 1.
- (4) As the operation is finished, the completion signals TFIN (G00#5) and FIN (G00#0) are set to 1 by PLC. If M, S, T codes are to be executed simultaneously, the completion signal FIN (G00#0) can't be set to 1 till they are finished.
- (5) M, S, T codes in a same block will be executed simultaneously, and only the completion signal FIN is set to 1 could next block be executed.

**Signal**      **M code login signal**  
**M00~M99 (F026~F033)**  
**M code strobe signal**  
**MFEFD (F007 #0)**

[Type]          Output signal

[Function]      These signals report PLC the specification of miscellaneous function.

[Output conditions]    For relative output conditions and procedure, please see “7.1 Miscellaneous function (M code)” .

**Note:**          1. The following miscellaneous functions are only processed in CNC: they are not output even programmed.

- \*    M98, M99, M198
- \*    M codes for calling subprograms
- \*    M codes for call customer macro programs

- 2. The code signal and strobe signal as well as decoding signal can be output for the miscellaneous function listed below.

M00, M01, M02, M30

- 3. M code is given by binary code by M00~M39.

For example: M5 corresponds to 00000000, 00000000, 00000000, 00000101.

**M decoding signal**

**DM00 (F009 #7)**

**DM01 (F009 #6)**

**DM02 (F009 #5)**

**DM30 (F009 #4)**

[Type]          Output signal

These signals point out particular miscellaneous function actually specified. The correspondence list of program instruction miscellaneous functions to output signals is as following:

Table 7-3-2

Instruction	Output signal
M00	DM00
M01	DM01
M02	DM02
M30	DM30

[Output conditions] M decoding signal is 1 while:

- The corresponding miscellaneous function is specified, and any move and dwell instructions specified in the same block are completed. These signals are not output when the **completion** signal of the miscellaneous function is returned before completion of such move instructions and dwell instructions.

M decoding signal is 0 while as:

- FIN signal becomes 1.
- Reset.

#### **M code completion signal**

**MFIN<G000#1>**

#### **M code response signal**

**MRESP<G063#0>**

[Type] Input signal

[Function] It means M code execution is completed.

[Action] For the control unit operation and procedure, please see “7.1 Miscellaneous function (M code)”.

#### **S code strobe signal**

**TF (F007 #2)**

[Function] These signals notify PLC the spindle speed function specified.

[Output conditions] For relative output conditions and procedure, please see “7.2 Miscellaneous function (S code)”.

#### **S code completion signal**

**SFIN<G000#4>**

#### **S code response signal**

**SRESP<G063#1>**

[Type] Input signal

[Function] It means S code execution is completed.

[Action] For the control unit operation and procedure, please see “7.2

Miscellaneous function (S code) ”.

#### T code strobe signal

##### BF (F007#3)

[Type] Output signal

[Function] These signals specifies the tool function.

[Output conditions] For relative output conditions and procedure, please see “7.3 Miscellaneous function (T code)” .

#### T code completion signal

##### TFIN<G000#5>

[Type] Input signal

[Function] It means T code execution is completed.

[Action] For the control unit operation and procedure, please see “7.3 Miscellaneous function (T code)” .

#### Miscellaneous function completion signal

##### FIN (G000#0)

[Type] Input signal

[Function] It specifies the completion of M, S, T codes.

[Action] For the control unit operation and procedure, please see section 7.1, 7.2, 7.3 .

#### Warning

All the functions above share the same completion signal FIN (G000#0) , and this signal must be set to 1 after all the functions are completed.

#### Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G000			TFIN	SFIN			MFIN	FIN
G063							SRESP	MRESP
F007					BF	TF		MF
F009	DM00	DM01	DM02	DM30				

7.4 Auxiliary function lock

**General**     Inhibits execution of a specified M, S, and T function. That is, code signals and strobe signals are not issued. This function is used to check a program.

**Signal**       **Auxiliary function lock signal**

**AFL (G021 #3)**

[Classification]     Input signal

[Function]     The signal selects the auxiliary function lock, i.e., the signal disables the execution of the specified M, S, T function.

[Operation]     When the signal becomes 1, the control unit functions are as follows:

- 1. The control unit does not execute M, S, and T functions. That is, the control unit stops the output of code signals and strobe signals.
- 2. If this signal becomes “1” after code signal output, the output operation is executed in the ordinary manner until its completion (that is, until the FDURING signal is received, and the strobe signal becomes to “0”.)
- 3. Among the miscellaneous function, M00,M01, M02 and M30 are executed even when this signal is “1”. All code signals, strobe signals, decode signals are output in the ordinary manner.
- 4. Even when this signal is “1”, M98 and M99 are executed in the control unit without outputting their execution results are executed in the ordinary manner.

**Warning**       Even when this signal is “1”, spindle analog output is executed.

**Auxiliary function lock check signal**

**MAFL (F004 #4)**

[Classification]     Output signal

[Function]     The signal reports the state of auxiliary function lock signal AFL.

[Output conditions]     When the signal is 1, the auxiliary function lock signal AFL is 1.  
When the signal is 0, the auxiliary function lock signal AFL is 0.

**Signal address**

	#7	#6	#5	#4	#3	#2	#1	#0
G021					AFL			
F004				MAFL				

## 8 Spindle Speed Function

### 8.1 Spindle speed control mode

**General** For 990MA system, the spindle is divided into gear spindle and analog spindle:

1. In gear spindle mode, CNC changes S code to switch value to output to the spindle to control the spindle speed.
2. In analog spindle, changes S code to analog value to output to the spindle to control the spindle speed. CNC.

The control mode is I / O or analog is set by bit parameter NO: 1#2.

#### 8.1.1 Gear spindle

**General** The gear spindle is defined that the spindle S code is controlled by I/O point.

**Signal** Spindle speed strobe signal

TF (F007#2)

**Gear spindle address signal**

F22#0~F22#7

[Classification] Output signal

[Function] These signals report the actually specified the spindle speed function.

[Output conditions] For the output conditions and procedure, see “7.2 S code I/O control”.

**Note:** S code range:S1~S8, the system alarms if it exceeds the range, S1~S8 separately corresponds to F address signal F022#0~#7.

**Note:** S code range: S1~S8, the alarm may occur if it exceeds its range, S1~S8 are separately corresponding with the F address signal F022#0~#7. User can refer to three gears (S1, S2 and S3) of the configured ladder diagram from the system. So, S4~S8 can not be used, the corresponding ladder diagram should be added if the user needs.

#### 8.1.2 Analog spindle

**General** The analog spindle is defined that the spindle speed is controlled by the analog voltage value from CNC. So, CNC changes S code into the analog voltage value to output to the spindle of machine tool to control the spindle speed.

The actual output analog voltage value equals to the S value controlled by the spindle multiplying the spindle override.

**Signal Spindle positive override signal (G22#5)**

Spindle negative override signal (G22#3)

Spindle override cancel signal OVC (G22#4)

[Classification] Input signal

[Function] The signal specifies the S override change controlled by spindle.

**Spindle override code detection signal (G019#0~G019#3)**

[Classification] Input signal

[Function] It is the spindle override code detection signal

Spindle override code detection signal has 4 binary system code signals which are corresponding with the override:

So, the spindle override can be selected based upon the 10% unit within the 0~150%. When the spindle speed control is performed instead of using the spindle speed override, the setting override value is 100%.

**Notice:** The spindle speed override function in the tapping cycle and thread cutting is disabled.

**Gear change process:**

Although S instructs the spindle speed, the actual is to control the spindle motor. So, CNC needs to confirm the corresponding relation between the spindle motor and gear. Like S instruction selection, CNC selects the gear according to the previously defined gear speed range by parameter to report PLC to select the corresponding the gear by using the gear change select signal (GR3, GR2, GR1). At the same time, CNC outputs the spindle motor speed according to the selected gear. CNC outputs the instruction corresponded to the spindle (GR1, GR2, GR3 output) speed by specifying S0~S99999 in MDI mode. 2 or 3 speed gear (GR1, GR2, GR3) is set by No.246~248 to simultaneously output to the gear select signal. When the gear select signal is changed, CNC simultaneously output SF signal).

Specification of gear change signal is as follows:

**Table 8-1-2-1**

	No. 2 gear	No. 3 gear	Remark
GR1	Low	Low	Low: low gear
GR2	High	Medium	Medium: middle gear
GR3		High	High: high gear

- When the instruction voltage is 10V, the low gear spindle speed is A (parameter No.246) ( $\text{min}^{-1}$ ) ( $\text{min}^{-1}$ ).
- When the instruction voltage is 10V, the high gear spindle speed is B (parameter

No.247) ( $\text{min}^{-1}$ ) (middle gear during 3<sup>rd</sup> gear).

- When the instruction voltage is 10V, the high gear spindle speed is c (parameter No.248) ( $\text{min}^{-1}$ ) (3<sup>rd</sup> gear) .

S and spindle motor speed instruction voltage (0~10V) and gear selection signal.

(GR1, GR2, GR3) is as the above figure.

**Signal: Gear selection signal**

**GR1,GR2,GR3**

**<F034#0~#2>**

[Classification] Output signal

[Function] These signals report PLC the selected gear.

[Output conditions] For the definition of these signals, see Gear change Mode.

**Gear change selection signal (input)**

**GR1,GR2,GR3<G002#0~#2>**

[Classification] Input signal

[Function] These signals report CNC the current selected gear.

[Output conditions] For the definition of these signals, see Gear change Mode.

**Gear change in-position signal**

**GEAR<G002#4>**

[Classification] Input signal

[Function] These signals report CNC in-position of the current selected gear.

[Output conditions] For the definition of these signals, see Gear change Mode.

**Signal address**

	# 7	# 6	# 5	# 4	# 3	# 2	# 1	# 0
G002				GEAR		GR3	GR2	GR1
G022			SPOV	OVC	SMOV		P	
F007						TF		
F034						GR3	GR2	GR1



**General** In a general tapping canned cycle, synchronous control is applied to the tapping operation of a tapping axis and the operation of the spindle.

Namely, during rigid tapping (G74, G84), CNC needs to detect the rotation direction signal of spindle to confirm the cutting feed direction and machining process.

Spindle rotating→ Z tool infeed tapping→ transmit M05 to spindle→ wait for spindle to completely stop→ transmit CCW instruction→ starting point of Z tool retraction→ spindle stops rotating

So, to realize the rigid tapping, the corresponding ladder must be written to report the rotation direction of CNC external spindle.

## RGTATP (G003#1)

[ Classification ]      Output signal

[Function] Report to PLC that CNC is in the rigid tapping mode.

[Output conditions] RGTAP 1: the current CNC is in the rigid tapping mode.

0: the current CNC is not in the rigid tapping mode.

	#7	#6	#5	#4	#3	#2	#1	#0
G003							RGTAP	

## 9 Programming Instruction

### 9.1 Customer macro program

**General** Although subprograms are useful for repeating the same operation, the customer macro function also allows use of variables, arithmetic and logic operations, and conditional branches for easy development of general programs. A machining program can call a custom macro with a simple instruction, just like a subprogram.

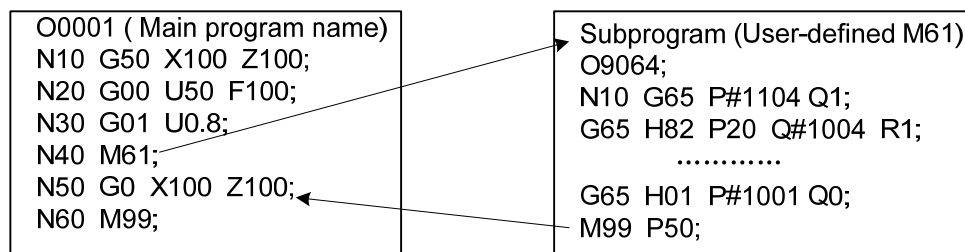


Fig. 9-1-1

This means some function programmed by macro program can be taken as the general function. i.e. the program can be written by the data variable (variable data or unknown data).

For example, the customer program can be used for technology.

#### Signal Customer macro program input signal

##### UI000~UI015 (G054, G055)

[Classification] Input signal

[Function] The signals do not provide any functions for the control unit. These signals which are taken as one of system variable is read by macro program, used for the interface signal between macro program and PLC.

The system variable corresponding to these signals are as follows:

Table 9-1-1

Signal	Address	Variable
UI000	G54#0	#1000
UI001	G54#1	#1001
UI002	G54#2	#1002
UI003	G54#3	#1003

UI004	G54#4	#1004
UI005	G54#5	#1005
UI006	G54#6	#1006
UI007	G54#7	#1007
UI008	G55#0	#1008
UI009	G55#1	#1009
UI010	G55#2	#1010
UI011	G55#3	#1011
UI012	G55#4	#1012
UI013	G55#5	#1013
UI014	G55#6	#1014
UI015	G55#7	#1015
UI000~ UI015	G54, G55	#1032

Note: #1032 is variable with 16-bit as follows:

### Signal address

	# 7	# 6	# 5	# 4	# 3	# 2	# 1	# 0
# 1032	UI007	UI006	UI005	UI004	UI003	UI002	UI001	UI000
# 1032	UI015	UI014	UI013	UI012	UI011	UI010	UI009	UI008

### Custom macro program output signal

**UO000~UO015**

**(F054~F055)**

**UO100~UO131**

**(F056~F059)**

[Classification] Output signal

[Function] The signals do not provide any functions for the control unit. These signals which are taken as one of system variable are read/written by macro program, used for the interface signal between macro program and PLC.

The system variable corresponding to these signals are as follows:

Table 9-1-2

Signal	Address	Variable
UO000	F54#0	#1100
UO001	F54#1	#1101
UO002	F54#2	#1102
UO003	F54#3	#1103
UO004	F54#4	#1104
UO005	F54#5	#1105
UO006	F54#6	#1106

UO007	F54#7	#1107
UO008	F55#0	#1108
UO009	F55#1	#1109
UO010	F55#2	#1110
UO011	F55#3	#1111
UO012	F55#4	#1112
UO013	F55#5	#1113
UO014	F55#6	#1114
UO015	F55#7	#1115
UO000~ UO015	F54, F55	#1132
UO100~ UO115	F56~F59	#1133

**Note:**

# 1132 is a variable with 16-bit.

# 1133 is a variable with 32-bit.

Composition is as follows:

	# 7	# 6	# 5	# 4	# 3	# 2	# 1	# 0
# 1132	UO007	UO006	UO005	UO004	UO003	UO002	UO001	UO000
# 1132	UO015	UO014	UO013	UO012	UO011	UO010	UO009	UO008

## 9.2 Canned cycle

**General** Canned cycles make it easier for the programmer to create programs. With a canned cycle, a frequently-used machining operation can be specified in a single block with a G function; without canned cycles, normally more than one block is required. In addition, the use of canned cycles can shorten the program to save memory.

One canned cycle consists of a sequence of six operations:

Operation 1: Positioning a hole

Operation 2: Rapid traverse up to R level

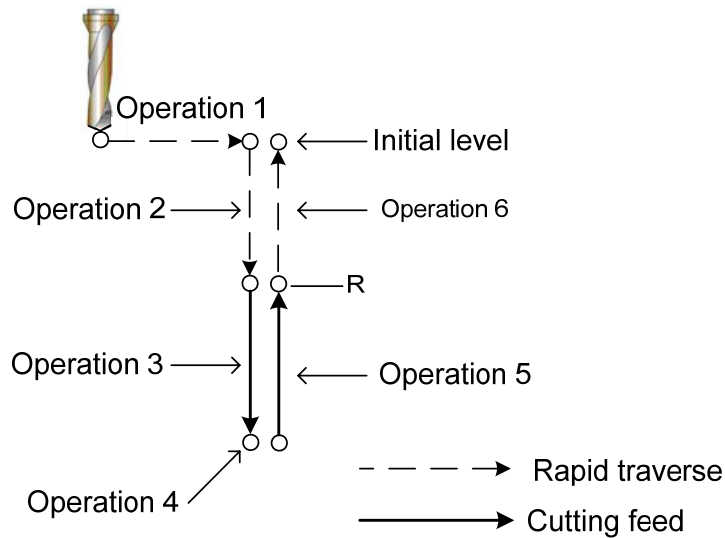
Operation 3: Hole machining

Operation 4: Operation at the bottom of a hole

Operation 5: Retraction to point R level

Operation 6: Rapid traverse up to the initial point

Operation sequence of canned cycle is as follows:



**Fig. 9-2-1** (Operation sequence of canned cycle)

The spindle control is required to output negative rotation spindle instruction in some canned cycle.

The following canned cycles require spindle control:

Reverse tapping cycle G74	Fine boring cycle G76
Tapping cycle G84	Boring cycle G86
Back boring cycle G87	Boring cycle G88

For spindle control, the following normal miscellaneous functions are used:

See the description of the miscellaneous functions.

M03: spindle CCW

M04: Spindle CW

M05: Spindle stops

M19: Spindle orientation

When the rotation direction of the spindle is to be switched from one direction to the other (for example, when M04 is output during M03 operation), a parameter can be specified whether to send M05 at the time switching).

**Tapping signal** In the tapping cycle, output the tapping signal. When the tapping cycle G code is valid, CNC also outputs the tapping signal.

**Override** In the tapping, the cutting feedrate override is always set to 100%.

**Feed hold** In the tapping, the traverse does not stop immediately when the feed hold is pressed down. But it stops when the tool returns to R level.

Dry run TDR (parameter 12#5) defines if the dry run is valid during the tapping.

**Signal Tapping signal**

**TAP<F001 #5>**

**[Classification] Output signal**

**[Function] The signal reports CNC is in tapping mode.**

**[Output conditions] The signal is 1 when:**

- CNC is in the tapping cycle mode G74, G84.
- CNC is in the tapping cycle mode G63. the signal is set to 0:

**The signal is 1 when:**

- CNC is neither in tapping cycle nor in tapping mode.
- The reset or emergency stop signal is input.

**Signal address:**

	#7	#6	#5	#4	#3	#2	#1	#0
F001			TAP	D TAP				

# 10

## Display/Setting

### 10.1 Clock Function

**General** Year, month and date are displayed on setting page.

The custom macro system variable can be used to read the time.

Time report can be read and written.

### 10.2 Displaying operation history

**General** This function displays a history of the key stroke and signal operations, performed by the CNC operator, when a failure or CNC alarm occurs.

### 10.3 Help function

**General** The help function displays on the screen detailed report about alarms issued in the CNC and about CNC operations. The screen displays detailed information about the alarms and how to recover from them. The detailed information is displayed only for a limited number of P/S alarms which are often misunderstood and are rather difficult to understand.

# 11

## Measurement

### 11.1 Skip function

**General** Linear interpolation can be commanded by specifying axis moving following the G31 instruction, like G01. If an external skip signal is input during the execution of this instruction, execution of the instruction is halted and the next block is executed.

The skip function is used when the end of machining is not programmed but specified with a signal from the machine, for example, in grinding. It is used also for measuring the dimensions of a workpiece.

The coordinate values when the skip signal is turned on can be used in a customer macro, because they are stored in the customer macro system variable #5061 ~ #5068, as follows:

#5061 1<sup>st</sup> axis coordinate value

#5062 2<sup>nd</sup> axis coordinate value

#5063 3<sup>rd</sup> axis coordinate value

:

**Signal** **Skip signal**

**SKIPP <G001#1>**

[Classification] Input signal

[Function] This signal terminates skip cutting. That is, the position where a skip signal turns to "1" in a block containing G31 is stored in a customer macro variable, and the move instruction of the block is terminated at the same time.

[Operation] When a skip signal turns to "1", the control unit works as described below.

- When a block contains a skip cutting instruction G31, the control unit reads and stores the current position of the specified axis at that time. The control unit stops the axis, then cancels the remaining distance that the block was supposed to be moved.
- The skip signal monitored is not for a rising edge, but for its state. So, if a skip signal continues to be "1", a skip condition is assumed to be satisfied immediately when the next skip cutting is specified.

**Note:**

The skip signal width requires at least 10ms.

**Signal address**

G001							SKIPP	
------	--	--	--	--	--	--	-------	--



# 12

## Panel locked setting

**Signal      Program edit lock signal**

**LEDT (G016#6)**

[Type]      Input signal

[Function] The signal locks the program edit function.

[Operation] When the signal is set to 1, program edit function is enabled.

When the signal is set to 0, program edit function is disabled, the program cannot be edited.

**Operator panel lock signal**

**LSYS (G016#7)**

[Type]      Input signal

[Function] The signal locks the press key on the machine operator panel.

[Operation] When the signal is set to 1, all keys on the panel are locked and disabled.

When the signal is set to 0, all keys on the panel are enabled.

**Signal address**

	# 7	# 6	# 5	# 4	# 3	# 2	# 1	# 0
G016	LSYS	LEDT						

## Appendix

### Signal list (By order of address)

#### Addresses between PLC and CNC

##### 1. CNC→PLC address: F000 ----- F064

Table appendix 1

Signal name	Symbol	Address
Feed hold signal	SPL	F000#4
Cycle start signal	STL	F000#5
Servo ready completion signal	SA	F000#6
Automatic operation signal	OP	F000#7
Alarm signal	AL	F001#0
Resetting signal	RST	F001#1
Tapping signal	TAP	F001#5
Tapping in process signal	D TAP	F001#6
Thread cutting signal	THRD	F002#3
Program restart signal	SRNMV	F002#4
Cutting feed signal	CUT	F002#6
Dry run check signal	MDRN	F002#7
Incremental feed selection check signal	MINC	F003#0
MPG feed selection check signal	MH	F003#1
JOG feed selection check signal	MJ	F003#2
Manual data input check signal	MMDI	F003#3
DNC operation selection confirm signal	MRMT	F003#4
Automatic operation selection check signal	MMEM	F003#5
Memory edit selection check signal	MEDT	F003#6
Machine zero return selection check signal	MZRO	F003#7
Skip optional block check signal	MBDT	F004#0
All-axis machine lock check signal	MMLK	F004#1
Single block check signal	MSBK	F004#3
Auxiliary function locked signal	MAFL	F004#4
Manual reference point return check	MREF	F004#5
Feedrate override OFF check signal	CFORD	F005#0
Spindle override OFF check signal	CSORD	F005#1
M function strobe signal	MFEFD	F007#0

S function strobe signal	TF	F007#2
T function strobe signal	BF	F007#3
Decode M signal	DM30	F009#4
	DM02	F009#5
	DM01	F009#6
	DM00	F009#7
Axis moving signal	MV1 --- MV5	F017
Axis moving direction signal	MVD1 --- MVD5	F019
Spindle I/O gear control signal	S00 --- S31	F22#0---#7
Spindle speed code signal	S00 --- S31	F22#0---#7
Miscellaneous function code signal	M**	F026 --- F033 ( see III Operation 3.5 )
Gear select signals	GR1,GR2,GR3	F034#0 --- #2
2nd reference point return completion signals		F042#0---#3
3rd reference point return completion signals		F043#0---#3
4th reference point return completion signals		F044#0---#3
Area check signals		F045#0---#2
Customer macro program output signal	U000 --- U015 U100 --- U131	F054,F055 F056 --- F059
Reference point establishment signal	ZRF1 ---- ZRF5	F060

## 2. PLC→CNC address: G000 ----- G064

Table appendix 2

Signal name	Symbol	Address
Miscellaneous function completion signal	FIN	G000#0
M function completion signal	MFIN	G000#1
S function completion signal	SFIN	G000#4
T function completion signal	TFIN	G000#5
Emergency stop signal	ESP	G001#0
Skip signal	SKIPP	G001#1
Gear select signal (input)	GR1,GR2, GR3	G002#0 --- #2
Gear shift in-position signal	GEAR	G002#4
Rigid tapping signal	RGTAP	G003#1
Overtravel signal	*+L1 --- *+L4	G012#0 ---- #3
	*-L1 ---- *-L4	G013#0 ---- #3
Edit lock signal	LEDT	G016#6
Operator panel lock signal	LSYS	G016#7
Zero return deceleration signal detect		G017#0 ---- #4

Zero return completion signal detect		G018#0 ---- #4
Edit mode		G20.0
Auto mode		G20.1
MDI mode		G20.2
Zero return mode		G20.3
Step mode		G20.4
Manual mode		G20.5
MPG mode		G20.6
DNC mode		G20.7
Skip		G21.0
Single block		G21.1
Dry run		G21.2
Auxiliary lock		G21.3
Machine loc k		G21.4
Optional stop		G21.5
Program restart		G21.6
Spindle override ( - )		G22.3
Spindle override cancel		G22.4
Spindle override ( + )		G22.5
Spindle JOG		G22.6
Cycle start		G23.6
Feed hold		G23.7
Feedrate override ( + )		G24.0
Feedrate override cancel		G24.1
Feedrate override ( - )		G24.2
Rapid switch		G24.7
Rapid Fo		G25.0
Rapid 25%		G25.1
Rapid 50%		G25.2
Rapid 100%		G25.3
Incremental step length 0.001		G26.0
Incremental step length 0.01		G26.1
Incremental step length 0.1		G26.2
Incremental step length 1		G26.3
MPG step length 0.001		G26.4
MPG step length 0.01		G26.5
MPG step length 0.1		G26.6
Manual feed axis +X		G27.0
Manual feed axis +Y		G27.1

Manual feed axis +Z		G27.2
Manual feed axis +4Th		G27.3
Manual feed axis -X		G28.0
Manual feed axis -Y		G28.1
Manual feed axis -Z		G28.2
Manual feed axis -4Th		G28.3
Spindle orientation		G29.0
Overtravel release		G30.0
Custom macro program interruption signal	UINT	G031#1
Custom macro program input signal	UI000 --- UI015	G054,G055
2 <sup>nd</sup> reference point check permission signal	PREF20----PREF23	G057#0 ----- #3
3 <sup>rd</sup> reference point check permission signal	PREF30----PREF33	G058#0 ----- #3
4 <sup>th</sup> reference point check permission signal	PREF40----PREF43	G059#0 ----- #3

### **III    PLC EDITTING**

# 1

## Summary

Presently, GSK990MA system only support the matched GSK ladder edit software.

GSK ladder edit software is the ladder editor in PC of GSK990MA series milling machine, machining center CNC system, providing the edit, change, searching error and print functions of GSK990MA series ladder. The software can run in Windows 97, Windows Me, Windows 2000, Windows XP and Windows 2003.

# 2 Software introduction

## 2.1 Start software

GSK ladder edit software is a green one that is not installed. The software package contains Lad Edit.exe, Diag.meas, and LadFile in which Ladder01 file is the system's standard ladder. Double-click Lad Edit.exe to run the software, and open Ladder01 ladder in LadFile as follows:

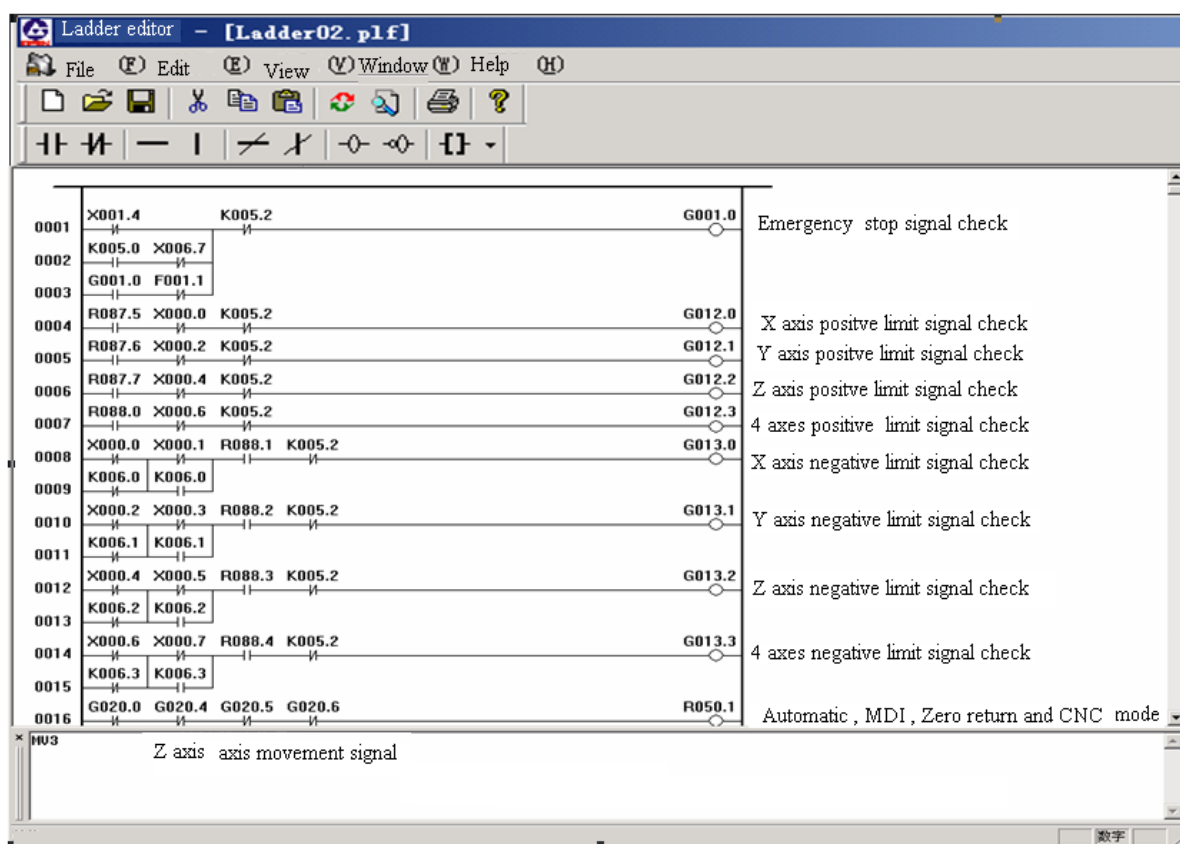


Fig.2-1-1

## 2.2 Function introduction

### ● File menu

The file menu contains New, Open and Save file, Create ladder or binary file, Print, Print Preview, Print Setup, Latest Open File List and other functions.



**Note:** use English instead of Chinese in “Ladder Version Number”, “Applicable Machine”, “Last Editor” in “Edit Ladder Message”, otherwise, the error occurs after the transmission is executed.

- **Edit menu**

Edit menu includes Cut, Copy, Paste, Search, Change, Edit and other functions

- **View menu**

Display or hide Tool Bar, Status Bar, Output Window and Instruction List Window.

- **Window menu**

Select and distribute each window.

- **Help menu**

Version information of the software.

# 3











## Software operation

### 3.1 Tool bar

There are two tool bars which are related to ladder edit in the main view frame,







#### 3.1.1 Main tool bar



-  creating a new ladder file
-  opening a ladder file
-  saving a ladder file
-  cutting the selected content to the clipboard
-  copying the selected content to the clipboard
-  pasting the content from the clipboard
-  changing a ladder
-  searching a element
-  printing a ladder
-  about dialog box

#### 3.1.2 Edit tool bar



-  adding a normally open contact
-  adding a normally closed contact
-  adding a horizontal conduct
-  adding a vertical conductive line (lower right of cursor)
-  deleting some element or horizontal conductive line
-  deleting a vertical conductive line at the lower right of element



adding a output coil



reversing an added output coil

function command button, there are two methods to edit function instructions:

1. Click the cursor at the right of button and the system pops-up the menu to select the function instruction.

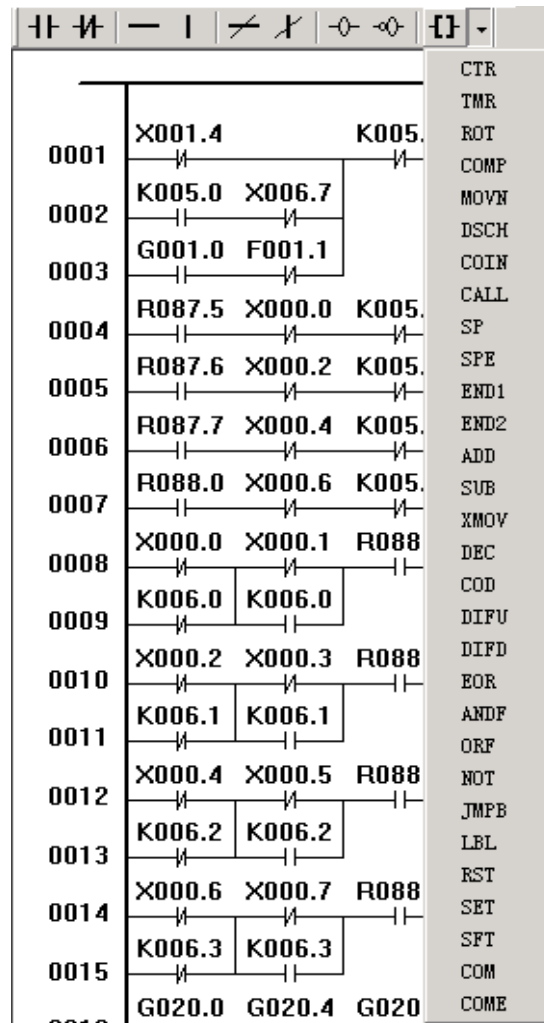


Fig. 3-1-2-1

2. Or, click the button, and the system pops-up the function instruction dialog box to execute the function instruction setting.

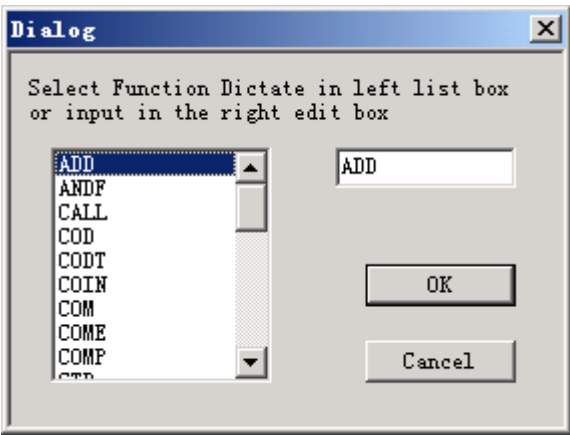


Fig. 3-1-2-2

3.2 Selecting a graph

In the edit view of ladder, the black rectangular shade is the cursor, and the user clicks the left mouse in the graphic edit area between the two busbars to select the position of the required edit graph as follows:

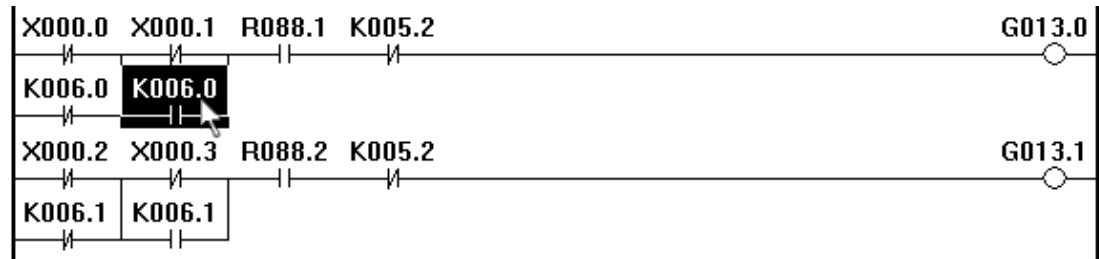


Fig. 3-2-1

When the block is selected, the user should press the left key of mouse in its initial position to drag the mouse to the end of the row, and the surroundings of the selected area is appeared by a rectangle with many dotted line before releasing the key.

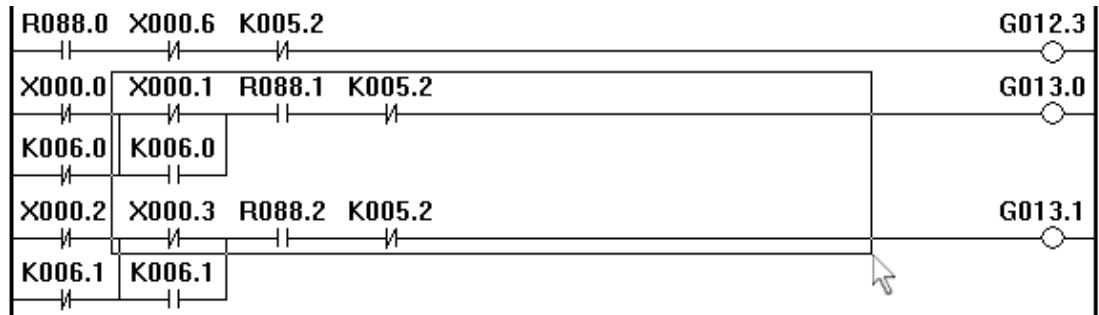


Fig. 3-2--2

After the mouse is released, the whole ladder becomes black, i.e., the ladder in the range is selected, and the user can execute the next operation, such as clip, delete, copy and so on.

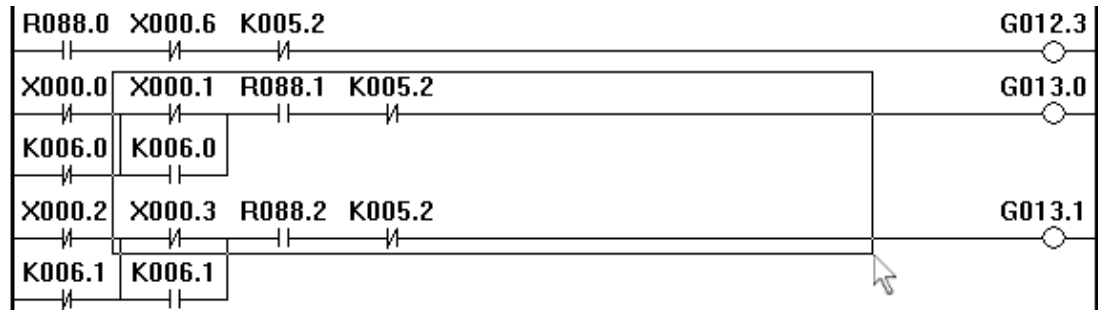


Fig. 3-2-3

### 3.3 Editing a graph

#### 3.3.1 Cutting

The user selects the required position to execute the operation by one of three methods as follows:

1. Click the right key of the mouse and the system pops-up the environmental menu to select the cut operation;
2. Click [Alt+E]--- [T] in the main menu;
3. Directly press the shortcut key [Ctrl+X];

After the cut content is placed to the clipboard, the user can execute the paste operation to copy it to the ladder.

#### 3.3.2 Copying

The user selects the required position to execute the operation by one of three methods as follows:

1. Click the right key of the mouse and the system pops-up the environmental menu to select the copy operation;
2. Click [Alt+E]--- [C] in the main menu;
3. Directly press the shortcut key [Ctrl+C];

After the copy operation is executed and the cut content is placed to the clipboard, the user can execute the paste operation to copy it to the ladder.

### 3.3.3 Paste

The user selects the required position to execute the operation by one of three methods as follows::

1. Click the right key of the mouse and the system pops-up the environmental menu to select the paste operation;
2. Click [Alt+E]--- [P] in the main menu;
3. Directly press the shortcut key [Ctrl+V];

### 3.3.4 Deleting

The user selects the required position to execute the operation by one of three methods as follows:

1. Click the right key of the mouse and the system pops-up the environmental menu to select [Alt+B]----delete the node;
2. Click the button- [Delete node] in the edit bar;
3. Directly press the shortcut key [Delete];

### 3.3.5 Inserting one row

The user moves the cursor to the required position to execute the operation by one of three methods as follows:

1. Click the right key of the mouse and the system pops-up the environmental menu to select the insert operation;
2. Click [Alt+E]--- [I] in the main menu;
3. Directly press shortcut key [Insert];

### 3.3.6 Deleting one row

The user moves the cursor to the required position to execute the operation by one of three methods as follows:

1. Click the right key of the mouse and the system pops-up the environmental menu to select the deletion operation;
2. Click [Alt+E]--- [D] in the main menu;
3. Directly press shortcut key [Ctrl+Delete];

### 3.3.7 Converting

The user changes the ladder program of current edit window into the instruction list program by one of three methods as follows:

1. Click [Alt+E]--- [V] in the main menu;
2. Click [Convert ladder] in the edit bar;
3. Directly press shortcut key [F7];

### 3.4 Ladder annotation

#### 3.4.1 Row annotation

Double-click the mouse left key outside the right busbar area of the ladder to input annotation in the edit box.

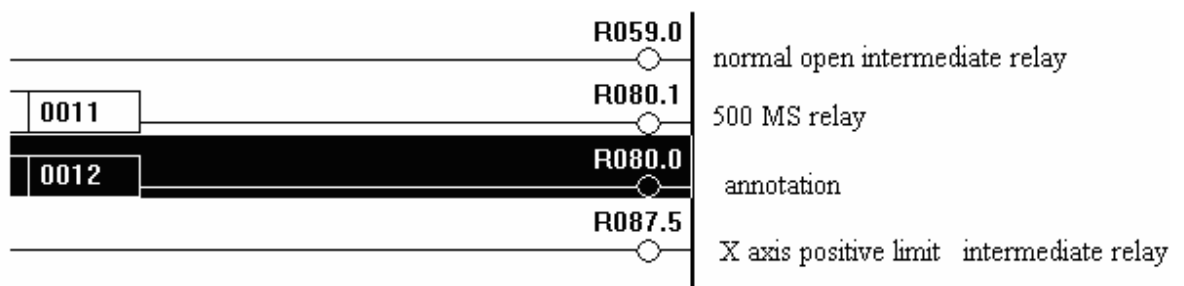


Fig. 3-4-1-1

#### 3.4.2 Element annotation

The user moves the cursor to the required position which element should be modified by one of two methods as follows:

1. Click the right key of mouse after the element is selected, and the system pops-up the environmental menu to select [Alt+M];

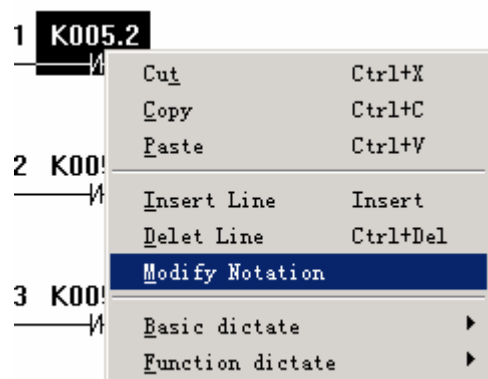


Fig. 3-4-2-1

2. Click [Alt+E]--- [M] in the main menu;

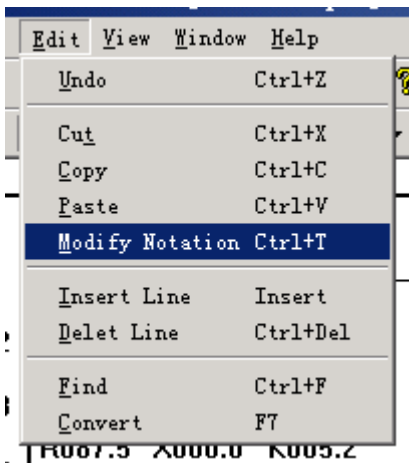


Fig. 3-4-2-2

After the system pops-up the dialog box, the user inputs the annotation and then click OK to save it.

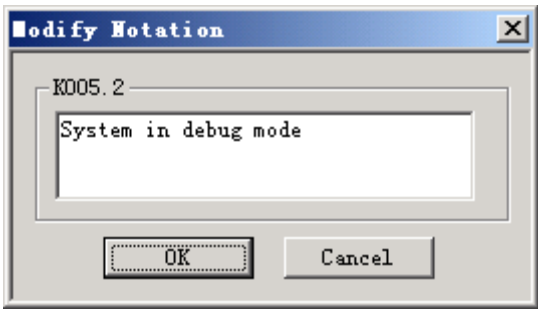


Fig. 3-4-2-3

The saved annotation appears in the output window at the bottom of the screen when the element is selected every time as follows:

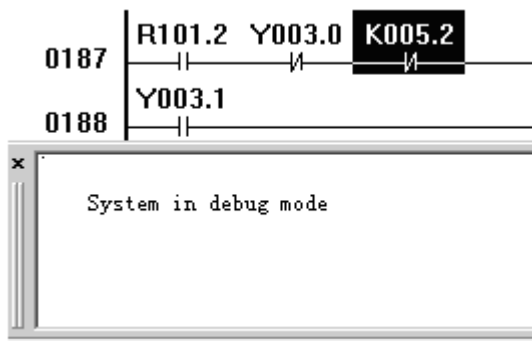


Fig. 3-4-2-4



### 3.5 Exporting

After the ladder is saved, the user should convert it to generate the executable file to send to CNC by the serial communication software. See **System Communication** in **GSK990MA Programming and Operation Manual**.

Generate a ladder file

Click [Alt+F]---[L] ( generate a ladder file )in the main menu, input the name and path to save them, and the system generates ladder file “.grp” which is applied to the milling machine and machining center of GSK990MA series.

Ladder configuration format is showed as appendix two.



## **IV Connection**

# 1 System Structure and Installation

## 1.1 System composition

GSK990MA CNC system mainly consists of the following units as Fig. 1.1.

- (1) GSK990MA CNC system
- (2) Additional operator panel (optional)
- (3) Digital AC servo drive unit (step drive unit)
- (4) Servo motor (stepper motor)
- (5) AC transformer

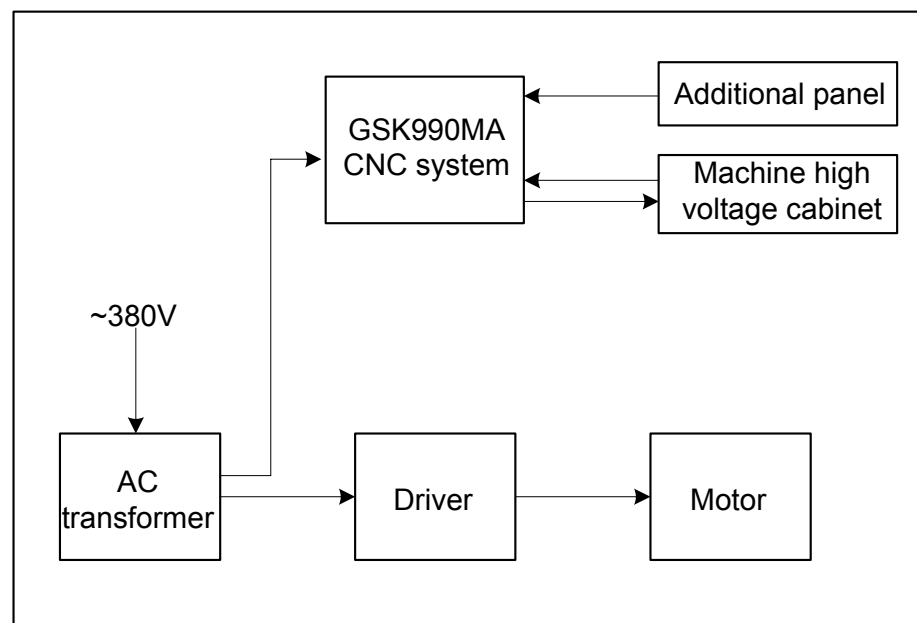


Fig. 1-1

## 1.2 System installation & connection

Firstly, check if the CNC system, driver, motor, and photoelectric encoder are ready, intact and matched.

The CNC system must be fixed stably, and there is some space around the system to ensure the air circulates, and the heat radiates. The installation position of CNC system must be convenient to the operation and avoid the position of processing chip and cooling.

The high/low voltage should be separated. The power supplies of CNC system and driver are provided by transformer, which are separated from the machine high voltage. All kind of signal line should be far from AC contactor to avoid the interference. The photoelectric encoder, limit signal and emergency stop signal should be directly connected to the CNC system. The power supply must be strictly grounded.

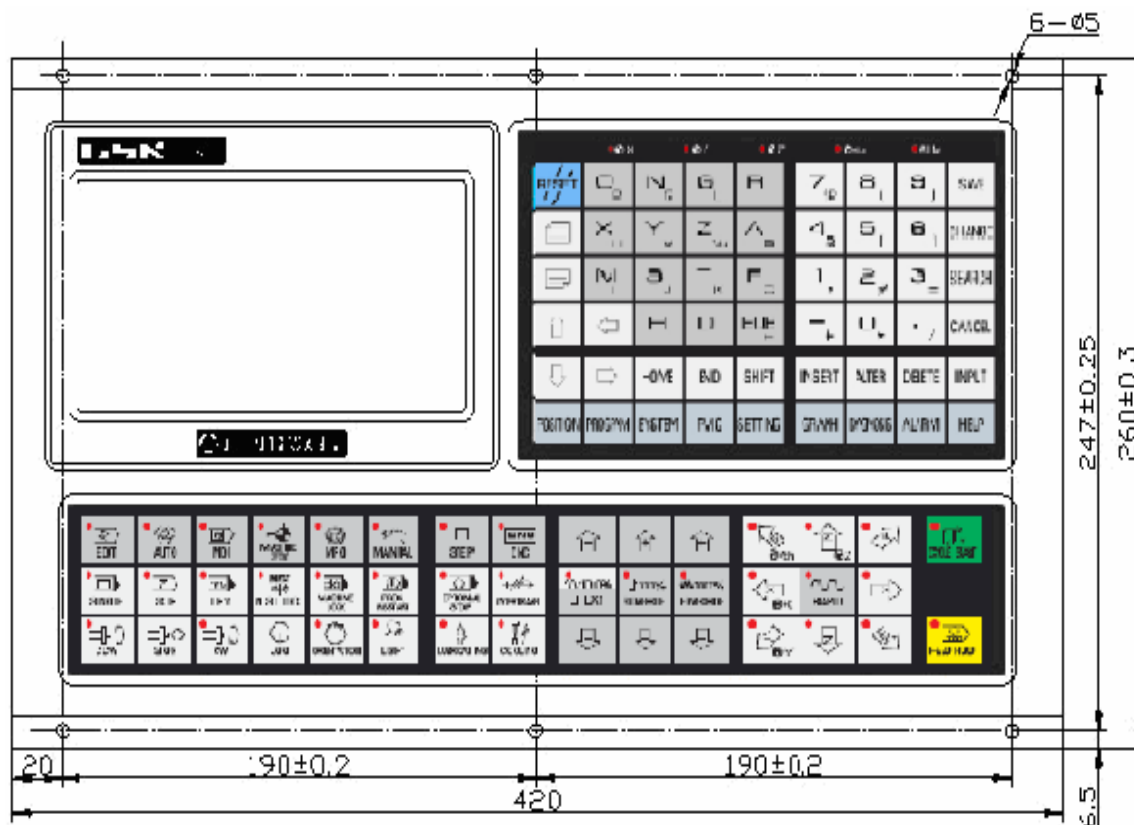
All kind of plug and bolt must be fixed stably, and forbid the signal connector is ON/OFF after the CNC system is turned on.

The system panel cannot be damaged by hard thing and sharp weapon when the CNC system is installed; the CNC system should be carried down to avoid dirtying the system panel.

There is no the source of high voltage, magnetic field around the CNC system, and the system should be far from the inflammable, explosive substance and all sort of dangerous thing.

### 1.3 CNC system installation dimension

### 1. Front view



**Fig. 1-3-1**

## 2. Side view

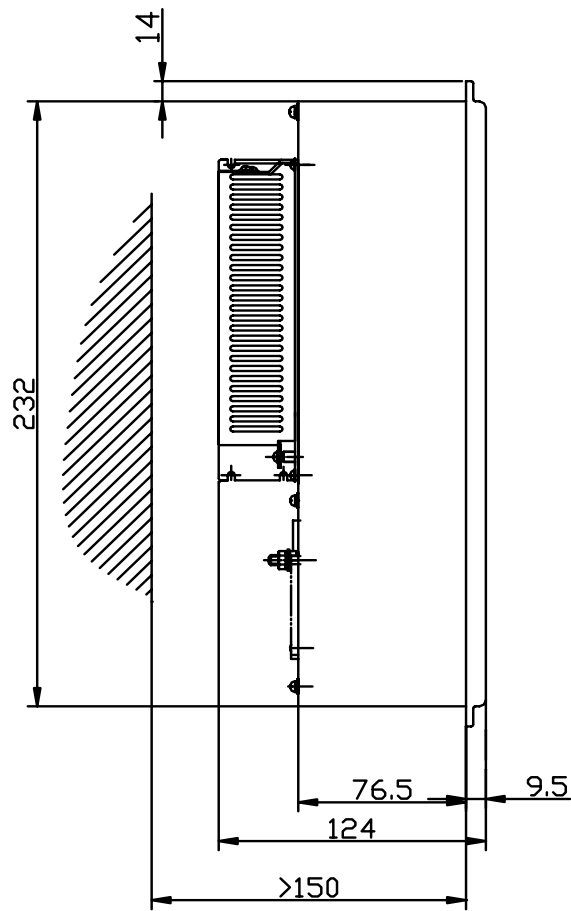


Fig. 1-3-2

## 3. Overlook

( without PC2 power supply )

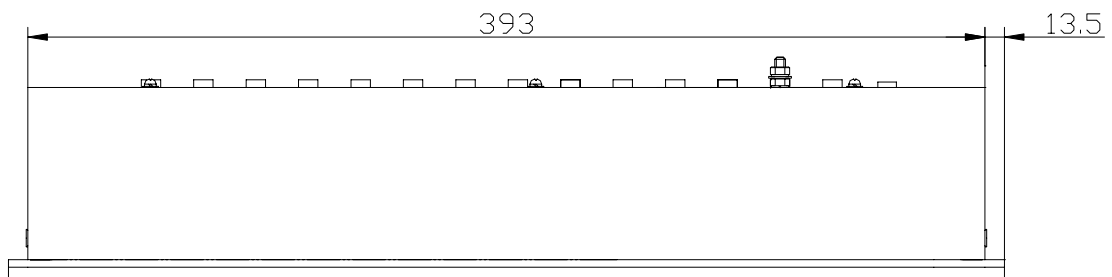


Fig. 1-3-3

#### 4. Back view

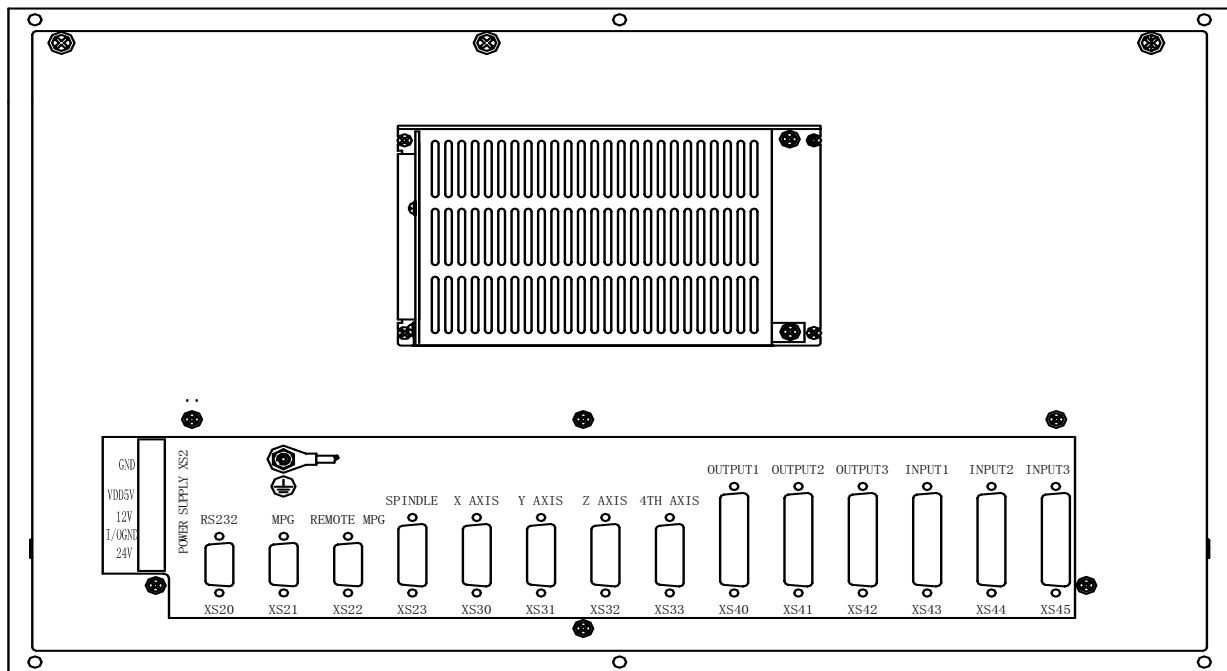


Fig. 1-3-4

#### 5. Additional panel

The user can select the additional panel for the system, and the functions of extension holes on the panel can be defined by the user, such as emergent stop, program lock, power on/off of the system, feed hold, cycle start, MPG and so on. The optional accessories of the system are as follows:

MPG: Changchun LGF-001-100;

Additional panel: (aluminum alloy 420×71mm) can be assembled under of GSK990MA operator panel;

Emergency stop button: LAY3-02ZS/1

No. self-locking button: KH-516-B11 (green or red) ;

Self-locking button: KH-516-B21 (green or red) ;

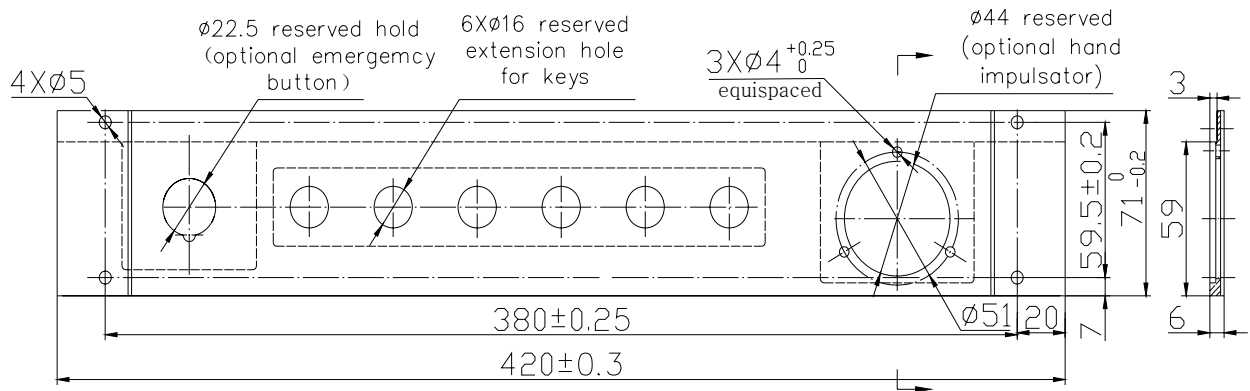


Fig. 1-3-5



2

Device Connection

2.1 CNC external connection

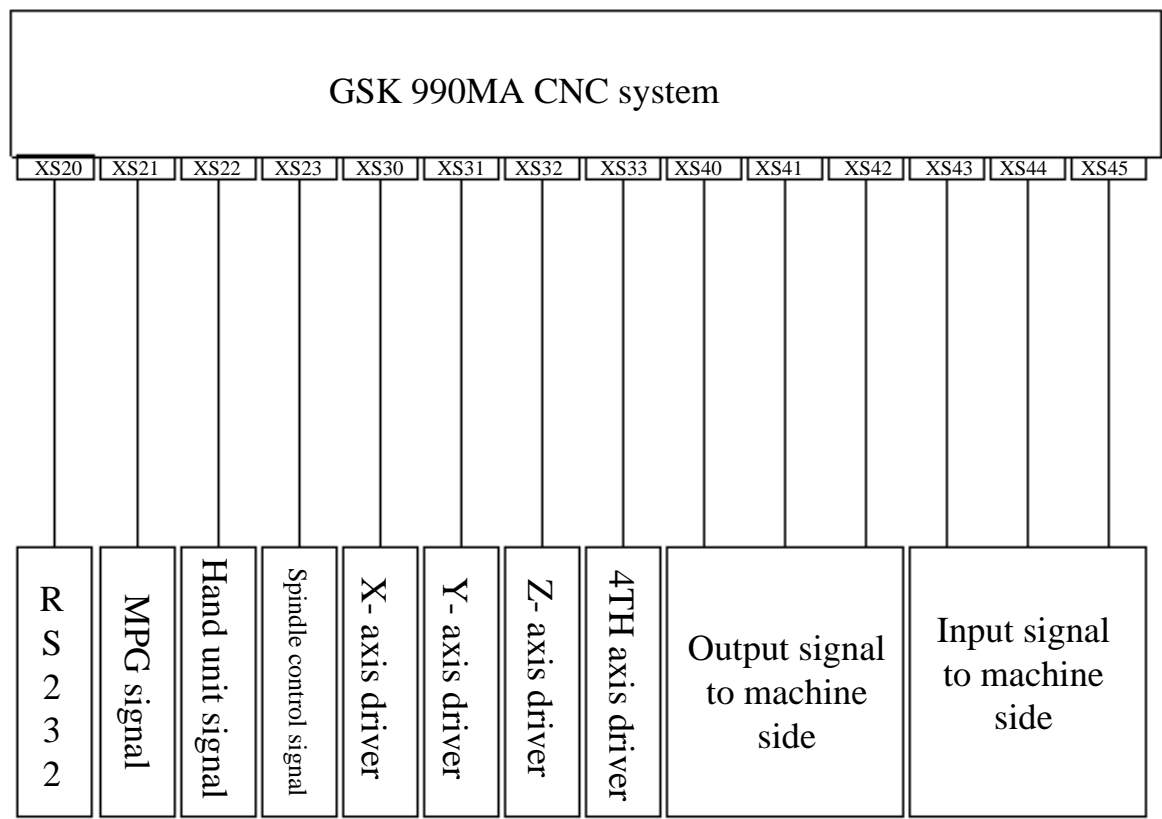


Fig. 2-1-1

2.2 Connection between system and driver

Interfaces to driver include XS30 (X axis) , XS31 (Y axis) , XS32 (Z axis) , XS33 (4<sup>TH</sup> axis) .

## 2.2.1 System interface

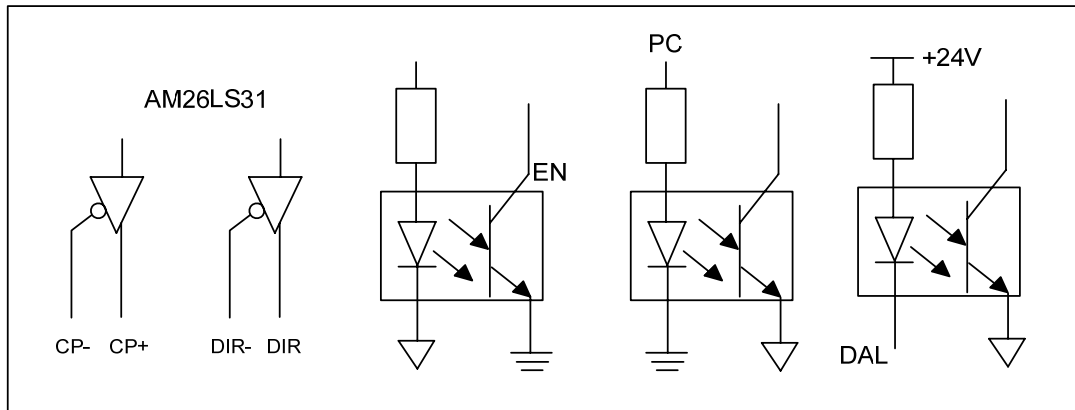


Fig. 2-2-1-1

## 2.2.2 Interface signal list

XS30: DB15 female (X axis)

1	XCP+	9	XCP-
2	XDIR+	10	XDIR-
3	XPC	11	0V
4	+24V	12	+5V
5	XDALM	13	+5V
6		14	0V
7	XEN	15	0V
8	0V		

XS31: DB15 female (Y axis)

1	YCP+	9	YCP-
2	YDIR+	10	YDIR-
3	YPC	11	0V
4	+24V	12	+5V
5	YDALM	13	+5V
6		14	0V
7	YEN	15	0V
8	0V		

XS32: DB15 female (Z axis)

1	ZCP+	9	ZCP-
2	ZDIR+	10	ZDIR-
3	ZPC	11	0V
4	+24V	12	+5V
5	ZDALM	13	+5V
6		14	0V
7	ZEN	15	0V
8	0V		

XS33: DB15 female (4TH axis)

1	4CP+	9	4CP-
2	4DIR+	10	4DIR-
3	4PC	11	0V
4	+24V	12	+5V
5	4DALM	13	+5V
6		14	0V
7	4EN	15	0V
8	0V		

Fig. 2-2-2-1

## 2.2.3 Signal specification

### 1) Pulse motion instruction signals

XCP+, XCP-, YCP+, YCP-, ZCP+, ZCP-, 4CP+, 4CP- are instruction pulse signals, XDIR+, XDIR-, YDIR+, YDIR-, ZDIR+, ZDIR-, 4DIR+, 4DIR- are motion direction signals, and they are differential signals.

Connection is as follows:

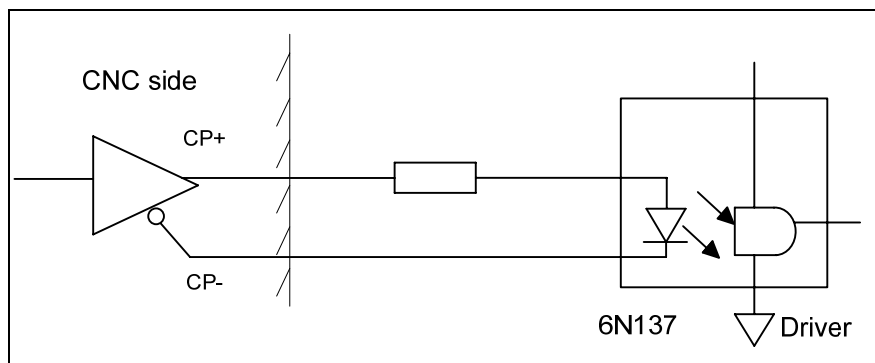


Fig.2-2-3-1

### 2) Driver alarm signal ALM (input)

The receiving method of signal in the CNC side is as follows. The parameter No.19 bit0 sets whether the driver fault is the low level "0" or the high level "1".

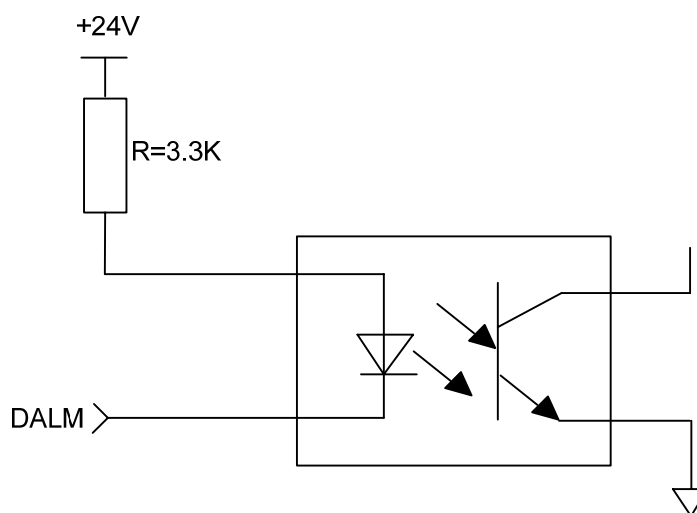
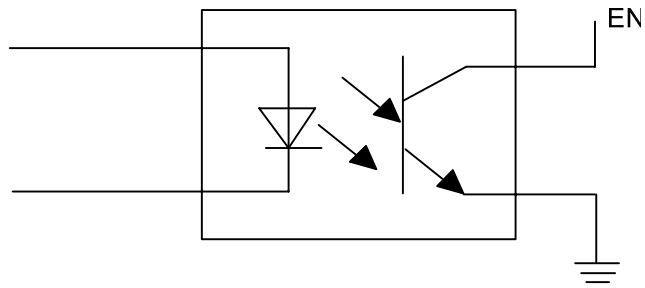


Fig. 2-2-3-2

### 3) CNC ready completion signal EN(contact output) CNC

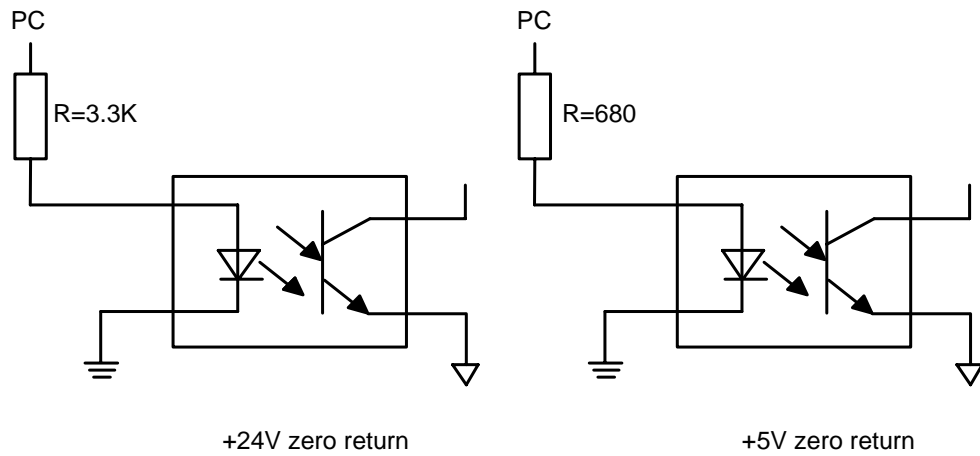
CNC ready has been completed when the contact is closed. When CNC has detected the alarm, the contact signal is OFF.



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

4) Reference point return with signal PC

The receiving method of signal in the CNC side is as follows.



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

Wave of PC signal provided by customer is as follows:

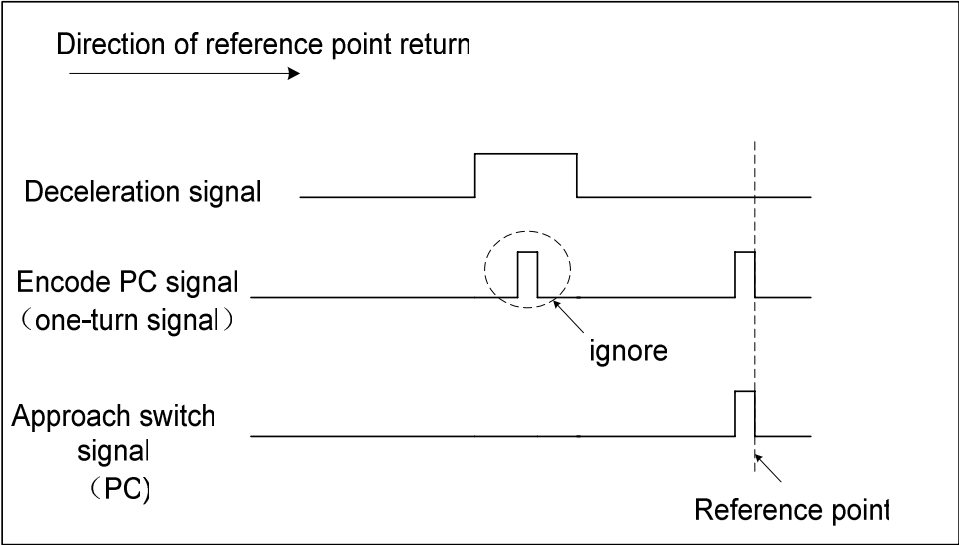


Fig. 2-2-3-5

5) Connection method of one approach switch as the deceleration switch and zero return switch.

2.2.4 Cable connection

1. Cable for 990MA connecting with DY3 series driver

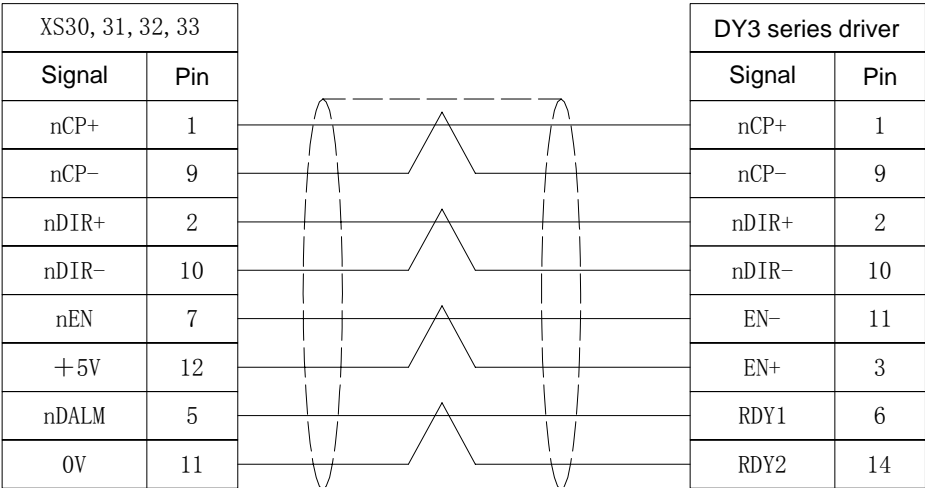


Fig. 2-2-4-1

2. Cable for 990MA connecting with DA98 series servo driver

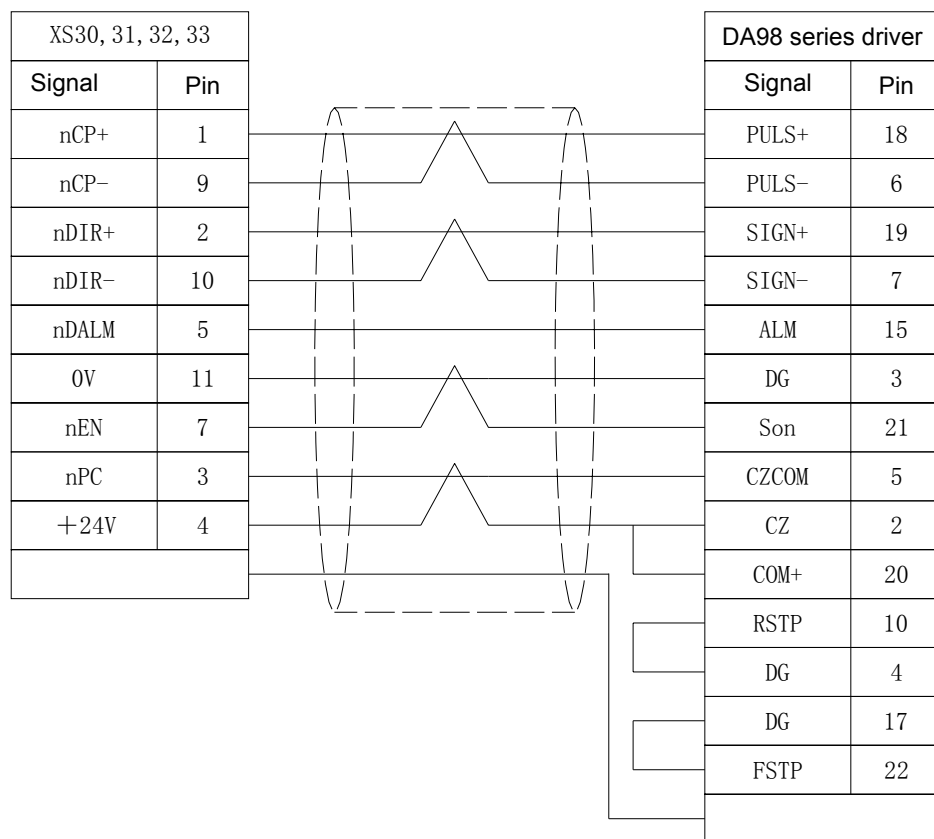


Fig. 2-2-4-2

## 2.3 RS232 standard serial interface

GSK990MA CNC system can communicate with the general-purpose PC (must match with 990MA communication software) by RS232-C. Its connection is as follows:

Connection of cable is as follows:

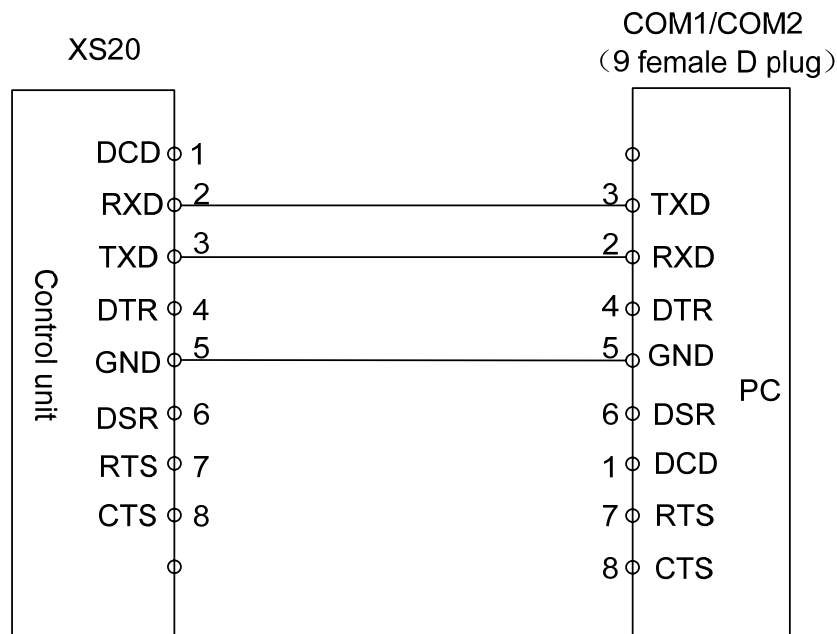


Fig. 2-3-1

2.4 MPG ( handwheel ), hand unit connection

2.4.1 Interface signal list

The 990MA CNC system can be matched with differential and nondifferentiable MPG or hand unit. When it is matched with differential MPG or hand unit, the MPG signal is connected to XS21 interface and the others of the hand unit are connected to XS22 interface;

When it is matched with nondifferentiable, it needs to short-circuit the 1、2、3 pin-outs of XS21 interface, and the other connections remain.

XS21: DB9 male				XS22: DB9 male			
1	+5V	6	VCOM	1	+5V	6	HX
2	HA+	7	HA—	2	HY	7	HZ
3	HB+	8	HB—	3	HU	8	H*1
4	ESP1	9	ESP2	4	H*10	9	H*100
5	0V			5	0V		

Fig. 2-4-1-1

## 2.4.2 Interface signal explanation

HA+, HA−, HB+, HB−: differential MPG or hand unit pulse signal (when match with nondifferentiable MPG or hand unit, HA+, HB+ connects with +5V; HA- connects with A of the MPG, HB- connects with B of the MPG )

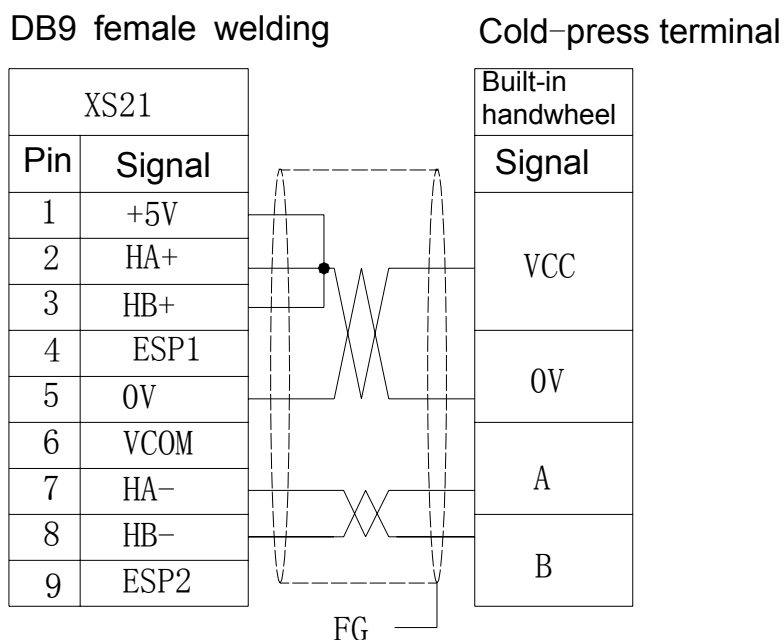
ESP1, ESP2: hand unit emergency stop signal;

HX, HY, HZ, HU: are respectively axis selection signal of X, Y, Z, 4<sup>TH</sup>.

H\*1, H\*10, H\*100: are respectively the override of MPG pulse equivalent;

VCOM: hand unit common terminal.

Connection for GSK990MA to nondifferentiable MPG:



**Fig. 2-4-2-1**

Connection for 990MA to PSG series MPG impulsator:



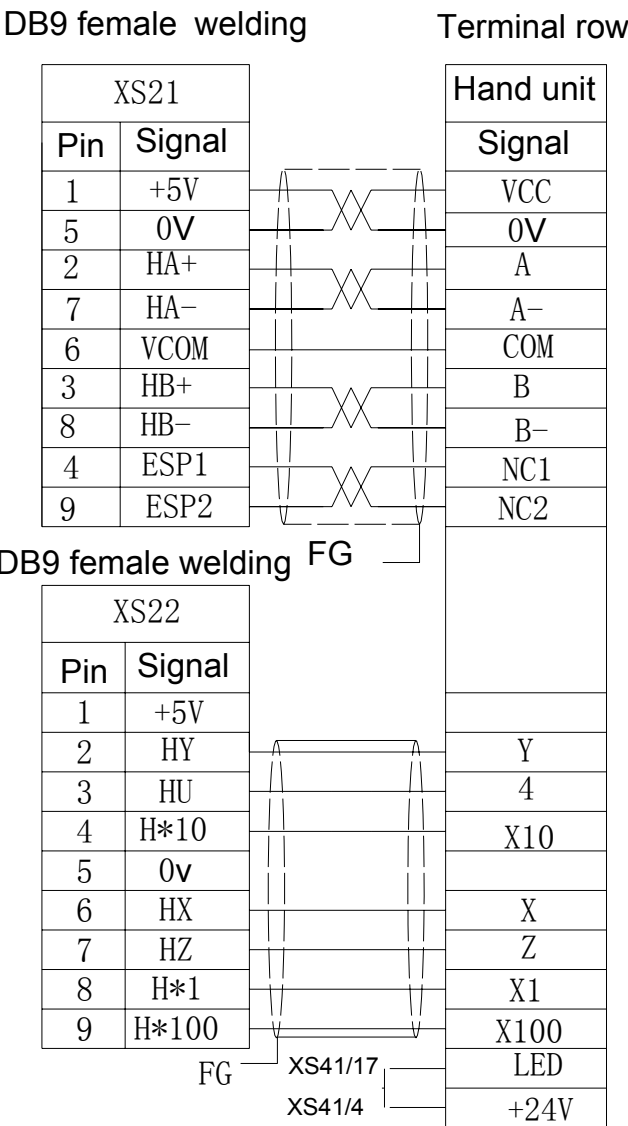


Fig. 2-4-2-2

Connection diagram of GSK990MA with handheld unit in voltage type (E) :

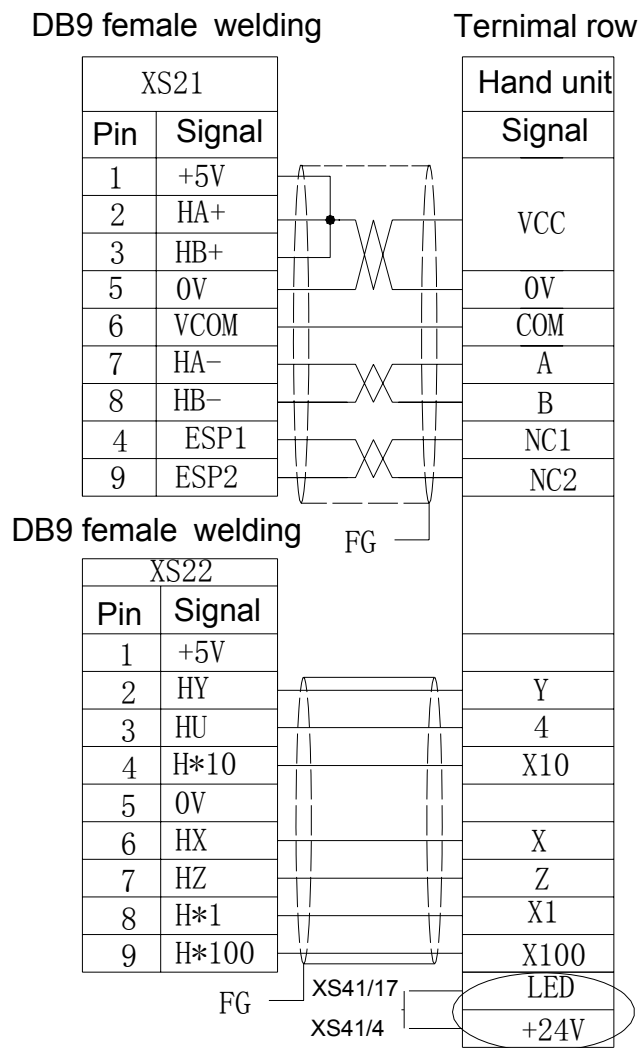


Fig. 2-4-2-3

## 2.5 Spindle unit connection

### 2.5.1 Interface signal list

The CNC interface is DB15 male and its pin definition is as follows:

XS23:DB15 male

1	SCOM	9	SVC
2		10	A+
3	A-	11	B+
4	B-	12	Z+
5	Z-	13	SDALM
6	+5V	14	
7		15	+24V
8	COM		

Fig. 2-5-1-1

## 2.5.2 Interface signal explanation

- (1) A+, A−, B+, B−, Z+, Z−: pulse signal of spindle encode;
- (2) SVC: spindle analog voltage signal;
- (3) SCOM: spindle analog power signal ground;
- (4) SDALM: spindle alarm input signal;

## 2.5.3 DAP01 interface connection

The GSK990MA matches DAP01 control line interface connection :

## Connection for GSK990MA to DAP01

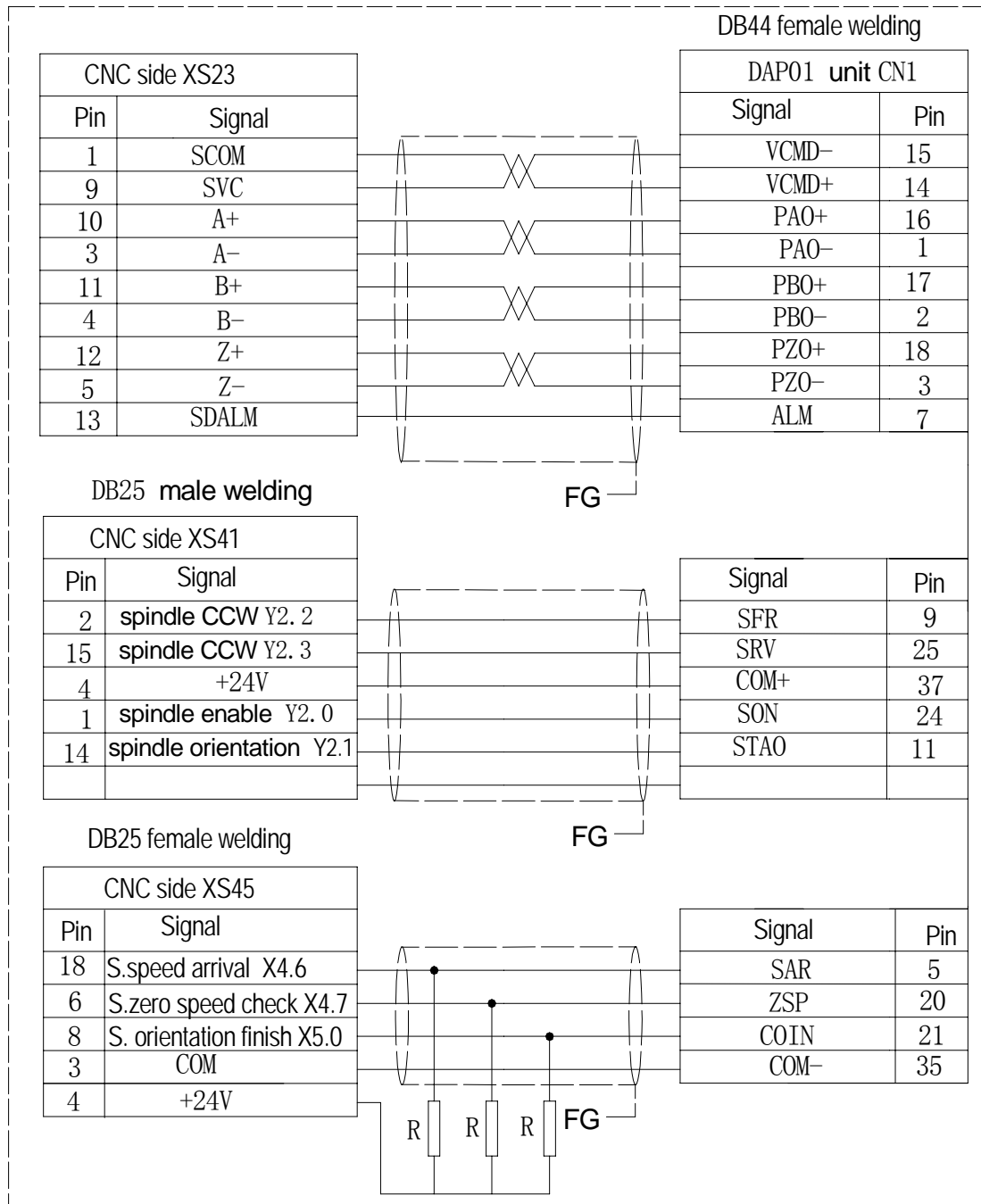


Fig. 2-5-3-1

**Note :** Resistance R is 2.2K, 0.5w.

# 2.5.4 Connection Circuit of DAP03 Interfaces

Interface wiring for GSK990MA CNC system to DAP03 control circuit:

## Connection for GSK990MA to DAP03

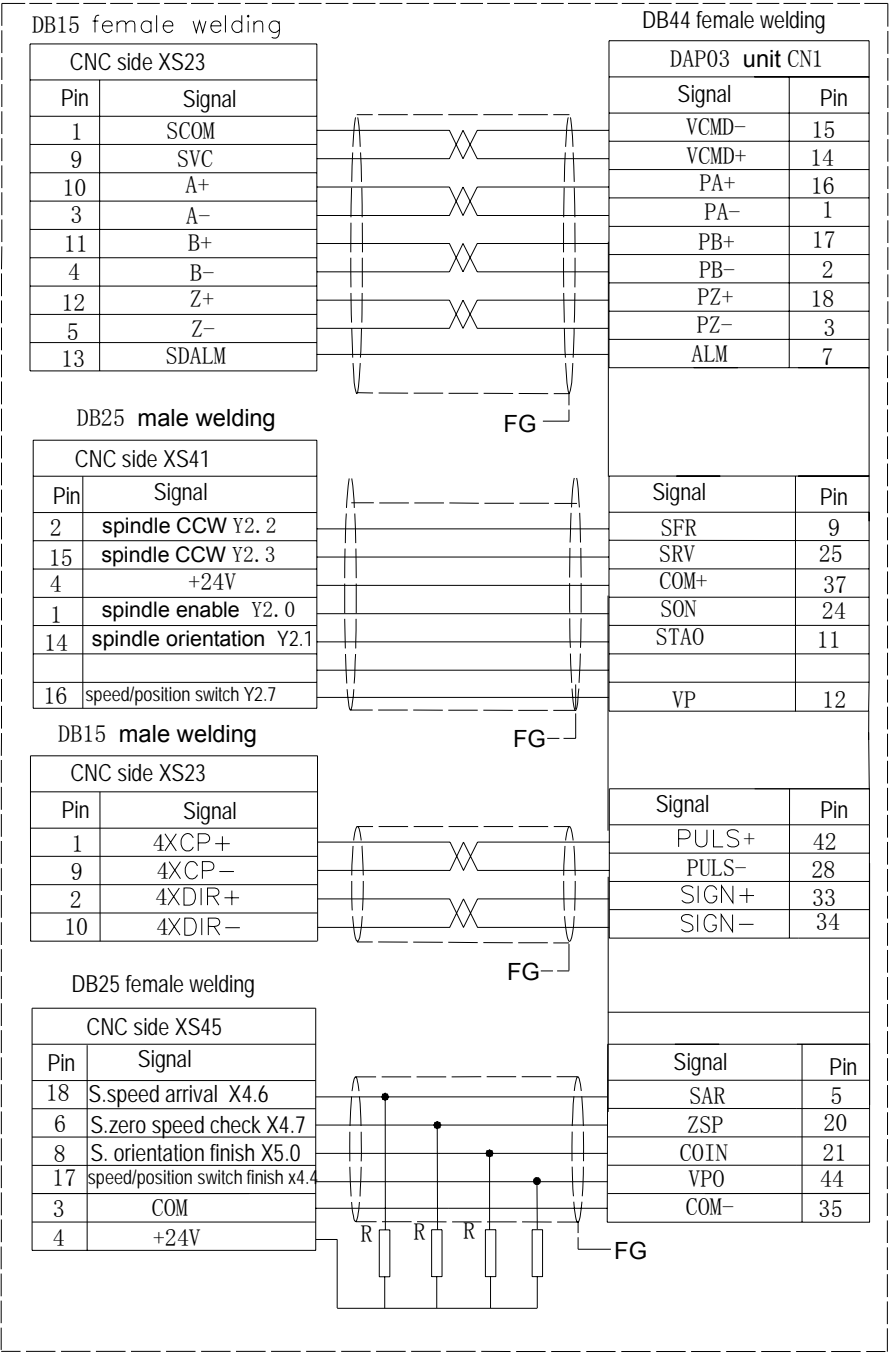


图 2-5-4-1

## 2.6 Power supply interface

The input voltage of the CNC has two groups: +5V, +24V, among which +5V is for the CNC internal system, and +24V for external interface. The power supply interface is as follows:

9	GND2
8	GND2
7	
6	+5V
5	+5V
4	
3	GND1
2	
1	+24V

**Fig. 2-6-1**

# 3 Machine Control I/O Interface

## 3.1 Interface signal list

XS43: DB25 male				XS44: DB25 male				XS45: DB25 male			
1	IN00	14	IN01	1	IN16	14	IN17	1	IN32	14	IN33
2	IN02	15	IN03	2	IN18	15	IN19	2	IN34	15	IN35
3	COM	16	COM	3	COM	16	COM	3	COM	16	COM
4	+24V	17	IN04	4	+24V	17	IN20	4	+24V	17	IN36
5	IN05	18	IN06	5	IN21	18	IN22	5	IN37	18	IN38
6	IN07	19	COM	6	IN23	19	COM	6	IN39	19	COM
7	COM	20	+24V	7	COM	20	+24V	7	COM	20	+24V
8	IN08	21	IN09	8	IN24	21	IN25	8	IN40	21	IN41
9	IN10	22	IN11	9	IN26	22	IN27	9	IN42	22	IN43
10	COM	23	COM	10	COM	23	COM	10	COM	23	COM
11	+24V	24	IN12	11	+24V	24	IN28	11	+24V	24	IN44
12	IN13	25	IN14	12	IN29	25	IN30	12	IN45	25	IN46
13	IN15			13	IN31			13	IN47		

Fig. 3-1

XS40, XS41, XS42 are output interfaces (DB25 female), XS43, XS44, XS45 are input interfaces (DB25 male)

## 3.2 Input interface

### 3.2.1 Input interface principle

DC input signal A is from the machine to the CNC, and they are from the press key in the machine side, limit switch and contact of relay.

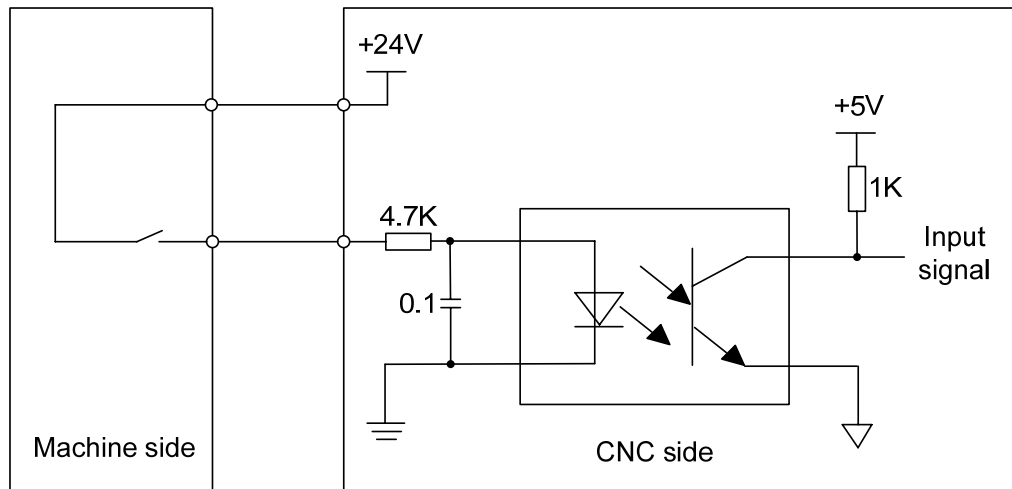
a) Contacts in the machine side should meet the following conditions:

Contact capacity: over DC30V, 16mA.

Leak current between contacts during open circuit: below 1mA(voltage 26.4V

Voltage-drop between contacts during closed-circuit: below 2V(current 8.5mA, including voltage-drop of cable).

b) Signal loop is as Fig 3-2-1-1:



**Fig.3-2-1-1**

### 3.2.2 Input signal interface definition

Pin definition of input interface is as follows:

**Table 3-2-2-1**

Address	Signal interface	Pin	Definition	Contact selection
X000.0	XS43	1	X-axis positive travel limit signal	Normally closed contact
X000.1	XS43	14	X-axis negative travel limit signal	Normally closed contact
X000.2	XS43	2	Y-axis positive travel limit signal	Normally closed contact
X000.3	XS43	15	Y-axis negative travel limit signal	Normally closed contact
X000.4	XS43	17	Z-axis positive travel limit signal	Normally closed contact
X000.5	XS43	5	Z-axis negative travel limit signal	Normally closed contact
X000.6	XS43	18	4TH-axis positive travel limit signal	Normally closed contact
X000.7	XS43	6	4TH-axis negative travel limit signal	Normally closed contact
X001.0	XS43	8	X-axis zero return deceleration signal	Normally closed contact
X001.1	XS43	21	Y-axis zero return deceleration signal	Normally closed contact
X001.2	XS43	9	Z-axis zero return deceleration signal	Normally closed contact
X001.3	XS43	22	4TH-axis zero return	Normally closed contact



			deceleration signal	
X001.4	XS43	24	Emergency stop switch	Normally closed contact
X001.5	XS43	12	External cycle start	Normally open contact
X001.6	XS43	25	External feed hold	Normally open contact
X001.7	XS43	13	Lubrication pressure or oil level check	Normally open contact
X002.0	XS44	1	Air supply pressure check	Normally open contact
X002.1	XS44	14	Undefined	
X002.2	XS44	2	Undefined	
X002.3	XS44	15	Undefined	
X002.4	XS44	17	External clamping / releasing tool control	Normally open contact
X002.5	XS44	5	Releasing tool check	Normally open contact
X002.6	XS44	18	Clamping tool check	Normally open contact
X002.7	XS44	6	Spindle tool check switch	Defined by parameter
X003.0	XS44	8	Edit lock	Normally open contact
X003.1	XS44	21	Operation lock	Normally open contact
X003.2	XS44	9	Undefined	
X003.3	XS44	22	Undefined	
X003.4	XS44	24	Undefined	
X003.5	XS44	12	Undefined	
X003.6	XS44	25	Undefined	
X003.7	XS44	13	Undefined	
X004.0	XS45	1	Undefined	
X004.1	XS45	14	Spindle gear 1 in-position	Normally open contact
X004.2	XS45	2	Spindle gear 2 in-position	Normally open contact
X004.3	XS45	15	Spindle gear 3 in-position	Normally open contact
X004.4	XS45	17	Spindle speed/position state output	Defined by parameter
X004.5	XS45	5	Undefined	
X004.6	XS45	18	Spindle speed/position arrival	Defined by parameter
X004.7	XS45	6	Spindle zero speed check	Normally closed contact
X005.0	XS45	8	spindle orientation in-position	Defined by parameter
X005.1	XS45	21	Tool magazine forward in-position	Defined by parameter
X005.2	XS45	9	tool magazine backward in-position	Defined by parameter
X005.3	XS45	22	Tool magazine CCW/CW in-position	Defined by parameter
X005.4	XS45	24	Tool magazine zero return in-position	Defined by parameter
X005.5	XS45	12	Current tool magazine count check switch	Defined by parameter
X005.6	XS45	25	Undefined	
X005.7	XS45	13	Undefined	
X006.0	XS22	6	External MPG X axis selection	Normally open contact
X006.1	XS22	2	External MPG Y axis selection	Normally open contact
X006.2	XS22	7	External MPG Z axis selection	Normally open contact
X006.3	XS22	3	External MPG A axis selection	Normally open contact
X006.4	XS22	8	External MPG step 0. 001	Normally open contact
X006.5	XS22	4	External MPG step 0. 01	Normally open contact
X006.6	XS22	9	External MPG step 0. 1	Normally open contact

X006.7	XS21	ESP (4, 9)	External emergency stop	Normally closed contact
--------	------	------------------	-------------------------	-------------------------

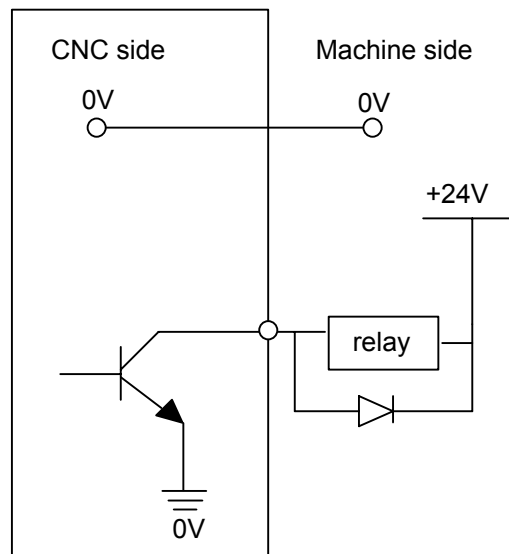
### 3.3 Output signal

#### 3.3.1 Output interface principle

a) Output transistor specification:

- ① When the output is ON, max. load current, including instantaneous current is below 200mA.
- ② When the output is ON and the current is 200mA, the saturation voltage is 1.6V and the typical value is 1V .
- ③ When the output is OFF, the withstand voltage including instantaneous voltage is below 24+20%.
- ④ When the output is OFF, the leak current is below 100μA.

b) Output loop:



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

The output signals of the CNC are provided by Darlington pipe and Darlington pipe is conducted when the output is enabled. Except for TL-, TL+, SPZD are pulse signals( ), other outputs are the level( output keep) signal and their common terminal is +24V.

### 3.3.2 Output signal interface definition

Table 3-3-2-1

Address	Signal interface	Pin	Definition
Y000.0	XS40	1	Z axis holding brake
Y000.1	XS40	14	Cooling
Y000.2	XS40	2	Tool releasing/clamping
Y000.3	XS40	15	Undefined
Y000.4	XS40	17	Spindle brake
Y000.5	XS40	5	Undefined
Y000.6	XS40	18	Red alarm light
Y000.7	XS40	6	Yellow alarm light
Y001.0	XS40	8	Green alarm light
Y001.1	XS40	21	Chip removal control
Y001.2	XS40	9	Lubrication control
Y001.3	XS40	22	Machine light control
Y001.4	XS40	24	Undefined
Y001.5	XS40	12	Spindle blow
Y001.6	XS40	25	Undefined
Y001.7	XS40	13	Undefined
Y002.0	XS41	1	Spindle enabling
Y002.1	XS41	14	Spindle orientation
Y002.2	XS41	2	Spindle CCW
Y002.3	XS41	15	Spindle CW
Y002.4	XS41	17	Hand unit light
Y002.5	XS41	5	Undefined
Y002.6	XS41	18	Chip wash hydrovalve output
Y002.7	XS41	6	Spindle position/speed modal change
Y003.0	XS41	8	Tool magazine CCW
Y003.1	XS41	21	Tool magazine CW
Y003.2	XS41	9	Tool magazine forward
Y003.3	XS41	22	Tool magazine backward
Y003.4	XS41	24	Spindle gear 1(frequency conversion\IO point control)
Y003.5	XS41	12	Spindle gear 2(frequency conversion\IO point control)
Y003.6	XS41	25	Spindle gear 3(frequency

			conversion\IO point control)
Y003.7	XS41	13	Spindle gear 4(frequency conversion\IO point control))
Y004.0	XS42	1	Undefined
Y004.1	XS42	14	Undefined
Y004.2	XS42	2	Undefined
Y004.3	XS42	15	Undefined
Y004.4	XS42	17	Undefined
Y004.5	XS42	5	Undefined
Y004.6	XS42	18	Undefined
Y004.7	XS42	6	Undefined
Y005.0	XS42	8	Undefined
Y005.1	XS42	21	Undefined
Y005.2	XS42	9	Undefined
Y005.3	XS42	22	Undefined
Y005.4	XS42	24	Undefined
Y005.5	XS42	12	Undefined
Y005.6	XS42	25	Undefined
Y005.7	XS42	13	Undefined

# 4

## Debugging Machine

The chapter introduces the installation and trial run operation methods and steps after GSK990MA CNC system is turned on firstly, the corresponding machine operation can be performed after the following steps are done.

### 4.1 Debugging preparation

GSK990MA debugging is as follows:

- System connection: the correct connection is the basis to successfully debug the system.
- PLC debug: it makes the safety functions(emergency stop, hardware limit) and operation functions be effective.
- Parameter setting of driver: set the motor type and control mode.
- Parameter setting of system: set the control parameter, speed parameter and so on.
- Data backup: after the system is debugged, the data including the parameter, the compensation data and PLC program is backup.

Notes before debugging GSK990MA:

- To ensure all cables are connected correctly, please check the polarity of diode of relay, electromagnetic valve.
- Check the connection phase sequence of cable with high voltage of motor.
- The position cable, encoder feedback cable and motor cable with high voltage of AC servo feed device correspond one by one.
- Ensure the analog voltage instruction type received by the spindle.
- Ensure all grounding are stably connected.
- Ensure the emergency stop button and emergency stop circuit are valid. When the emergency stop button or emergency stop circuit is turned off, the power supply of drive device, spindle drive device can be turned off.
- Ensure the voltage and the polarity are correct.
- Ensure the specifications of power supply are correct.
- Ensure the specifications and the inlet/outlet directions of transformer are correct.
- Ensure the inlet/outlet lines of power supply of breakers are correct.

## 4.2 System power on

- Press the emergency stop button to ensure all air switches in the CNC are turned off.
- The power air switch in the electric cabinet is turned on.
- The air switch or the fuse is connected with DC 24V, and ensure DC24V is normal.
- Ensure other power supplies are normal.
- GSK990MA CNC device is turned on.

## 4.3 Emergency stop and limit

The CNC has the soft limit function, it should be also adopted with the hardware limit function to get the safe operation, and the travel limit switch should be fixed in the positive/negative direction of axis.

In 【MACHINE SIDE】 screen of <DIAGNOSIS> window, the user can monitor and view the state of the emergency stop signal by viewing NO:1#4 (\*ESP) , it is required that all the air switches must be turned off after pressing emergency stop button.

In Manual or MPG (handwheel) mode, the CNC can verify the validity of each axis overtravel limit switch, the accuracy of alarm display, and the validity of overtravel release button by slow moving each coordinate axis; when the overtravel occurs or the emergency stop is pressed, the CNC alarms, which can be released by pressing the overtravel releasing to execute the reverse move.

- Emergency stop signal

\*ESP: Emergency stop signal

Parameter diagnosis (CNC side input state)

state		X1.4						
address								
Pin		XS43.24						

In order to properly give an alarm of "an axis (X, Y or Z axis) is on a moving direction (positive or negative)" in case of servo axis overtravel, as well as to ensure the axis that has an overtravel

alarm not move to the overtravel direction continuously with the overtravel releasing, this GSK990MA CNC system provides two connection methods for the travel limit switch for the user:

**A. when has two travel limit switch:**

(i.e. an axis positive direction limit use a travel switch, negative direction use a travel switch )

**1. Connecting by the following table**

**Table 4-3-1**

Address	Signal Interface	Pin	Definition	Contact Selection
X000.0	XS43	1	X axis positive travel limit signal	Normally-closed contact
X000.1	XS43	14	X axis negative travel limit signal	Normally-closed contact
X000.2	XS43	2	Y axis positive travel limit signal	Normally-closed contact
X000.3	XS43	15	Y axis negative travel limit signal	Normally-closed contact
X000.4	XS43	17	Z axis positive travel limit signal	Normally-closed contact
X000.5	XS43	5	Z axis negative travel limit signal	Normally-closed contact

**2. Modify the following parameter:**

**Table 4-3-2**

Address	Definition	State 0	State 1	Setting Value
K006.0	X axis limit switch selection	2	1	0
K006.1	Y axis limit switch selection	2	1	0
K006.2	Z axis limit switch selection	2	1	0

**B. When has one travel limit switch:**

( an axis positive/negative direction limit use a same travel switch )

**1. Connecting by the following table**

**Table 4-3-3**

Address	Signal Interface	Pin	Definition	Contact Selection
X000.0	XS43	1	X axis travel limit signal	Normally-closed contact
X000.2	XS43	2	Y axis travel limit signal	Normally-closed contact
X000.4	XS43	17	Z axis travel limit signal	Normally-closed contact

**2. Modify the following parameter:**

**Table 4-3-4**

Address	Definition	State 0	State 1	Setting Value
K006.0	X axis limit switch selection	2	1	1
K006.1	Y axis limit switch selection	2	1	1
K006.2	Z axis limit switch selection	2	1	1

### 3. Cautions:

The alarm direction may be inconsistent with the actual one owing to using a stroke switch, (example: when X axis moves positively, alarm " 0510 hardware limit overtravel: -X" occurs, but the actual alarm is " 0510 hardware limit overtravel: +X."), which can be adjusted by the following parameters.

**Table 4-3-5**

Address	Definition	State 0	State 1	Setting Value
K007.0	X axis limit alarm direction reverses			
K007..1	Y axis limit alarm direction reverses			
K007.2	Z axis limit alarm direction reverses			

Note: when there is no stroke switch with the machine tool, please short limit switch signal to +24V of system.

State parameter

<b>0</b>	<b>1</b>	<b>1</b>	<b>BFA</b>	<b>LZR</b>						
----------	----------	----------	------------	------------	--	--	--	--	--	--

**LZR** =1: The travel check is executed during the period from power-on time to the completion of the manual reference point return.

=0: The travel check is not executed during the period from power-on time to the completion of the manual reference point return.

**BFA** =1: The CNC alarms after overtravel when it transmits the overtravel instruction.

=0: The CNC alarms before overtravel when it transmits the overtravel instruction.

System parameter number

<b>0</b>	<b>3</b>	<b>1</b>	<b>G13</b>							
----------	----------	----------	------------	--	--	--	--	--	--	--

**G13** =1: The system is executed by G13 when it is turned on or clearing.

=0: The system is executed by G12 when it is turned on or clearing.

System parameter number

<b>0</b>	<b>6</b>	<b>1</b>	<b>LALM</b>							
----------	----------	----------	-------------	--	--	--	--	--	--	--

**LALM** =1: Ignore the limit alarm.

=0: Do not ignore the limit alarm.

## 4.4 Gear ratio adjustment

When the machine move distance is not uniform with the displacement distance of coordinate



display, NO.160~ NO.169 are modified to adjust the electronic gear ratio to meet to the different machine driving ratio.

Division/multiplying of positioning INSTRUCTION pulse (electronic handwheel/MPG).

In position control mode, it can match with all pulse source by setting parameters to get the required resolution (angle/pulse).

**Computation formula:**

$$G = \frac{\text{Frequency division numerator}}{\text{Frequency division denominator}} = \frac{4C}{L/\zeta} \times \frac{Z_M}{Z_D}$$

G: Electronic gear ratio;

L : Lead screw lead

$\zeta$  : System min. output instruction unit (mm/pulse)

C : Photoelectric encoder pulse/rev, the system C=2500.

ZM : Gear teeth quantity of lead screw }  
ZD : Geat teeth number of motor } When has a change gear

CNC side:

Frequency division numerator : parameter NO.160、NO.161、NO.162、NO.163、NO.164 (instruction override coefficient)

Frequency division denominator: system parameter NO.165、NO.166、NO.167、NO.168、NO.169 (instruction frequency division coefficient)

Digital servo side:

Frequency division numerator: Parameter PA12 (instruction override coefficient)

Frequency division denominator: Parameter PA13 (instruction frequency division coefficient)

【Example 1】 If lead screw lead is 8mm, system min. input instruction unit is 0.001mm, and motor encoder pulse is 2500, then:

$$G = \frac{4C}{L/\zeta} \times \frac{Z_M}{Z_D} = \frac{4 \times 2500}{8/0.001} \times \frac{1}{1} = \frac{5}{4}$$

Data parameter NO. 160 (CMRX) =5, NO.165 (CMDX) =4;

The ratio between the system gear and the parameter has the same function that of digital servo gear and parameter. When the system is employed with the digital servo with the electronic gear ratio function, the electronic gear ratio is set to 1:1, which is set to the digital servo.

When the CNC is adopted with the stepper driver, it should use the stepper subdivision driver as possible, and the proper machine driving ratio, and set the electronic gear ratio to 1:1 to avoid the large difference between the numerator and denominator.

【Example 2】 Gear ratio computation formula of rotation axis is as follow:

$$G = \frac{N \times C \times 4}{P} = \frac{1 \times 2500 \times 4}{360 \times 1000 \times \text{deceleration ratio (drive gear/driven gear)}}$$

## 4.5 Backlash compensation

Use the gauge, micrometer gauge or laser master gauge to measure the backlash, the backlash compensation must be precise, otherwise it cannot improve the precision of processing and it is suggested that the following methods should be adopted to measure the leading screw backlash instead of MPG (handwheel) or single step:

- Edit program:

```
O0001;
N10 G01 G91 X1 F800 ;
N20 X1 ;
N30 X1 ;
N40 X-1 ;
N50 M30 .
```

- The backlash error compensation value is set to zero before measuring.
- The program runs in Single block mode, and the CNC looks for the measure datum point A after positioning two times, the program runs 1mm and reversely runs 1mm to point B, and the CNC reads the current data.

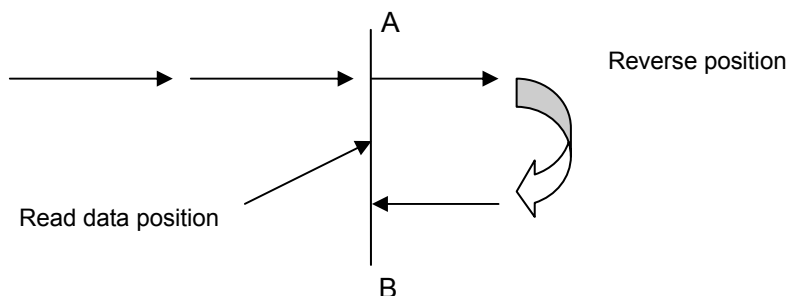


Fig. 4-5-1 backlash measure method

- Backlash error compensation value= |data recorded by A point –data recorded by B point|;

input the operation result to the corresponding system parameter.

Data A: read the data of gauge at the A point;

Data B: read the data of gauge at the B point;

Pulse equivalent: 1 micron

Note:

1. The backlash compensation value of each axis is set by data parameter **P190~P199**.
2. The data parameter **P195~P199** can set the backlash compensation mode and the compensation frequency.
3. To ensure the machine precision, the backlash must be checked after the machine has been used for 3 months.

System parameter number

0	1	8				RBK				
---	---	---	--	--	--	-----	--	--	--	--

**RBK** =1: cutting and rapid respectively perform backlash compensation

=0: cutting and rapid do not respectively perform backlash compensation

## 4.6 Parameter of servo

If the machine moving direction is inconsistent with the one required by the bit instruction, it can be adjusted by modifying the system parameter NO:3#1~ NO:3#5.

System parameter number

0	0	3				DIR4	DIRZ	DIRY	DIRX	INM
---	---	---	--	--	--	------	------	------	------	-----

**DIRX** =1: X axis feeds direction

=0: X axis feeds direction reverse

**DIRY** =1: Y axis feeds direction

=0: Y axis feeds direction reverse

**DIRZ** =1: Z axis feeds direction

=0: Z axis feeds direction reverse

**DIR4** =1: 4TH axis feeds direction

=0: 4TH axis feeds direction reverse

Check the driver alarm or driver connection if an alarm of X, Y, Z axis, or spindle alarm is shown at system power on. If there is no such case, the cause may be that the level set by the system alarm parameter is not matched with the driver alarm level, which may be modified by bit

parameter No. 19#0 ~ NO:19#5 for high or low level alarm active. When matching with our driver, the bits #0 ~ #5 of bit parameter NO:19 should be set to 0. After the parameter is modified, the alarm can be cancelled by pressing the "RESET" key. For security, please set the system parameter switch to "OFF".

If there is no drive alarm provided by the driver, please don't connect that signal, and set the bits #0 ~ #5 of bit parameter NO:19 to 1. If the driver alarm is shown by the system, make a judgment where the fault might be, at CNC side or at driver side.

System parameter number

0	1	9			ALMS	ALMS5	ALMS4	ALMSZ	ALMSY	ALMSX
---	---	---	--	--	------	-------	-------	-------	-------	-------

**ALMX** =1: driver alarms, high level is valid.

=0: driver alarms, low level is valid

**ALMY** =1: driver alarms, high level is valid.

=0: driver alarms, low level is valid

**ALMZ** =1: driver alarms, high level is valid.

=0: driver alarms, low level is valid

**ALM4** =1: driver alarms, high level is valid

=0: driver alarms, low level is valid

**ALM5** =1: driver alarms, high level is valid

=0: driver alarms, low level is valid

**ALMS** =1: driver alarms, high level is valid

=0: driver alarms, low level is valid

System parameter number

0	6	1	FALM			SALM				
---	---	---	------	--	--	------	--	--	--	--

**SALM** =1: ignore the spindle driver alarm

=0: do not ignore the spindle driver alarm

**FALM** =1: ignore the feed spindle driver alarm.

=0: do not ignore the feed spindle driver alarm.

## 4.7 Machine pitch compensation

- Compensation value Setting

1. The set compensation value is related to the position between the zero and compensation point machine move direction and compensation backlash and so on.
2. The compensation value of compensation point N(N=0,1,2,3,...127) is determined by the machine error between N, N-1.
3. The machine zero is taken as the compensation origin point, and the set compensation of every axis is taken as the parameter value.
4. Compensable axis: X, Y, Z, 4 axis. Compensation points: 128 points for each axis.
5. Compensation value range: each compensation point (-7~+7) × compensation override. The input exceeding -7~+7 is disabled.
6. The setting method is the same as that of input method of system parameter, see Operation.

- Notes for compensation value setting

1. Bit parameter NO: 37#1 determines whether the pitch compensation is executed, and NO: 37#2 determines to select the unidirectional or bidirectional compensation.
2. Data parameter P216~P220: Pitch error compensation number of reference point of each axis (setting of compensation zero).
3. Data parameter P221~P225: Compensation points of pitch error compensation of each axis
4. System parameter P226~P230: Pitch error compensation backlash of each axis. Execute the compensation with the value when the positive compensation value is input; execute the compensation with the absolute value of the value when the negative compensation value is input
5. System parameter P231~P235: pitch error compensation override of each axis. The CNC defaults 0.001.
6. The compensation is not executed when the input compensation backlash is zero.
7. After the pitch error parameter is set, the CNC is turned on again, and the pitch error parameter is enabled after the machine zero return is executed.

- The following is the compensation parameters, taking X axis as example:

Table 4-7-1

Parameter	Setting value
<b>P216:</b> compensation number of X-axis reference point	127
<b>P221:</b> X-axis pitch error compensation points	128
<b>P226:</b> X-axis pitch error compensation backlash	10
<b>P231:</b> X-axis pitch error compensation override	0.001

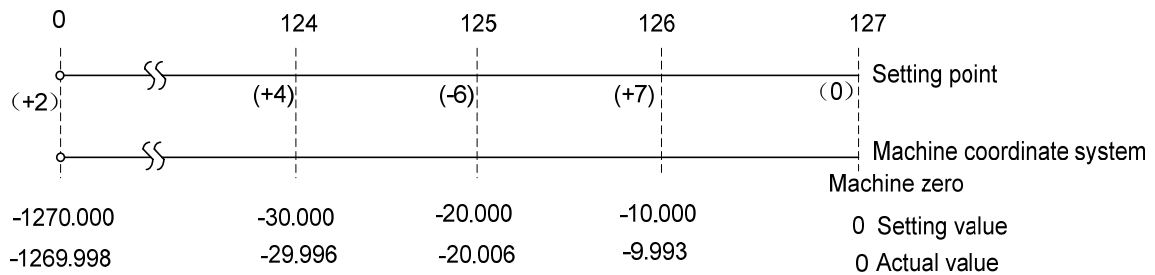


Fig. 4-7-1

The set compensation value in 【Pitch compensation X】.

Table 4-7-2

Number	0	...	...	124	125	126	127
Compensation value	+2	...	...	+4	-6	+7	0

The error compensation takes the machine zero as reference point. The pitch error compensation is executed when the negative coordinate system of machine zero is moving.

Table 4-7-3

Parameter	Setting value
<b>P216:</b> compensation number of X axis reference point	0
<b>P221:</b> X-axis pitch error compensation points	128
<b>P226:</b> X-axis pitch error compensation backlash	10
<b>P231:</b> X-axis pitch error compensation override	0.001

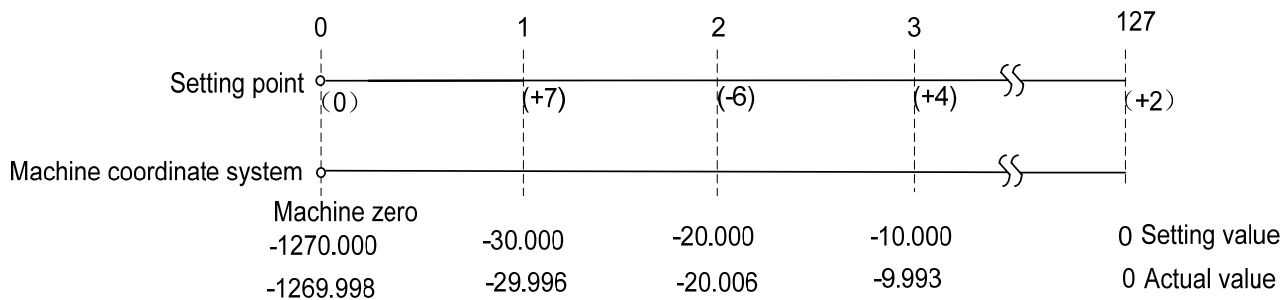


Fig. 4-7-2

The set compensation value in 【Pitch compensation X】.

Table 4-7-4

Number	0	1	2	3	...	...	127
Compensation value	0	+7	-6	+4	...	...	+2

The setting method of the Y and Z axis is the same as above

- **Bidirectional compensation setting takes X axis as the example:**

The error compensation takes the machine zero as the reference point. The pitch error compensation can be executed when the positive/negative coordinate system of machine zero moves.

**Operation in the positive coordinate:** the first compensation length is set in No. 000 in 【Compensation X】, the second is in No. 001 and the N is in No. N-1.

**Operation in the negative coordinate:** the first compensation length is set in No. 1000 in 【Compensation X】, the second is in No. 1001 and the N is in No. 1000+N-1.

Table 4-7-5

Parameter	Setting value
<b>P216:</b> compensation number of X-axis reference point	40
<b>NO.221:</b> X-axis pitch error compensation points	128
<b>NO.226:</b> X-axis pitch error compensation backlash	50
<b>NO.231:</b> X-axis pitch error compensation override	0.001mm

Output the compensation value at the compensation point of corresponding area. The example for the compensation is as follows:

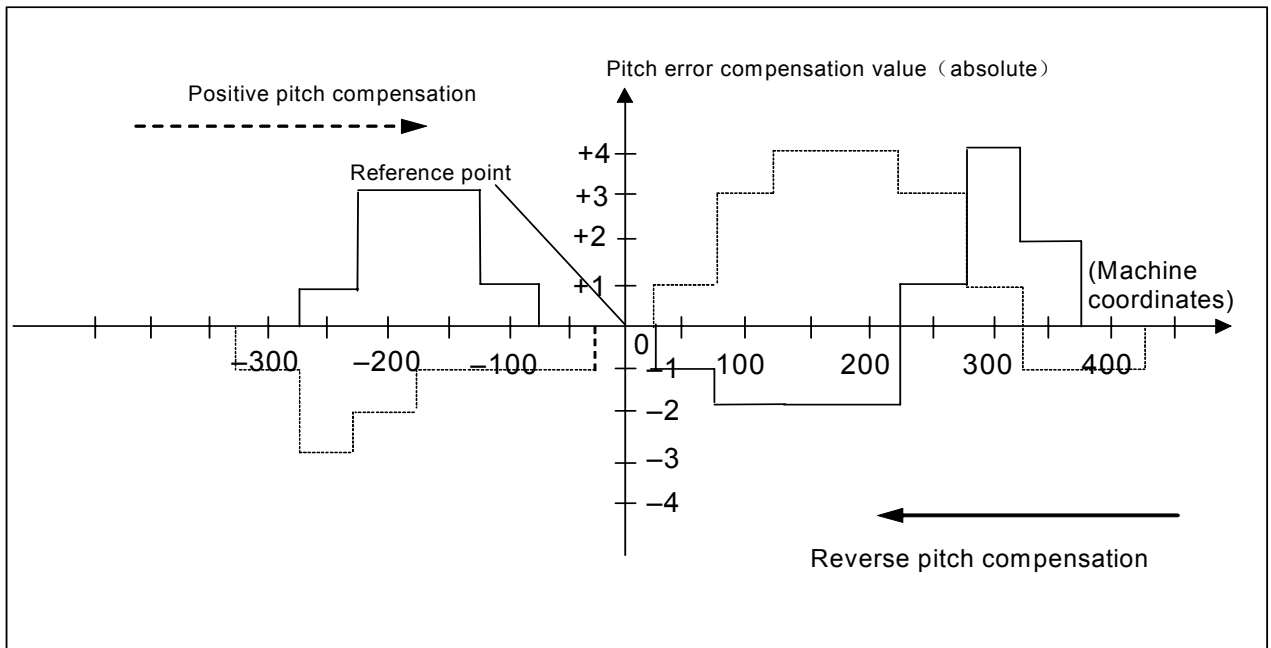


Fig.4-7-3

Positive pitch compensation (short line drawing):

Number	34	35	36	37	38	39	40	41	42	43	44	45	46	47	49
Compensation value	+1	+2	-1	-1	0	0	-1	+1	+3	+1	0	-1	-2	-2	+1

Negative pitch compensation (long line drawing):

Number	1034	1035	1036	1037	1038	1039	1040	1041	1042	1043	1044	1045	1046	1047
Compensation value	-1	-2	0	+2	+1	0	-1	-1	0	0	+3	+3	-2	-2

## 4.8 Machine zero return

### 1、Conception of machine zero

The machine coordinate system is the inherent coordinate system of machine. Its origin is called mechanical zero (or machine zero), as is called reference point in this manual. It is usually fixed at the maximum stroke point of X axis, Y axis or Z axis. This origin that is a fixed point is set after the machine is designed, manufactured and adjusted. As the machine zero is not confirmed by the CNC system at power-on, the auto or manual machine zero return is usually performed.

The machine zero return has two types: one-revolution-signal, non-one-revolution-signal. It is set by bit parameter No.6#6. For the zero return of the non-one-revolution-signal by the motor, it is classified for the A, B two types. It is set by bit parameter No.6#7.



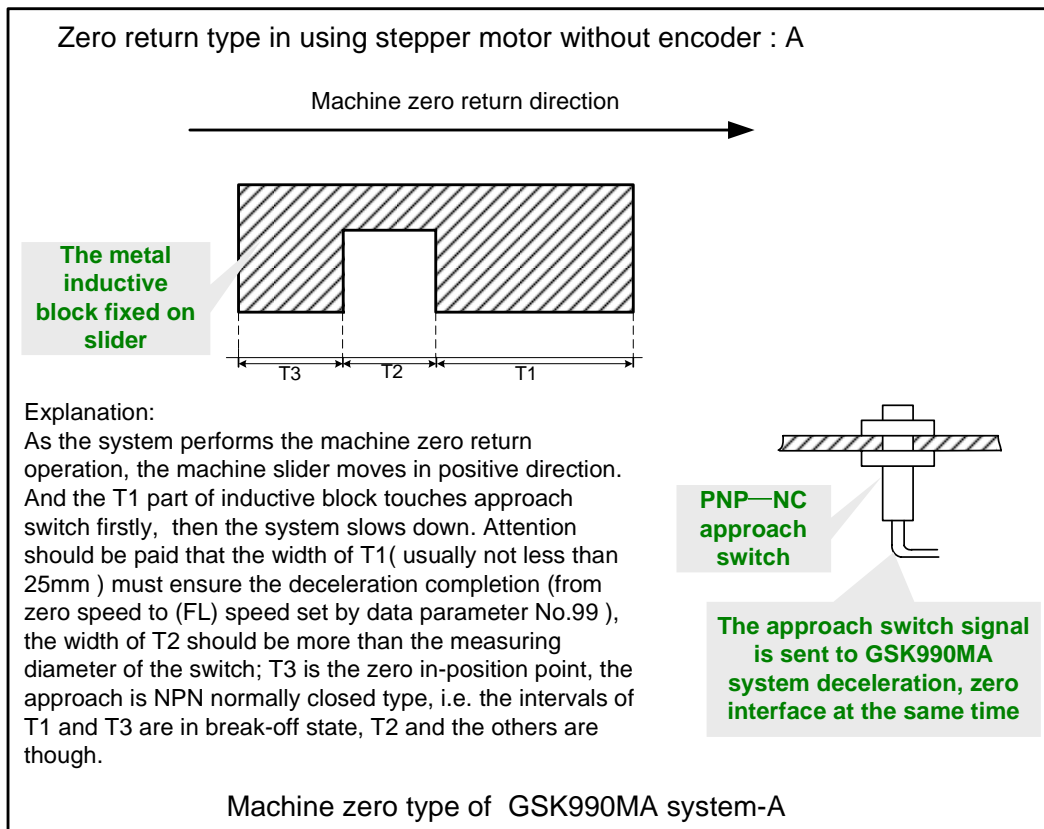


Fig. 4-8-1

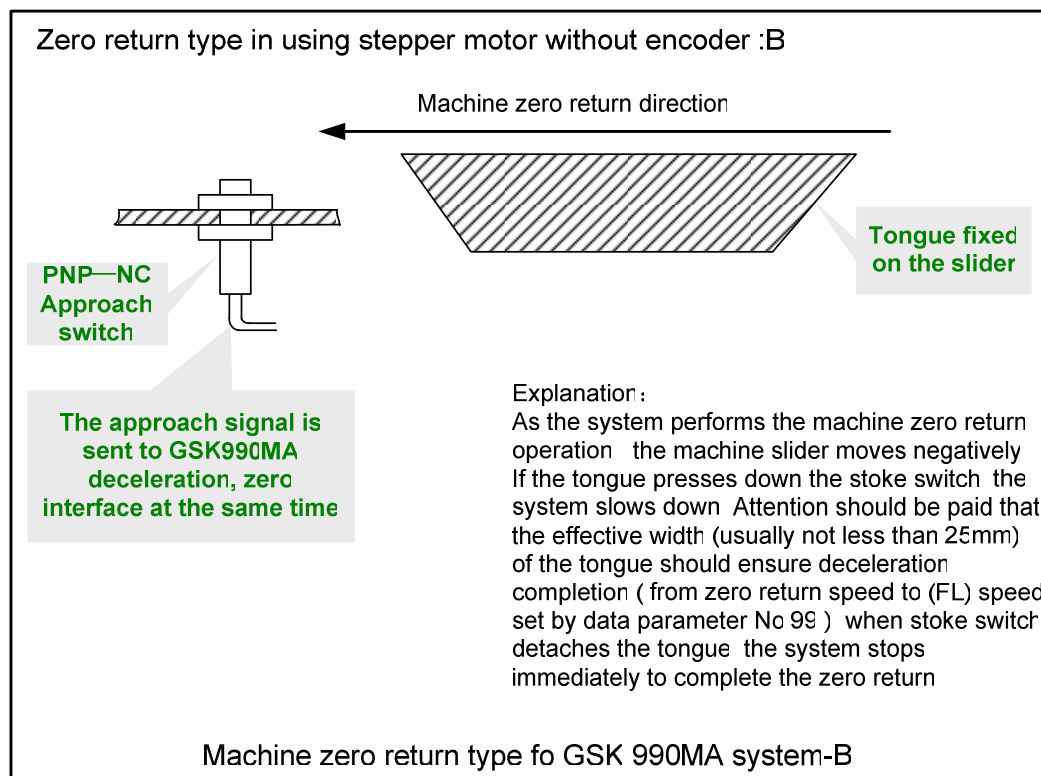



Fig. 4-8-2

## 2 . Steps for machine zero



- (1) Press  to enter Machine Zero mode, the characters “machine zero “will be displayed at the down-right of the LCD screen;
- (2) Select the axis X, Y, or Z for machine zero and its direction is set by bit parameter **No. 7#3～No.7#5**;
- (3) The machine moves towards the machine zero. Before the deceleration point is reached the machine traverses rapidly(traverse speed set by number parameter No.100～No.102), then moves to the machine zero point(i.e. reference point ) by a speed of FL(set by number parameter No.099) if the machine touches the deceleration switch. As the machine zero is reached, the corresponding axis moving stops and the Machine Zero indicator lights up.

## 3 . Machine zero steps by program

After the bit parameter No.4#3 is set for 0, the machine zero can be specified by G28 instruction. Because it detects the stroke tongue, this instruction is equivalent to manual machine zero.

Note:

1. If the machine zero is not fixed on your machine, don't perform the machine zero operation.
2. The indicator of the corresponding axis lights up when the machine zero is finished.
3. The indicator is gone out on condition that the axis is moved out from the machine zero by the operator.
4. Refer to the machine builder's manual for the direction of the machine zero (reference point).

### ● Signals

DECX: X-axis deceleration signal;

DECY: Y-axis deceleration signal;

DECZ: Z-axis deceleration signal;

DEC4: 4<sup>th</sup>-axis deceleration signal;

Parameter diagnosis(machine side input state)

State address					X1.3	X1.2	X1.1	X1.0
Pin					XS43.22	XS43.09	XS43.21	XS43.08

State parameter No.001

0	0	1	SJZ						
---	---	---	-----	--	--	--	--	--	--

**SJZ** =1: reference point memory: do.  
 =0: reference point memory: not.

System parameter number

0	0	6	MAOB	ZPLS					
---	---	---	------	------	--	--	--	--	--

**ZPLS** =1: zero type selection: one-revolution signal.  
 =0: zero type selection: non- one-revolution signal.  
**MAOB** =1: zero type selection for non-one-revolution: B  
 =0: zero type selection for non-one-revolution: A

State parameter No.007

0	0	7		ZMI4	ZMIz	ZMIy	ZMIx		
---	---	---	--	------	------	------	------	--	--

**ZMIx** =1: Direction setting of X axis reference point return: negative  
 =0: Direction setting of X axis reference point return: positive  
**ZMIy** =1: Direction setting of Y axis reference point return: negative  
 =0: Direction setting of Y axis reference point return: positive  
**ZMIz** =1: Direction setting of Z axis reference point return: negative  
 =0: Direction setting of Z axis reference point return: positive  
**ZMI4** =1: Direction setting of 4th axis reference point return: negative  
 =0: Direction setting of 4th axis reference point return: positive

Data parameter No.099

0	9	9	ZRNFL
---	---	---	-------

ZRNFL: low rate of X, Y, Z-axis reference point return(all axes).

Data parameters No.100~No.103

1	0	0	X-axis reference point return speed
1	0	1	Y-axis reference point return speed
1	0	2	Z-axis reference point return speed
1	0	3	4TH-axis reference point return speed

Speed setting of all-axis reference point return

## 4.9 Input/output signal control of spindle CW/CCW

- Signals

M03: spindle CCW

M04: spindle CW

M05: spindle stop

ENB: spindle enable

SAR: spindle speed arrival

ZSPD: spindle zero speed check

SION: spindle orientation

Parameter diagnosis (machine side output state)

state address	Y1.7	Y1.6						
Pin	XS40.13	XS40.25						

Y1.6=spindle CCW signal output; Y1.7= spindle CW signal output.

state address								Y2.0
Pin								XS41.01

Y2.0=spindle enable

Parameter diagnosis(machine side input state)

state address	X4.7	X4.6						
Pin	XS45.06	XS45.18						

X4.6=spindle speed arrival signal input; X4.7=spindle zero speed check signal input.

state address								X5.0
Pin								XS45.8

X5.0=spindle orientation completion signal

Data parameter No.245

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

Time for check spindle speed arrival signal

Data parameter No.257

2	5	7	
---	---	---	--

Spindle speed up limit in tapping cycle

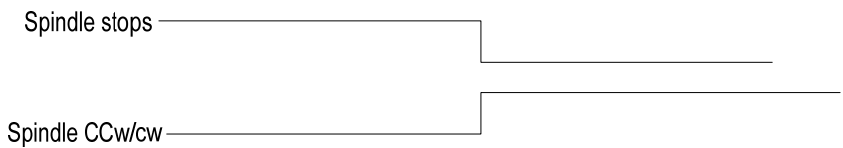
Data parameter No.258

2	5	8	
---	---	---	--

Spindle speed up limit

● Operation time sequence

Time sequence of spindle operation is as Fig. 3-3-1:



**Fig. 3-3-1 time sequence for spindle CW**

● Control logic

- 1 The spindle stops and M05 outputs when CNC is turned on;
- 2 After M3/M4 is executed, it is enabled and M05 stops output.

**4.10 Spindle automatic gear change control**

● Signals

Y3.4~Y3.6: Spindle automatic gear shift output signal

X4.1~X4.3: Spindle gear change completion signal

When CNC selects the spindle frequency conversion control (0~10V analog voltage output), it can support 3-gear spindle automatic gear change control and 3-gear change completion check function.

● Signal diagnosis

Parameter diagnosis(machine side output state)

state		Y3.6	Y3.5	Y3.4				
address								
Pin		XS44.25	XS44.12	XS44.24				

Y3.4=spindle gear 1 output; Y3.5=spindle gear 2 output; Y3.6=spindle gear 3 output.

Parameter diagnosis(machine side input state)

state					X4.3	X4.2	X4.1	
address								
Pin					XS45.15	XS45.02	XS45.13	

X4.1= spindle gear 1 in-position; X4.2=spindle gear 2 in-position; X4.3=spindle gear 3 in-position.

- Control parameter

State parameter

0	0	1						SPT		
---	---	---	--	--	--	--	--	-----	--	--

**SPT** =1: Spindle control: I/O point.

=0: Spindle control: frequency conversion or other modes.

Data parameter No.246

2	4	6	
---	---	---	--

Corresponding to max. speed of gear 1.

Data parameter No.247

2	4	7	
---	---	---	--

Corresponding to max. speed of gear 2.

Data parameter No.248

2	4	8	
---	---	---	--

Corresponding to max. speed of gear 3.

Data parameter No.250

2	4	9	
---	---	---	--

The maximum speed of the motor during the spindle gear change (The maximum speed is the motor one when the transducer is corresponding to 10V voltage.)

Data parameter No.250

2	5	0	
---	---	---	--

Motor speed during the spindle gear change (relative to the speed of data parameter No.249)

Note:

1. When the machine is with the automatic gear change device, K8.4 is set as 1; otherwise, 0. When the automatic gear change is invalid, the maximum speed of gear 1 is defaulted and  $246 \geq 247 \geq 248$ .
2. When the spindle gear detection isn't with the detection switch, K9.3 should be set as 1; otherwise, 0.
3. When the spindle is I/O point control, K4.0 should be set as 1.

- Control logic

1. Up to 1 of S1~S3 is enabled;
2. S1~S3 stops output after S0 is executed.
3. When some S\*\* is executed, the corresponding S\*\* output is enabled and kept, and CNC automatically stops other S\*\* output.

4.11    External cycle start and feed hold

- Signals

ST:    External automatic cycle start signal has the same function as that of automatic cycle start key on the machine panel.

\*SP:    Feed hold signal has the same function that of the feed hold key on the machine panel.
- Signal diagnosis

Parameter diagnosis (machine side output state)

State address		X1.6	X1.5					
Pin		XS43.25	XS43.12					

- Internal circuit of signal

\*SP/ST signal internal circuit is as **Fig. 4-11-1**:

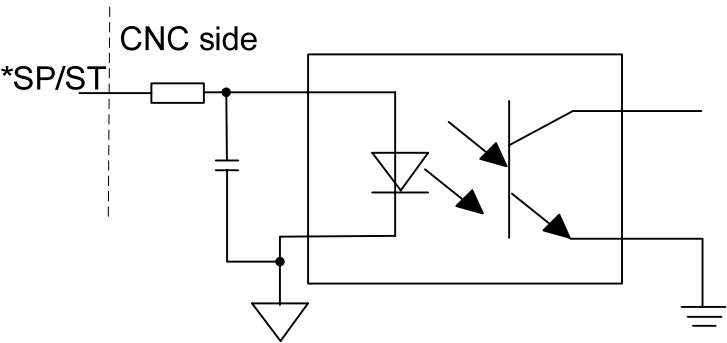


Fig.4-11-1

- External circuit

\*SP,ST signal external circuit is as **Fig. 4-11-2**.

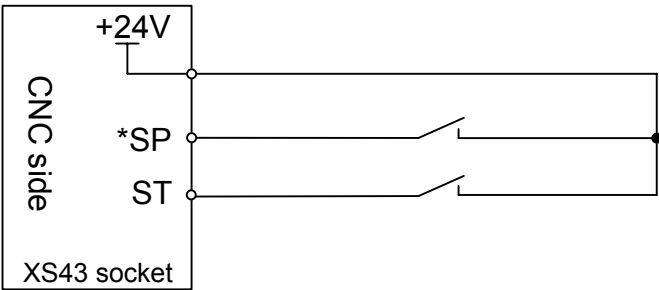


Fig.4-11-2

System parameter number

0	5	9	OTOP						
---	---	---	------	--	--	--	--	--	--

**OTOP** =1: Use external start and stop.

=0: No use external start and stop.

## 4.12 External editing lock and external operator panel lock

- Signals

**LEDT**: External editing lock signal, if it is 1, the program edit can be done. And its function is the same with that of the system program switch.

**LSYS**: External operator panel lock signal, if it is 1, all the machine operator keys are locked.

### Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G01	LSYS	LEDT						

System diagnosis (CNC side input state)

State address							X3.1	X3.0
Pin							XS44.21	XS44.8

X3. 0= external editing lock    X3. 1= external operator panel lock

System parameter number

0	5	9		OTOP	LOPT					
---	---	---	--	------	------	--	--	--	--	--

**LOPT** =1: Use external operator panel lock.

=0: Not use external operator panel lock.

**OTOP** =1: Use external editing lock.

=0: Not use external editing lock.

## 4.13 Cooling, lubricant and chip removal control

- Signals

M08: cooling ON

M35: Chip removal ON

- Signal diagnosis



Parameter diagnosis(machine side output state)

State address							Y0.1	
Pin							XS40.14	

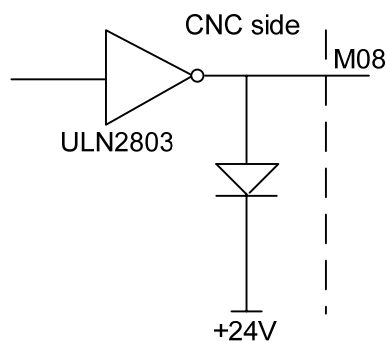
Y0.1=cooling switch control

Parameter diagnosis(machine side output state)

State address							Y1.1	
Pin							XS40.21	

Y1.1=chip removal switch control.

- Internal circuit is as **Fig. 4-13-1**:



**Fig. 4-13-1 M08 internal circuit**

## 4.14 Parameter of feedrate

System parameter number

0	1		FDR	RDR	TDR	RFO			LRP	RPD
---	---	--	-----	-----	-----	-----	--	--	-----	-----

**RPD** =1: Manual rapid traverse is enabled during the period from power-on time to the completion of the reference point return.

=0: Manual rapid traverse is disabled during the period from power-on time to the completion of the reference point return.

**LRP** =1: The positioning(G00) interpolation type is linear.

=0: The positioning (G00) interpolation type is non-linear.

**RFO** =1: The machine tool stops moving when the feedrate override is F0 in rapid traverse.

=0: The machine tool does not stop moving when the feedrate override is F0 in rapid traverse.

**TDR** =1: Dry run is valid in the tapping.

=0: Dry run is invalid in the tapping.

**RDR** =1: Dry run is valid in cutting feed.

=0: Dry run is invalid in cutting feed.

**FDR** =1: Dry run is valid in rapid positioning

=0: Dry run is invalid in rapid positioning

System parameter number

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

**HFC** =1: Clamp combined by straight line and arc for helical interpolation feedrate

=0: Clamp by straight line and arc separately for helical interpolation feedrate

0086	Dry run speed	5000
------	---------------	------

Setting range: 0~9999 (mm/min)

0087	The cutting feedrate during Auto mode when power-on.	300
------	--	-----

Setting range: 0~9999 (mm/min)

0088	X-axis rapid running speed	5000
------	----------------------------	------

Setting range: 0~9999 (mm/min)

0089	Y-axis rapid running speed	5000
------	----------------------------	------

Setting range: 0~9999 (mm/min)

0090	Z-axis rapid running speed	5000
------	----------------------------	------

Setting range: 0~9999 (mm/min)

0091	4TH-axis rapid running speed	5000
------	------------------------------	------

Setting range:0~9999 (mm/min)

0093	All-axis rapid running override F0 speed (all axes)	30
------	---	----

Setting range: 0~1000 (mm/min)

0094	Max. feedrate(all axes)	8000
------	-------------------------	------

Setting range: 0~9999 (mm/min) max. controlled speed in non-prediction control mode

0095	Min. feedrate (all axes)	0
------	--------------------------	---

Set range:0~500 (mm/min) min. controlled speed in non-prediction control mode

0096	Max. control speed in predict control mode(all axes)	6000
------	--	------

Setting range: 0~9999 (mm/min)

0097	Min. control speed in predict control mode(all axes)	0
------	--	---

Setting range: 0~500 (mm/min)

0098	All-axis JOG continuous feedrate	2000
------	----------------------------------	------

Setting range: 0~5000 (mm/min)

0099	Reference point return speed (all axes)	40
------	---	----

Setting range: 0~500

0100	X-axis reference point return speed	4000
------	-------------------------------------	------

Setting range:

0101	Y-axis reference point return speed	4000
------	-------------------------------------	------

Setting range: 0~9999

0102	Z-axis reference point return speed	4000
------	-------------------------------------	------

Setting range: 0~9999

0103	4TH-axis reference point return speed	4000
------	---------------------------------------	------

Setting range: 0~9999

## 4.15 Parameter of tapping

Position parameter number

0	4	4		FHD	PCP	DOV			VGR	G84
---	---	---	--	-----	-----	-----	--	--	-----	-----

**G84** =1: Use M codes in rigid tapping

=0: Not use M codes in rigid tapping

**VGR** =1: Arbitrary gear ratio of the spindle and position encoder enabled in rigid tapping.

=0: Arbitrary gear ratio of the spindle and position encoder disabled in rigid tapping.

**DOV** =1: Override is valid in the rigid tapping retraction.

=0: Override is invalid in the rigid tapping retraction.

**PCP** =1: To change rigid tapping for high-speed peck drilling cycle.

=0: Not change rigid tapping for high-speed peck drilling cycle.

**FHD** =1: Single block effective for feed dwell during rigid tapping.

=0: Single block ineffective for feed dwell during rigid tapping.

System parameter number

0	4	5				OV3	OVU	TDR		NIZ
---	---	---	--	--	--	-----	-----	-----	--	-----

**NIZ** =1: To perform the rigid tapping finishing.

=0: Not perform the rigid tapping finishing.

**TDR** =1: Use the same time constant in the rigid tapping advance and retraction.

=0: Not use the same time constant in the rigid tapping advance and retraction.

**OVU** =1: 10% retraction override for rigid tapping.

=0: 1% retraction override for rigid tapping.

**OV3** =1: Spindle speed effective by program instruction.

=0: Spindle speed ineffective by program instruction.

System parameter number

0	4	6			ORI				SSOG	DGN
---	---	---	--	--	-----	--	--	--	------	-----

**DGN** =1: Difference of the spindle and the tapping axis errors

=0: Synch error in rigid tapping.

**SSOG** =1: For servo spindle control at the beginning of rigid tapping.

=0: For following spindle control at the beginning of rigid tapping.

**ORI** =1: To perform spindle dwell when rigid tapping starts.

=0: Not perform spindle dwell when rigid tapping starts.

K parameter number

0	0	7	PAP							
---	---	---	-----	--	--	--	--	--	--	--

**DGN** =1: Use spindle position mode

=0: No use spindle position mode

It is set to 1 when using rigid tapping, and set to 0 when using flexible tapping.

**PS: the setting method of rigid and flexible tapping for GSK990MA system matching with DAP03 spindle servo driver**

**Note:**

- (1) To perform the rigid tapping , the 990MA system must be matched with DAP03 spindle servo driver.
- (2) To perform the rigid tapping, 990MA must be connected correctly with DAP03 spindle servo driver

**Setting :**

一、 Rigid tapping setting: Use the system default parameter, only modify the following parameters:

1. Bit parameter: **NO:46#1 is set to 1. (servo mode)**
2. PLC parameter : **K7#7** is set to 1. (use spindle position mode)
3. Data parameter: top speed of parameter P257 for tapping should be consistent with spindle
4. Data parameter: the speed of parameter **P29** for tapping corresponding to gear-1 should be consistent with tapping top speed.
5. Parameter **PA4 of DAP03 is set to 5. (position and speed switching mode)**
6. Data parameter: P298 is the time constant of linear acceleration/deceleration for spindle and tapping axis, it is set to 300.
7. Data parameter: P302 is the time constant for spindle and tapping axis in retracting.
8. Provided that it is switched to position mode, and input instruction M29 S100, then the actual speed is 25(r), the electric gear ratio can be calculated by formula  **$P \cdot G = N \cdot C \cdot 4$**  by altering the data parameter P323 and P326 of the system or PA12 and PA13 of DAP03.  
 P: input signal pulses    G: electric gear ratio;    N: motor rotations number;  
 C: photoelectricity encoder pulses.
9. when gear ratio of spindle and motor is not 1:1, it needs to calculate the corresponding gear ratio to input to the parameter PA35 and PA36 of DAP03.
10. Ensure the CCW/CW of spindle is consistent in rigid tapping. The rotation speed fluctuation should not be too large;,it is the best within 3 revolutions.
11. For cutting some soft material (aluminums alloy, copper etc.), it is better to use helical tapping, the effect will be better if it can match with the coolant.

二、 **Flexible tapping setting:** Use the system default parameter, only modify the following parameters:

In flexible tapping, it only needs to set the bit parameter **NO:46#** to 0 (following mode), set **K7#7** to 0, use M29 instruction. The effect will better if 1~2 second pause time is instructed in the program.

**Note: The setting method above is for the machine with no gear change**

## Appendix 1:

### Guide of ladder for GSK990MA matching with turret tool magazine

#### 一. Notices of GSK990MA matching with turret tool magazine

- (1) Install wiring according to the ladder.
- (2) Set PLC parameters correctly to ensure that the ladder matches with the machine tool when the ladder is used
- (3) If the machine tool has special control requirements to add program in the ladder, the persons responsible for modification should be master the electric and PLC method, and the ladder.
- (4) The ladder is only suitable for general CNC milling machine and the machining center with the turret tool magazine, otherwise, other machine tools may result in the unexpected accident.
- (5) The ladder is referred, and the ladder is different for the different machine tool.

#### 二. Allocation and definition of GSK990MA PLC/IO address and internal auxiliary relay and register

**Table 1 Input signal interface definition**

Address	Signal interface	Interface pin	Definition	Contact selection
X000.0	XS43	1	X axis positive travel limit signal	Normally closed contact
X000.1	XS43	14	X axis negative travel limit signal	Normally closed contact
X000.2	XS43	2	Y axis positive travel limit signal	Normally closed contact
X000.3	XS43	15	Y axis negative travel limit signal	Normally closed contact
X000.4	XS43	17	Z axis positive travel limit signal	Normally closed contact
X000.5	XS43	5	Z axis negative travel limit signal	Normally closed contact
X000.6	XS43	18	4TH axis positive travel limit signal	Normally closed contact
X000.7	XS43	6	4TH axis negative travel limit signal	Normally closed contact
X001.0	XS43	8	X axis zero return deceleration signal	Normally closed contact
X001.1	XS43	21	Y axis zero return deceleration signal	Normally closed contact
X001.2	XS43	9	Z axis zero return deceleration signal	Normally closed contact
X001.3	XS43	22	4TH axis zero return deceleration signal	Normally closed contact
X001.4	XS43	24	Emergency stop switch	Normally closed contact

## Appendix

X001.5	XS43	12	External cycle start	Normally open contact
X001.6	XS43	25	External feed hold	Normally open contact
X001.7	XS43	13	Lubrication pressure or oil level detection	Normally open contact
X002.0	XS44	1	Air supply pressure detection	Normally open contact
X002.1	XS44	14	Skip signal	
X002.2	XS44	2	Undefined	
X002.3	XS44	15	Undefined	
X002.4	XS44	17	External clamping/releasing tool control	Normally open contact
X002.5	XS44	5	Releasing tool check	Normally open contact
X002.6	XS44	18	Clamping tool check	Normally open contact
X002.7	XS44	6	Spindle tool check switch	Determined by the parameter
X003.0	XS44	8	Edit lock	Normally open contact
X003.1	XS44	21	Operation lock	Normally open contact
X003.2	XS44	9	Undefined	
X003.3	XS44	22	Tool magazine zero return (external)	Normally open contact
X003.4	XS44	24	Tool magazine CCW(external)	Normally open contact
X003.5	XS44	12	Tool magazine CW (external )	Normally open contact
X003.6	XS44	25	Tool magazine forward (external)	Normally open contact
X003.7	XS44	13	Tool magazine backward(external)	Normally open contact
X004.0	XS45	1	Undefined	
X004.1	XS45	14	Spindle gear-1 in-position	Normally open contact
X004.2	XS45	2	Spindle gear-2 in-position	Normally open contact
X004.3	XS45	15	Spindle gear-3 in-position	Normally open contact
X004.4	XS45	17	Spindle speed/position status output	Determined by the parameter
X004.5	XS45	5	Undefined	
X004.6	XS45	18	Spindle speed/position arrival	Determined by the parameter
X004.7	XS45	6	Spindle zero speed check	Normally closed contact
X005.0	XS45	8	Spindle orientation in-position	Determined by the parameter
X005.1	XS45	21	Tool magazine forward in-position	Determined by the parameter
X005.2	XS45	9	Tool magazine backward in-position	Determined by the parameter
X005.3	XS45	22	Tool magazine CCW/CW in-position	Determined by the parameter

X005.4	XS45	24	Tool magazine zero return in-position	Determined by the parameter
X005.5	XS45	12	Current tool magazine count check switch	Determined by the parameter
X005.6	XS45	25	Undefined	
X005.7	XS45	13	Undefined	
X006.0	XS22	6	External MPG X axis selection	Normally open contact
X006.1	XS22	2	External MPG Y axis selection	Normally open contact
X006.2	XS22	7	External MPG Z axis selection	Normally open contact
X006.3	XS22	3	External MPG A axis selection	Normally open contact
X006.4	XS22	8	External MPG step 0.001	Normally open contact
X006.5	XS22	4	External MPG step 0.01	Normally open contact
X006.6	XS22	9	External MPG step 0.1	Normally open contact
X006.7	XS21	ESP (4, 9)	External emergency stop	Normally closed contact

**Note:**

- (1) Refer to **Volume Four** about PLC input (X) connection and precautions
- (2) Contact selection: normally open contact and normally closed contact. It is determined by the parameter, thereinto, it is determined by parameter means that this input point to normally open contact or normally closed contact is determined by the KAPA parameter in the PLC, see **KAPA parameter** for details.
- (3) When the ladder is used, the user can add the new function for the undefined input part. (it is needed to modify the ladder diagram when using the undefined part, please do it cautiously) .

**Table 2 Output signal interface definition**

Address	Signal interface	Interface pin	Definition
Y000.0	XS40	1	Z axis holding brake
Y000.1	XS40	14	Cooling
Y000.2	XS40	2	Tool releasing/clamping
Y000.3	XS40	15	Undefined
Y000.4	XS40	17	Spindle brake
Y000.5	XS40	5	Undefined
Y000.6	XS40	18	Red alarm light
Y000.7	XS40	6	Yellow alarm light
Y001.0	XS40	8	Green alarm light
Y001.1	XS40	21	Chip removal control
Y001.2	XS40	9	Lubrication control
Y001.3	XS40	22	Machine light control
Y001.4	XS40	24	Undefined



## Appendix

Y001.5	XS40	12	Spindle blowing
Y001.6	XS40	25	Undefined
Y001.7	XS40	13	Undefined
Y002.0	XS41	1	Spindle enabling
Y002.1	XS41	14	Spindle orientation
Y002.2	XS41	2	Spindle CCW
Y002.3	XS41	15	Spindle CW
Y002.4	XS41	17	Hand unit light
Y002.5	XS41	5	Undefined
Y002.6	XS41	18	Washing hydrovalve output
Y002.7	XS41	6	Spindle position/speed mode change
Y003.0	XS41	8	Tool magazine CCW
Y003.1	XS41	21	Tool magazine CW
Y003.2	XS41	9	Tool magazine forward
Y003.3	XS41	22	Tool magazine backward
Y003.4	XS41	24	Spindle gear 1 <sup>st</sup> (frequency conversion\IO point control)
Y003.5	XS41	12	Spindle gear 2 <sup>nd</sup> (frequency conversion\IO point control)
Y003.6	XS41	25	Spindle gear 3 <sup>rd</sup> (frequency conversion\IO point control)
Y003.7	XS41	13	Spindle gear 4 <sup>th</sup> (frequency conversion\IO point control))
Y004.0	XS42	1	Undefined
Y004.1	XS42	14	Undefined
Y004.2	XS42	2	Undefined
Y004.3	XS42	15	Undefined
Y004.4	XS42	17	Undefined
Y004.5	XS42	5	Undefined
Y004.6	XS42	18	Undefined
Y004.7	XS42	6	Undefined
Y005.0	XS42	8	Undefined
Y005.1	XS42	21	Undefined
Y005.2	XS42	9	Undefined
Y005.3	XS42	22	Undefined
Y005.4	XS42	24	Undefined
Y005.5	XS42	12	Undefined
Y005.6	XS42	25	Undefined
Y005.7	XS42	13	Undefined

**Note:**

- (1) Refer to **Volume Four** for PLC input (X) connection and precautions
- (2) When the ladder is used, the user can add the new function for the undefined input part ( it is needed to modify the ladder diagram when using the undefined part, please do it cautiously ).

**Table 3 KAPA address definition**

Address	Definition	Status 0	Status1	Setting value by customer
K000.0	Permit PLC parameter to be modified	No permission	Permission	
K000.1	Permit PLC signal to be debugged	No permission	Permission	
K000.2	All Y signals are cleared after PLC enters the debugging mode.	Not clearing	Clearing	
K000.3	Permit A address information to be edited	No permission	Permission	
K000.4	Permit K address information to be edited	No permission	Permission	
K000.5	Permit X address information to be edited	No permission	Permission	
K000.6	Permit Y address information to be edited	No permission	Permission	
K000.7	Permit the instruction table to be operated	No permission	Permission	
K001.0	Permit the tool magazine to be used	No permission	Permission	
K001.1	If the ladder downloads automatically when it is converted	No.	Yes	
K001.2	If clear the external output interface when PLC stopped			
K004.0	If the spindle uses the gear control(I/O point)	No	Yes	
K004.1	If the manual reference point only controls one axis	Many axes	one axis	
K005.0	if the machine tool has external handwheel (MPG)	No	Yes	
K005.1	if the machine tool has external cycle Start function	No	Yes	
K005.2	If the system enters the debugging mode	No	Yes	
K006.0	X axis limit switch selection	2 PCS	1 PCS	
K006.1	Y axis limit switch selection	2 PCS	1 PCS	
K006.2	Z axis limit switch selection	2 PCS	1 PCS	
K006.3	4TH axis limit switch selection	2 PCS	1 PCS	
K006.4	If X axis zero return switch selects the normally open type	No	Yes	
K006.5	If Y axis zero return switch selects the normally open type	No	Yes	
K006.6	If Z axis zero return switch selects the normally open type	No	Yes	

## Appendix

K006.7	If 4TH axis zero return switch selects the normally open type	No	Yes	
K007.0	X axis limit alarm reverse			
K007.1	Y axis limit alarm reverse			
K007.2	Z axis limit alarm reverse			
K007.3	4TH axis limit alarm reverse			
K007.4	If cancel the 4TH axis hard limit	No	Yes	
K007.7	If the spindle position mode is used	No	Yes	
K008.0	If the spindle positioning detection is normally open.	Normally closed	Normally open	
K008.1	The tool clamping/releasing control selection	External button	Button on panel	
K008.2	If the tool clamping/releasing device is not used.	No	Yes	
K008.3	if the automatic lubricating control is used	Yes	No	
K008.4	If the spindle has the gear change device	No	Yes	
K008.5	If spindle speed (speed mode)/position (position mode) check arrival is the normally closed	Normally closed	Normally open	
K008.6	If the check switch is closed (it is turned off in position mode) in the spindle speed mode	Closed	open	
K008.7	If the spindle position/speed mode conversion checks the signal	Yes	No	
K009.0	If the spindle position arrival signal checks the signal	Yes	No	
K009.1	If the lubrication pressure or oil level is checked	No	Yes	
K009.2	If the air pressure is checked	No	Yes	
K009.3	If the spindle gear detection has a detecting switch			
K010.0	The tool magazine rotation in-position mode selection	No reaction	Reaction	
K010.1	If the tool magazine has the zero switch	No	Yes	
K010.2	The tool magazine zero return contact selection	Normal open	Normal close	
K010.3	The tool magazine infeed tool contact selection	Normal open	Normal close	
K010.4	The tool magazine tool retraction contact selection	Normal open	Normal close	
K010.5	The tool counting switch contact selection	Normal open	Normal close	
K010.6	If the tool origin point setting is enabled	Disabled	Enabled	
K010.7	If the tool magazine enters the regulation mode	No	Yes	
K011.0	If the manual tool clamping/releasing prompts the tool number which corresponds to the spindle	Yes	No	
K011.1	If the manual operation returns the tool	No	Yes	

	change position			
K011.2	If the No.1244 alarm cleared by modifying the parameter	No	Yes	
K011.3	Clear the No.1244 alarm	Yes	No	
K011.4	If the output is break-off after the spindle positioning completion.	No		
K011.5	If the CCW signal is output when spindle positioning	No	Yes	
K012.0	If spindle tool detection is valid	No	Yes	
K012.1	If the spindle tool detection switch is normally open	No	Yes	
K012.2	if current magazine tool detection is valid	No	Yes	
K012.3	If the current tool magazine number detection switch is normally closed	No	Yes	
K015.0	if it is the operator panel B	No	Yes	

**Notes:**

- When the system normally runs, K0000, K0001, K0002, K0003, K0004, K0005, K0006, K0007, K0052, K0107 must be 0, otherwise there may be the unexpected accident.
- The modified K0010 is valid when the system is restarted, when K0010=0(i.e. the tool magazine is not used), the setting of the K0100, K010.1, K010.2, K010.3, K010.4, K010.5, K010.6, K010.7, K011.0, K011.1, K011.2 and K011.3 is disabled.
- When K0010=1(i.e. the tool magazine is enabled), K0082 setting is disabled.
- When K0082=0(i.e. the tool clamping/releasing device is no used), K0081 setting is disabled.
- K006.0 setting:  
When two travel limit switches on X axis have been fixed (the positive limit switch connects with X0.0, the negative limit switch connects with X 0.1) , K0060 set to 0 is enabled, and K007.0 setting is disabled.  
When one travel limit switch on X axis is fixed (connecting with X0.0), k006.0 sets to 1 is enabled, If the X axis moves in the positive direction to the limit, the negative direction hard limit occurs.  
When the system alarms or X axis moves in the negative direction to the limit, the system alarms for the positive direction, when the K0070 is modified, the alarm reverses, namely, the error alarm is regulated.  
The settings of K006.1 and K007.1 on Y axis, K006.2 and K007.2 on Z axis, K0063 and K0073 on the 4<sup>TH</sup> axis are the same as that of X axis.
- K0052=1, all alarm interlocks are released, and the system enters the debugging mode, which is used when the system is being tested, when the system run normally, the parameter is set to 0, otherwise, there may be the unexpected accident.
- Refer to **the usage and maintain of tool magazine on chapter three** for the parameter K0100, K0101, K0102, K0103, K0104, K0105, K0106, K0107, K0110, K0111
- K004.0 (if the spindle uses I/O control) =1: K008.4 (whether the spindle has the gear change device or not?) setting is disabled, i.e. the spindle has no gear change device.
- K015.0=0: the user should use the operator panel of GSK990MA; k015.0=1: use that of GSK218M.

**Table 4 PLC external alarm definition**

PLC alarm number	A address	Alarm content
1200	A000.0	Air pressure check abnormal
1201	A000.1	Lubricating check abnormal
1202	A000.2	Lubricating motor check abnormal
1203	A000.3	Cooling motor check abnormal
1204	A000.4	Chip removal motor check abnormal
1205	A000.5	Pressure oil pump check abnormal
1206	A000.6	Spindle cooling unit check abnormal
1207	A000.7	Machine tool light check abnormal
1208	A001.0	Machine tool control box check abnormal
1209	A001.1	Machine tool bed temperature check abnormal
1210	A001.2	Machine tool vibration check abnormal
1211	A001.3	Pressure oil temperature check abnormal
1212	A001.4	Oil pressure low
1213	A001.5	Machine not ready
1214	A001.6	Spindle gear can't rotate because of the abnormality
1215	A001.7	it is needed to execute T instruction before executing the M60
1216	A002.0	Do not rotate the spindle when the tool releases
1217	A002.1	The tool cannot release when the spindle rotates
1218	A002.2	reconfirm the tool number when the tool magazine stops abnormally
1219	A002.3	The spindle cannot rotate when the tool magazine is in the feed tool position
1220	A002.4	Spindle tool clamping check abnormal
1221	A002.5	Spindle tool releasing check abnormal
1222	A002.6	Spindle unit temperature check abnormal
1223	A002.7	Spindle speed check abnormal
1224	A003.0	Spindle motor enabling check abnormal
1225	A003.1	Spindle orientation in-position check abnormal
1226	A003.2	Spindle gear change abnormal
1227	A003.3	The tool magazine cannot rotate when it is not in the origin point
1228	A003.4	The tool magazine cannot execute the cycle start in the feed tool position
1229	A003.5	Set spindle tool number
1230	A003.6	The tool change cannot be executed when the spindle tool releases
1231	A003.7	The tool change cannot be executed when the tool magazine is in the retraction position
1232	A004.0	The tool magazine rotation in-position check abnormal
1233	A004.1	When the tool magazine rotates, the motor check is abnormal

1234	A004.2	The program stops run when the tool magazine is in the feed position
1235	A004.3	The tool magazine move in-position check abnormal
1236	A004.4	The tool magazine zero return check abnormal
1237	A004.5	It needs to execute the zero return when the tool magazine position lose
1238	A004.6	The feed in-position check abnormal
1239	A004.7	The retraction in-position check abnormal
1240	A005.0	The tool magazine executes the zero return because of the abnormal
1241	A005.1	The tool magazine feed check abnormal
1242	A005.2	The tool magazine retraction check abnormal
1243	A005.3	The tool magazine zero position setting is valid
1244	A005.4	Stop the abnormal tool change
1245	A005.5	There is no tool number or there is the repetitive tool number
1246	A005.6	The feed tool cannot be executed because it is not in tool change position
1247	A005.7	The tool magazine does not execute the infeed tool because the spindle does not perform the positioning
1248	A006.0	The tool magazine does not execute the retraction when the tool is released
1249	A006.1	The spindle with the tool does not execute the infeed tool
1250	A006.2	The spindle and the current tool number of tool magazine does not execute the infeed tool
1251	A006.3	Please cut off
1252	A006.4	The tool clamps and the Z axis cannot be moved
1253	A006.5	Debug the tool magazine carefully
1254	A006.6	The system does not execute the cycle start when the tool magazine is in the debugging mode
1255	A006.7	The clamped tool cannot return to the origin position
1256	A007.0	The clamped tool cannot return to the tool change position
1257	A007.1	The spindle cannot return to the tool exchange position
1258	A007.2	The tool magazine cannot return to the tool change position
1259	A007.3	The tool change cannot be executed because the T number abnormal
1260	A007.4	The cycle start cannot be executed when returning to the tool change position is executed manually
1261	A007.5	The retraction cannot be executed in the origin position
1262	A007.6	Exceed the safety position
1263	A0077	The cycle start cannot be executed in the debugging mode
1264	A0080	The spindle speed position mode conversion is

## Appendix

---

		abnormal
1265	A0081	The spindle position speed mode conversion is abnormal
1360	A020.0	Lubrication pressure low or oil level low

PLC alarm diagnosis:

Alarm information: 1200 the air pressure check is abnormal.

Fault cause: no

Troubleshooting: check the state of the X002.0

Alarm information: 1201 the lubricant check is abnormal.

Fault cause: defined by customer

Troubleshooting: no

Alarm information: 1202 the lubricating motor check is abnormal.

Fault cause: defined by customer

Troubleshooting: no

Alarm information: 1203 the cooling motor check is abnormal.

Fault cause: defined by customer

Troubleshooting: no

Alarm information: 1204 the chip removal motor check is abnormal.

Fault cause: defined by customer

Troubleshooting: no

Alarm information: 1205 the pressure oil pump motor check is abnormal.

Fault cause: defined by customer

Troubleshooting: no

Alarm information: 1206 the spindle cooling unit check is abnormal

Fault cause: defined by customer

Troubleshooting: no

Alarm information: 1207 the machine light check is abnormal.

Fault cause: defined by customer

Troubleshooting: no

Alarm information: 1208 the machine control box temperature check is abnormal.

Fault cause: defined by customer

Troubleshooting: no

Alarm information: 1209 the machine bed temperature check is abnormal.

Fault cause: defined by customer

Troubleshooting: no

Alarm information: 1210 the machine vibration frequency check is abnormal.

Fault cause: defined by customer

Troubleshooting: no

Alarm information: 1211 pressure oil temperature check is abnormal.

Fault cause: defined by customer

Troubleshooting: no

Alarm information: 1212 the oil pressure is low.

Fault cause: defined by customer

Troubleshooting: no

Alarm information: 1213 the machine is not ready.

Fault cause: defined by customer

Troubleshooting:

Alarm information: 1214 Spindle gear can't rotate because of the abnormality

Fault cause: Spindle gear abnormality is detected when the spindle is rotated by the instruction.

Resolution: the gear 1 in-position detection switch (X0041) is not closed within the setting time of T0021 when the spindle shifts for the gear 1 (i.e. Y3.4=1) ; adjust the T0021 setting time or check if X0041 is closed; the gear 1 in-position detection switch (X0042) isn't closed within the setting



time of T0022 when the spindle shifts for the gear 2 (i.e. Y3.5=1), adjust the T0022 setting time or check if X0042 is closed; the gear 1 in-position detection switch (X0043) isn't closed within the setting time of T0023 when the spindle shifts for the gear 3 (i.e. Y3.6=1), adjust the T0023 setting time or check if X0043 is closed.

Alarm information: 1215 it is needed to execute the T instruction before executing the M60 instruction.

Fault cause: The valid T instruction isn't executed when executing the M60 instruction.

Troubleshooting: Execute the valid T instruction before executing the M60 instruction.

Alarm information: 1216 do not rotate the spindle when the tool releases.

Fault cause: execute the spindle rotating when the tool releases: in Manual mode, press the spindle CCW, CW, JOG, POSITION, or execute M03,M04, M19 in AUTO mode, the spindle will rotate.

Troubleshooting: in Manual mode, press "CLAMP/RELEASE", and the tool is clamped (i.e. Y0.2=0) to check whether the tool clamp check switch (X2.6) is 1, when it is 1, the rotating spindle does not appear alarm.

Alarm information: 1217 the tool does not release when the spindle rotates.

Fault cause: the tool release instruction is executed when the spindle rotates.

Troubleshooting: when the spindle stops, the tool release instruction is executed to avoid the alarm.

Alarm information: 1218 reconfirm the tool number because the tool magazine abnormally stops.

Fault cause: M6 is executed when the tool magazine rotates, the alarm appears, or press "RESET", the alarm appears.

Troubleshooting: 1. the tool magazine executes the zero return.

2. set the spindle tool number and tool magazine number again.  
(see **chapter three** for details )

Alarm information: 1219 the spindle cannot rotate when the tool magazine in the infeed position

Fault cause: the spindle rotation is executed when the tool magazine does not retract to the in-position.

Troubleshooting: execute the tool magazine returning to the retraction position (if the tool magazine has returned to the retraction position by X5.2.

Alarm information: 1220 the clamped tool corresponding to the spindle is abnormal.

Fault cause: when the tool is clamped (i.e. Y0.2=0), the tool magazine clamp check switch (X2.6) is not closed during the time set by T010.

Troubleshooting: 1. Check whether the tool magazine clamp check switch is normal.  
2. Check whether Y0.2 outputs.  
3. Regulate again T010 setting time.

Alarm information: 1221 the released tool corresponding to the spindle is abnormal.

Fault cause: when the tool is clamped (i.e. Y0.2=1), the tool magazine clamp check switch (X2.5) is not closed during the time set by T009.

Troubleshooting: 1. Check whether the tool magazine release check switch is normal.  
2. Check whether Y0.2 outputs.  
3. Regulate again T009 time.

Alarm information: 1222 the spindle unit temperature check is abnormal

Fault cause: defined by customer

Troubleshooting: No

Alarm information: 1223 the spindle speed check is abnormal

Fault cause: Defined by customer

Troubleshooting: No

Alarm information: 1224 the spindle motor enabling check is abnormal.

Fault cause: Defined by customer

Troubleshooting: No

Alarm information: 1225 the spindle orientation in-position check is abnormal.

Fault cause: The spindle orientation check switch (X5.0) isn't closed within the setting time of T013 when the spindle is executing the positioning (i.e. Y2.1=1),

Troubleshooting: 1. The spindle driver or spindle encoder is normal.  
2. Check whether Y2.1 outputs.  
3. Regulate again T013 time.

Alarm information: 1226 the spindle gear is abnormal

Fault cause: the spindle gear abnormality is checked

Resolution: the gear 1 in-position detection switch (X0041) is not closed within the setting time of T0021 when the spindle shifts for the gear 1 (i.e. Y3.4=1) ; adjust the T0021 setting time or check if X0041 is closed; the gear 1 in-position detection switch (X0042) isn't closed within the setting time of T0022 when the spindle shifts for the gear 2 (i.e. Y3.5=1), adjust the T0022 setting time or check if X0042 is closed; the gear 1 in-position detection switch (X0043) isn't closed within the setting time of T0023 when the spindle shifts for the gear 3 (i.e. Y3.5=1), adjust the T0023 setting time or check if X0043 is closed.

Alarm information: 1227 the tool magazine which is not in the origin point cannot rotate.

Fault cause: The tool magazine is not in the retraction position and Z axis is not in the origin point, or press "MAG. CCW" or "MAG. CW", the alarm appears.

Troubleshooting: The tool magazine returns to the retraction position or Z axis returns to the origin point.

Alarm information: 1228 the tool magazine cannot execute the cycle start when it is in the infeed position

Fault cause: When the tool magazine is not in the retraction position, or press "CYCLE START", the alarm appears.

Troubleshooting: The tool magazine returns to the retraction position

Alarm information: 1229 please set the spindle tool number

Fault cause: Press "CLAMP/RELEASE" when the tool number is 0, which causes the alarm appears.

Troubleshooting: the alarm only prompts the tool cannot be installed on the spindle but KAPA0110 is modified to shield the alarm when the tool number is 0

Alarm information: 1230 the tool change cannot be executed when the spindle tool is released.

Fault cause: M06 or M50 is executed when the tool is released (Y0.2=1),

Troubleshooting: ensure that the spindle is clamped(Y0.2=0).

Alarm information: 1231 the tool change cannot be executed when the tool magazine is not in the retraction position.

Fault cause: M06 or M50 is executed when the tool magazine is not in the retraction position.

Troubleshooting: ensure that the spindle is in the retraction position.

Alarm information: 1232 the tool magazine rotating in-position check is abnormal

Fault cause: Defined by customer

Troubleshooting: No

Alarm information: 1233 the tool magazine rotation motor check is abnormal

Fault cause: defined by customer

Troubleshooting: No

Alarm information: 1234 the program stops the run when the tool magazine is in the infeed position.

Fault cause: when the program is running, the tool is not in the retraction position, which causes the alarm.

Troubleshooting: Executed the program when the tool magazine is in the retraction position.

Alarm information: 1235 the tool magazine moving in-position check is abnormal

Fault cause: Defined by customer

Troubleshooting: No

Alarm information: 1236 the tool magazine zero return check is abnormal

Fault cause: defined by customer

Troubleshooting: No

Alarm information: 1237 the tool magazine needs to return to zero because its origin is lost.

Fault cause: When the tool magazine is performing the zero return, it is stopped abnormally to cause an alarm.

Troubleshooting: the tool magazine returns to the zero once again

Alarm information: 1238 the infeed in-position check is abnormal

Fault cause: Defined by customer

Troubleshooting: No

Alarm information: 1239 the retraction in-position check is abnormal.

Fault cause: Defined by customer

Troubleshooting: No

Alarm information: 1240 the tool magazine needs to return the zero again because it is abnormal.

Fault cause: 1. the tool magazine stops the rotation in the time set by T102, the system has checks that the tool magazine count switch (X5.3) abnormally creates the pulse signal.  
2. when the tool magazine rotates, the system checks that the tool magazine count switch 0 or 1 exceeds the time set by T103.

Troubleshooting: 1. the tool magazine CCW or CW output part (Y3.0 Y3.1) is abnormal.  
2. the tool magazine count switch is abnormal.  
3. set T102 and T103 value again.

***(See details for its operation in the 3rd Section The usage and maintenance for GSK 990MA CNC system matching with the carousel-type tool magazine)***

Alarm information: 1241 the tool magazine infeed check is abnormal

Fault cause: The tool magazine infeed check switch (X5.1) doesn't act within the setting time of T104 when the tool magazine executes the infeed action (i.e. Y3.2=1).

Troubleshooting: 1. Check if the tool magazine infeed check switch is normal.  
2. Check if the Y3.2 outputs.  
3. Adjust the T104 setting time again.

Alarm information: 1242 the tool magazine retraction check is abnormal

Fault cause: The magazine infeed detecting switch (X5.2) doesn't act within the setting time of T105 when the tool magazine performs the retraction action (i.e. Y3.3=1)

Troubleshooting: 1. Check if the tool magazine infeed check switch is normal.  
2. Check if Y3.3 outputs.  
3. Adjust the setting time of T105 again

Alarm information: 1243 the tool magazine zero setting is valid.

Fault cause: when the tool magazine has no zero return switch (i.e. K010.1=0), and K0106=1, the system alarms and prompts that the tool magazine zero setting is invalid.

Troubleshooting: set K0106 to 0.

***(See details for its operation in the 3rd Section The usage and maintenance for GSK 990MA CNC system matching with the carousel-type tool magazine)***

Alarm information: 1244   stopping for abnormal tool change to ensure the magazine.

Fault cause: The tool change stops because of its abnormal run when the tool magazine is executing automatic tool change or it is checked not in the retraction position when starting the system, this alarm occurs, which only prompts that there may be the disorder of tool magazine and the spindle tool number, it is needed to reset the tool magazine and spindle tool number after clearing the alarm to ensure the tool magazine normal.

***(See details for its operation in the 3rd Section The usage and maintenance for GSK 990MA CNC system matching with the carousel-type tool magazine)***

Troubleshooting: 1. when K001.2=1(i.e. clear the NO.1224 alarm by modifying the parameter), set the K011.3 (clear the message of the NO.1244 alarm) to 0, and then press "RESET" to clear the alarm; when K011.2=0 (i.e. it is no need to modify the parameter to clear the NO.1244 alarm), press "RESET " to clear the alarm.

2. Readjust the tool magazine and confirm if the spindle tool number is congruous, including if the tool magazine has returned to the retraction position, if the spindle is clamped, if the current cutter head number is coincided with the current value of CTR100, if the spindle tool number is coincide with the value of D245, ensure there is no tool in the spindle.

Alarm information: 1245   there is no T tool number or repeated tool number in the tool list.

Fault cause:       there is no tool number specified by T code or there is the repeated tool number specified by T code in the tool list (D001-D099).

Troubleshooting:   modify the tool number value in the tool list.

Alarm information: 1246   the tool magazine cannot execute the tool change because it is not in the tool change position.

Fault cause:       Z axis is not the tool change position, and the tool magazine infeed is executed.

Troubleshooting: execute G91G30Z0 to make Z axis return to the tool change position.

Alarm information: 1247   the tool magazine cannot execute the infeed because the spindle does not position.

Fault cause:       the spindle does not position and the tool magazine infeed is executed.

Troubleshooting: position the spindle.

Alarm information: 1248 the tool magazine does not execute the retraction when the tool is released.

Fault cause: the retraction is executed when the spindle tool is released.

Troubleshooting: execute the retraction after the spindle tool is clamped.

Alarm information: 1249 the spindle with the tool cannot execute the infeed.

Fault cause: the infeed is executed when Z axis is in the origin and the spindle has the tool.(D245 is not 0)

Troubleshooting: dismount the tool on the spindle and set D245 to 0.

Alarm information: 1250 the infeed cannot be executed because the tool number on the spindle is not the same that of the current tool magazine.

Fault cause: Z axis is in the tool change position, and the infeed is executed when the tool number on the spindle is not the same that of the current tool magazine.

Troubleshooting: rotate the tool magazine to ensure that the tool number on the spindle is the same that of current tool magazine.

Alarm information: 1251 please turn off the power supply

Fault cause: the modified parameter is enabled after power-off.

Troubleshooting: restart the system.

Alarm information: 1252 Z axis cannot move because the tool is clamped.

Fault cause: when the tool is in the infeed position and the spindle tool is clamped, Z axis moves.

Troubleshooting: 1. the tool magazine is in the retraction position.  
2. the spindle tool is released.

Alarm information: 1253 carefully operate the system because the tool magazine is in the debug mode.

Fault cause: When K0107 is set to 1, the system alarms, which prompts the tool magazine enters the debug mode, and which is not related to its other alarms and interlock signal, at this time, we should carefully operate the system, otherwise, there maybe damage the machinery.

Troubleshooting: press "RESET".(Note: it doesn't mean that the system is not in the tool magazine debug mode , it is need to set the K010.7 to 0 to exit the tool magazine debug mode )

Alarm information: 1254 the system cannot execute the cycle start

Fault cause: when K0107 is set to 1, the "CYCLE START" is pressed in AUTO or MDI or DNC mode, which causes the system alarms.

Troubleshooting: set K0107 to 0.

Alarm information: 1255 the tool cannot return to the origin point because it is clamped.

Fault cause: Z axis is executed to return to the origin point when the tool magazine is in the infeed position and the spindle tool is clamped.

Troubleshooting: 1. the tool magazine is in the retraction position.  
2. the spindle tool is released.

Alarm information: 1256 the tool cannot return the tool change position because it is clamped.

Fault cause: Z axis is executed to return to the tool change position when the tool magazine is in the infeed position and the spindle tool is clamped.

Troubleshooting: 1. the tool magazine is in the retraction position.  
2. the spindle tool is released.

Alarm information: 1257 the spindle cannot return the tool change position because the spindle does not position

Fault cause: Z axis is executed to return to the tool change position when the tool magazine is in the infeed position and the spindle is not positioned.

Troubleshooting: 1. The tool magazine is in the retraction position.  
2. Position the spindle

Alarm information: 1258 the tool magazine cannot return to the tool change position because it is abnormal.

Fault cause: reserved

Troubleshooting: no

Alarm information: 1259 the tool change cannot be executed because the T tool number is abnormal.

Fault cause: the error T instruction is executed before executing M06, (error T instruction means: there is no tool number specified by T code or there is the repeated tool number specified by T code in the tool list (D001-D099)).

Troubleshooting: Execute the M06 instruction after executing the correct T instruction.



Alarm information: 1260 the system cannot execute the cycle start because the manual tool change is executed.

Fault cause: when K0111 is set to 1(i.e. the manual tool change position return is valid), the “CYCLE START” is pressed in AUTO or MDI or DNC mode, which causes the system alarms.

Troubleshooting: set K0111 to 0.

Alarm information: 1261 the tool cannot execute the retraction because it is not in the origin point.

Fault cause: the tool magazine retraction is executed when the tool magazine is in the infeed position and Z axis is not in the origin point.

Troubleshooting: the retraction is executed after Z axis returns to the origin point.

Alarm information: 1262 exceed the safety position.

Fault cause: Z axis exceeds the tool change position when the tool magazine is in the infeed position and Z axis moves.

Troubleshooting: move Z axis to the position between the tool change position and origin point.

Alarm information: 1263 the system cannot execute the cycle start when it is in the debug mode.

Fault cause: Defined by customer

Troubleshooting: No

Alarm information: 1264 the spindle position mode conversion is abnormal

Fault cause: when M29 is executed, the system has not received the spindle position mode completion signal in the time set by T24.

Troubleshooting: adjust T24 setting time or ensure the spindle position mode completion signal outputs

Alarm information: 1265 the spindle speed mode conversion is abnormal

Fault cause: when M28 is executed, the system has not received the spindle speed mode completion signal in the time set by T28.

Troubleshooting: adjust T28 setting time or ensure the spindle speed mode completion signal outputs

Alarm information: 1360 lubrication pressure low or oil level low

Fault cause:

Troubleshooting: check X001.7

**Note:** when the alarm is for “User definition” and there is no the alarm in the ladder, the user should modify the ladder to increase it.

**Table 5 TMR parameter definition**

Address	Statement	Initial value (ms)	Setting value by customer (ms)
T0001	Delay timer for spindle CCW completion	500	
T0002	Delay timer for spindle CW completion	500	
T0003	Delay timer for spindle gear change completion	500	
T0004	Delay timer for spindle positioning completion	0	
T0005	Delay timer for auxiliary function(M.S.T) completion	0	
T0006	Timer for spindle gear change check	500	
T0007	Delay timer for spindle tool release completion	0	
T0008	Delay timer for spindle tool clamp completion	0	
T0009	Time setting for spindle tool release check	8000	
T0010	Time setting for spindle tool clamp check	8000	
T0011	Pulse signal time 1 in 1 second	500	
T0012	Pulse signal time 2 in 1 second	500	
T0013	Time setting for spindle positioning check time	8000	
T0014	Time setting for spindle CCW check	500	
T0015	Time setting for spindle CW check	500	
T0016	Time unit setting for lubricating off	60000	
T0017	Time unit setting for lubricating on	1000	
T0018	Time setting for spindle positioning delay check	2000	
T0019	Delay timer for spindle JOG	2000	
T0020	Delay timer for program restart	10	
T0021	Delay timer for spindle gear 1	10000	
T0022	Delay timer for spindle gear 2	10000	
T0023	Delay timer for spindle gear 3	10000	
T0024	Check time of spindle speed position mode conversion	10000	
T0025	Completion time of M29 execution (enabled without check signal)	4000	
T0026	Positioning completion time in spindle position mode(enabled without check signal)	1500	
T0027	Completion time of M28 execution(enabled without check signal)	2000	

## Appendix

T0028	Check time of spindle position speed mode conversion	10000	
T0029	The spindle brake time	1000	
T0030	Spindle gear 1: The delay time of gear change without the detection switch	3000	
T0031	Spindle gear 2: The delay time of gear change without the detection switch	3000	
T0032	Spindle gear 3: The delay time of gear change without the detection switch	3000	
T0033	Canceling the delay finish time in the spindle gear control without the detection unit	3000	
T0034	M5 finish delay time	500	
T0035	Disconnecting in the delay time after the spindle gear change finish	800	
T0036	The delay time after the index worktable releasing finish	500	
T0037	The delay time after the index worktable releasing finish	500	
T0038	The delay time after the index worktable releasing detection	5000	
T0039	The delay time after the index worktable clamping detection	5000	
T0040	The delay time after the index worktable releasing finish	4000	
T0041	The delay time after the index worktable clamping finish	4000	
T0100	Delay time 1 for turret tool magazine manually rotating	2000	
T0101	Delay time 2 for turret tool magazine manually rotating	2000	
T0102	Delay check time setting for turret tool magazine stopping	2000	
T0103	Delay check time setting for turret tool magazine rotating	3000	
T0104	Delay check time setting for tool magazine infeed	10000	
T0105	Delay check time setting for turret tool magazine retraction	10000	
T0106	Delay timer for turret tool magazine infeed completion	0	
T0107	Delay timer for turret tool magazine retraction completion	0	
T0108	Setting the delay detection time of the turret magazine rotation 2	3000	
T0109	Setting the time for compelling to stop when the magazine rotation abnormal occurs	5000	

Table 6 DATA parameter definition

Definition	Statement	Setting value by customer
D000	Spindle tool number display	Cannot set
D001	No.1 tool number	
D002	No.2 tool number	
⋮	⋮	
D098	No.98 tool number	
D099	No.99 tool number	
D100	Tool magazine capacity	
D241	T code tool number	Cannot set
D243	Current cutter head number	Cannot set
D245	Spindle tool number	

**Notes:**

1. D100 setting value must be less than 100, and must be the same that of CTR100, otherwise, there may be the abnormal.

For example: D100=16, the data table D001-D016 is valid.

D100=24, the data table D001-D024 is valid.

2. D000 only displays the spindle tool number, the spindle tool number cannot be modified at the D000 but at the D245.
3. D241 value cannot be modified.
4. D240~D247 is used by the system and cannot be defined by the user.
5. See **part three** for the details.

Table 7 CTR parameter definition

Address	Statement	Initial value	Setting value by customer
C100	Tool magazine capacity setting	16	

**Notes:**

1. CTR100 setting value must be less than 100 and must be the same as that of D100, otherwise there may be the abnormal.

For example: CTR100 =16, the total tool magazine number is 16.

CTR100=24, the total tool magazine number is 24.

**Table 8 M code definition**

M code	F signal	Function	Remark
M00	F031.7	Program pause	
M01	F030.4	Optional stop	
M02	F030.5	End of program	
M03	F030.0	Spindle CCW	
M04	F030.1	Spindle CW	
M05	F030.2	Spindle stop	
M06	F030.3	Automatic tool change	
M08	F031.0	Cooling ON	
M09	F031.1	Cooling OFF	
M10	F031.2	A axis clamp	Reversed
M11	F031.3	A axis release	Reversed
M16	F026.0	Spindle release instruction	Reversed
M17	F026.1	Spindle clamp instruction	Reversed
M18	F028.3	Cancel the spindle exact stop	
M19	F026.2	Spindle exact stop instruction	
M21	F026.3	Search tool instruction when retraction	
M22	F026.4	Search tool instruction when startup the current tool	
M23	F026.5	Tool magazine forward	
M24	F026.6	Tool magazine backward	
M26	F0.27.5	Start chip removal hydrovalve	
M27	F0.27.6	Close chip removal hydrovalve	
M28	F032.3	Rigid tapping instruction cancel	
M29	F032.2	Rigid tapping instruction	
M30	F028.0	End of program	
M32	F031.4	Lubricating ON	
M33	F031.5	Lubricating OFF	
M35	F028.1	Start helical chip removal conveyor	
M36	F028.2	Close helical chip removal conveyor	
M40	F***.*	X axis image	
M41	F***.*	Y axis image	
M42	F***.*	Z axis image	
M43	F***.*	Cancel image	
M44	F026.7	Start spindle blow	
M45	F027.0	Stop spindle blow	
M50	F027.1	Start automatic tool change	
M51	F027.2	End of automatic tool change	
M53	F027.3	Judge whether the tool is correct after the tool change is executed	Reversed
M55	F027.4	Judge whether the spindle has the tool	Reversed

**Note:**

1. "F\*\*\*.\*\*\*" and M instructions with "Reserved" in F signal table cannot be used.
2. M16, M17, M21, M22, M23 and M24 are valid when the tool change is being executed, and they cannot run separately.

### 三. Usage and maintenance of GSK 990MA CNC System matching with turret tool magazine

#### 1. Tool magazine installation and related PLC parameter setting

Operation aim: ensure the ladder fit to the allocation of tool magazine

##### A. requirements of the ladder matched with the turret tool magazine to the machine tool:

1. The machine tool has the spindle tool automatically clamping/releasing device which has the normally open in-position check switch.
2. The spindle has the positioning function and its positioning angle can be regulated.
3. The tool magazine capacity must be less than 100.
4. The tool magazine can execute CCW/CW.
5. The tool magazine has the count switch, forward in-position check switch, and retraction in-position check switch.
6. The tool magazine has zero return switch (optional).

##### B. Wire connection related to tool magazine

###### 1) Input .

Address	Statement	Remark
X002.4	External clamp/release control	Optional
X002.5	Release check	Normally open contact
X002.6	Clamp check	Normally open contact
X005.0	Spindle orientation in-position	Normally-closed contact
X005.1	Tool magazine forward in-position	Normally open or normally closed
X005.2	Tool magazine backward in-position	Normally open or normally closed
X005.3	Tool magazine CCW/CW in-position	Normally open or normal closed
X005.4	Tool magazine zero return in-position	Optional

###### 2) . Output:

Address	Statement	Remark
Y000.2	Tool release/clamp	
Y002.0	Spindle enabling	
Y002.1	Spindle orientation	
Y003.0	Tool magazine CCW	
Y003.1	Tool magazine CW	
Y003.2	Tool magazine forward	
Y003.3	Tool magazine backward	

### C. Tool magazine switch type and rotation in-position mode selection

Tool magazine no zero return switch	set K010.1 to 0
Tool magazine zero return switch	set K10.1 to 1
Tool magazine zero return switch is normally open	set K010.2 to 0
Tool magazine zero return switch is normally closed	set K010.2 to 1
Tool magazine infeed switch is normally open	set K010.3 to 0
Tool magazine infeed switch is normally closed	set K010.3 to 1
Tool magazine retraction switch is normal open	set K010.4 to 0
Tool magazine retraction switch is normal closed	set K010.4 to 1
Tool magazine count switch is normally open	set K010.5 to 0
Tool magazine count switch is normally closed	set K010.5 to 1

#### Note:

1. K010.2 setting is disabled when K010.1 is set to 0.

For example: the tool magazine has no zero return switch, all check switches are normal open and the rotation in-position is A mode, the parameter setting is as follows:

K010.1=0 K010.2=0 K010.3=0 K010.4=0 K010.5=0 K010.0=0

### D. tool magazine capacity setting

Input tool magazine capacity in DATA100 and CTR100

#### Notes:

1. The tool magazine capacity is defined that total tool magazine numbers in the tool magazine.
2. DATA100 and CTR100 setting values must be less than 100.

### E. Tool number setting:

Data table D001-D099 respectively corresponds to the tool magazine number 1-99,

and the setting values in the data table D001-D009 respectively corresponds to the tool number in the tool magazine number 1-99. D245 is the spindle tool number.

**Notes:**

1. In D001-D099, there is no the same tool number(except for 0), otherwise the system alarms when the tool change is executed.
2. The tool number setting range should meet the requirements set by the parameter 0206, otherwise the system alarm when the T instruction is executed.

**For example:** when the tool magazine capacity is 16:

When the tool magazine is set orderly to 1-16 in D1-D16, when T8M6 is executed, the tool change is executed after No. 8 tool magazine number will rotate to the tool change position.

When the tool magazine is set orderly to 10, 20, 30...160 in D1-D16, and T80M6 is executed, the tool change is executed after No. 8 tool magazine number will rotate to the tool change position

When D1 and D2 are set to 8, and T8M6 is executed, the system alarms.

**F. Time parameter setting related to the tool magazine:**

Address	Statement	Setting range	Initial value (ms)
T0004	Delay timer for spindle positioning	More than or less than 0	0
T0007	Delay timer for spindle tool releasing completion	More than or equal to 0	0
T0008	Delay timer for spindle tool clamping completion	More than or equal to 0	0
T0009	Time setting for spindle tool releasing check	More than releasing tool time	8000
T0010	Time setting for spindle tool clamping check	More than clamping tool time	8000
T0013	Time setting for spindle positioning check	More than positioning time	8000
T0018	Time setting for spindle positioning delay check	Related to the positioning width	2000
T0100	Delay time 1 for turret tool magazine manually rotating	More than rotating one tool selection	2000
T0101	Delay time 2 for turret tool magazine manually rotating	More than rotating one tool selection	2000
T0102	Delay check time for turret tool magazine stopping	More than rotating one tool selection	2000
T0103	Delay check time for tool magazine rotating	More than rotating one tool selection	3000
T0104	Delay check time for turret tool magazine infeed	More than infeed time	10000
T0105	Delay check time for turret tool magazine retracting	More than retraction time	10000
T0106	Delay timer for turret tool magazine infeed completion	More than or equal to 0	0



T0107	Delay timer for turret tool magazine retraction completion	More than or equal to 0	0
T0108	Setting the delay time 2 of the turret magazine rotation detection	Greater than the time of rotating one tool post	3000
T0109	Setting the time of compelling to stop the magazine rotation due to the abnormal	Greater than the time of rotating one tool post	5000

### Notes:

1. The above parameter (TMR) is related to the tool magazine type, the tool magazine speed and other performances. Please refer to the tool magazine performance to properly set the parameter.
2. When the above parameter (TMR) setting is not proper, the system alarms to cause that the tool change cannot be executed normally.

The ladder match with the tool magazine after the above A-F steps are operated, but the tool magazine cannot normally run, the tool magazine executes the CCW, CW, infeed and retraction in Manual mode to check whether each operation of tool magazine is normal through the following No. 2 setting and operation

## 2. Tool magazine manual and zero return operation

Operation aim: check whether the each operation of tool magazine is normal.

The detailed operation is as follows:

### A. the tool magazine is enabled.

1. Because the tool magazine is turret, the following bit parameters must be set.

Bit parameter 53.0=1 bit parameter 53.1=0 bit parameter 53.2=0 bit parameter 53.3=0

### B. Confirming the rotation direction of tool magazine

In Manual mode, press "MAG. CCW" and the tool magazine rotates according to the prescribed positive direction of machine tool, press "MAG. CW" and the tool magazine rotates according to the prescribed negative direction of machine tool, otherwise, the tool magazine count will be disorder to cause that the tool change is executed wrongly, which can be resolved by regulating the phase sequence of the motor.

### C. Tool magazine zero return:

Tool magazine zero return operation is divided into zero return switch and no zero return switch.

1. The tool magazine has the zero return switch: press "MAG. ZERO" in "ZERO

RETURN” mode, and the zero return is completed when the indicator is light(the light flashing indicates the tool magazine is executing the zero return.)




2. The tool magazine has no the zero return switch

- a. In Manual mode, press “CCW” or “CW” to make the No. 1 tool magazine number rotate to the tool change position.
- b. Set K0106 to 1 in MDI mode.
- c. press “MAG. ZERO” in zero return mode until its indicator slight.
- d. Set K010.6 to 0.

3. Spindle positioning angle and tool change coordinate position regulation

- A. About the spindle positioning angle regulation, please refer to the manual of the spindle drive unit.

Regulating steps of the spindle positioning angle (example):

1. Input the system debugging password. Continuously press  on MDI keypad to enter the password interface; in MDI mode, input the system debugging password: 888888, and then press  key, “PASSWORD CORRECT” is shown on the left bottom corner on the screen.
2. Press “PROGRAM CONTROL” key on MDI keypad; on **【PLCPAR】** interface, press **【KPAR】** softkey, the system enters the check and setting interface of the relay in keep type, press four direction keys on the keypad and the cursor can position whether K010.7 magazine enters the debugging mode. Press the , the data column reminds: “KPAR download is done!”, which means saving is successful, and the magazine debugging mode becomes valid (the advance and retraction of the magazine can be operated without any condition.). Then, the system reminds: 1253 caution to operate the magazine debugging mode, and pressing the reset key to cancel the alarm.
3. Prepare one knife handle to pull the pop-rivet.
4. In ZERO RETURN mode, Z axis zero return is executed.
5. In JOG mode, press “tool advance” key, the magazine is in the advance position.
6. Load the knife handle without the pop-rivet on the magazine and adjust it.
7. In MPG mode, Z axis can be moved upward and downward, which should be caution to operate. Then, the motor rotor or the spindle connecting the motor rotor is directly twisted for one more circle, and the spindle position is adjusted in JOG mode until the key on the spindle key enters the magazine slot. (About the spindle

positioning angle regulation, please refer to the manual of the spindle drive unit.)

### B. Regulating the tool change coordinate position

1. According to the above method, after the orientation regulation completes, the spindle orientation is executed in the JOG mode, and then the tool releasing movement is operated, the pop-rivet of the knife handle is installed (the magazine is still in the tool advance position.). In MPG mode, Z axis can be moved upward or downward to adjust the tool change coordinate position (Please make sure the tool is released.), and record the machine coordinate of Z axis (such as -120.000).
2. Firstly execute the tool clamping movement, and then press "tool retraction" key on the keypad, and the magazine returns to the tool retraction position. (Please make sure the tool is clamped.)
3. In MDI mode, set K010.7 to 0 as the above method.
4. The previously recorded Z axis machine coordinate is set in the data parameter 0052, and the parameter should be set correctly; otherwise, the accident may happen to cause the mechanical damage.

Through the operation of the three steps, the correct tool change can be executed, the operation of T code and M code is operated as below:

TxxM6; and Txx; M6; The executed effect is same.

T0M6; The tool on the spindle is returned to the magazine.

Warning: When the spindle tool number is 0, the spindle can't be with any tools; otherwise, the accident may happen during tool change and causes the mechanical damage!

### Appendix: The setting method of DAP01 positioning angle:

1. Power on the control device, default to display on LED:
2. Press key to enter the monitoring mode, the system displays:
3. There are 25 display states in Monitor mode, and select the state to look by pressing . Find (The motor coded disc is the positioning disc, and the spindle coded disc is taken as the positioning disc, can be found); press , the system displays or other numerical values.
4. To realize the spindle positioning function, the motor rotor should be rotated at least one circle to find the position. When the motor is OFF, directly twist the motor rotor or the spindle connecting the motor rotor for one more circle; it is suggested that the user directly twist for the convenient operation.
5. Adopt the above method to twist the motor for one more circle, and adjust gradually, the motor (spindle) rotor can position on the positioning point. After the operation, the absolute position of the rotor can be observed. or other numerical value is displayed on the

monitoring window. The value is recorded in the parameter PA58 and saved, that is the position. (For example: If 

E	213
---	-----

 is displayed on the monitor window, the numerical value "213" is input into the parameter PA58.)

4. The right method of handling the emergency stop, power off, resetting and alarm during the magazine running

1. In MDI mode, the tool change is commanded, or it is operated in Auto mode. When the magazine is in the advance limit position, the spindle comes downward the tool hold position. After clamping the tool, if the emergency stop, power off, alarm or resetting occurs when the magazine is going to retract, press "tool retraction" on the system to exit the magazine in JOG mode, then the tool takes off the current magazine chuck, the machine can be used. (Points for attention: If the spindle is released, Z axis is lifted and the magazine is exited, the magazine corresponding to the current spindle is loaded the tool, then hitting tool may happen during the tool change in the next time.)
2. In MDI mode, the tool change is commanded, or it is operated in Auto mode. When the magazine is in the advance limit position, the spindle has already released the tool. If the emergency stop, power off, alarm or resetting occurs, Z axis is going to be lifted, or the toolpot doesn't rotate after Z axis is lifted, press "tool release" button on the spindle in JOG mode, and then the tool is released, Z axis is lifted to reach the safe position in JOG or MPG mode, press "tool retraction" key on the system to exit the magazine, the operation can be executed normally. (Remark: The magazine chuck corresponding to the current spindle is loaded the tool, if the spindle is without the tool, the system should display "T0000".)
3. In MDI mode, the tool change is commanded, or it is operated in Auto mode. When the magazine is in the advance limit position, Z axis is lifted to the 1<sup>st</sup> reference position. When the tool is rotating, the current tool number may not comply with the actual one if the emergency stop, power off, alarm or resetting occurs, and the system alarms: "1218 The magazine stops and confirm the tool number again due to the abnormal." Or "1244 To guarantee the magazine safety, the tool change stops due to the abnormal", etc. If the operation continues, press "tool retraction" key on the system to exit the magazine in JOG mode, after the magazine returns zero, again and the spindle tool number and the magazine tool number can be reset, the operation can be executed normally. (About the details, please refer to: *Chapter III: The usage method and maintenance of GSK990MA CNC system with the turret magazine.*)

### 四. Macro program statement of GSK990MA CNC System matching with turret tool magazine

O91001;	(program name)
G65 H81 P50 Q#1003 R1;	(M.S.T and machine are locked, execute N50, end of program )
M50;	(start the tool change and check its conditions of tool change, if not, the system alarms.)
G69 G50 G15 G80;	(cancel the related modes)
G65 H81 P40 Q#1001 R1;	(spindle tool number=T tool number: not execute the tool change but N40, end of program)
G65 H81 P20 Q#1000 R1;	(spindle tool number=0: the spindle has no tool, execute N20 instead of the retraction tool instruction)
M19 G91 G30 Z0;	(spindle positioning, return to the coordinate position of tool change)
M21;	(retraction-> tool magazine rotating tool magazine forward spindle releasing )
N20 M19 G91 G28 Z0;	(return to machine origin point)
G65 H81 P30 Q#1002 R1;	(T code tool number=0: not execute the tool search but N30)
M22;	(tool search ->tool magazine rotating tool magazine forward spindle releasing tool)
G91 G30 Z0;	(return to coordinate point of tool change)
N30 M17;	(spindle clamping tool)
M24;	(tool magazine retraction)
N40 M51;	(end of tool change)
N50 M99;	(end of program)
%	

## Appendix 2

### Configuration file format of ladder

The signal in configuration file **LadChixx.txt** of ladder ("**xx**" corresponds with the running ladder file number set by system) stores the following information by fixed sequence:

#### 一. F signal and meaning for M00-M99 of M code

Format: MXX + space + Fyyy + space + Chinese notes + EOB symbol(enter)

Example: "M00 F0317 program dwell"

Thereinto: "xx" from up to down is 00\01\02\... till to 99, it totals 100. and its sequence cannot be altered.

"Space" generally includes one Space, which cannot be adulterated by any other characters.

"yyy" means a value to F signal by M code, i.e. **F0317** represents **F31.7**". it may set from 0260 to 0337 (i.e. F signal from 26.0 to 33.7). if it is set to "**-001**", it means no registration is allowed and it cannot be identified by system.

"Chinese remark" contains up to 32 characters; it may has 16 Chinese characters or equivalent Chinese characters + characters. The following is same as this.

"EOB symbol" means the end of the line and the characters following will not be identified. The following is same as this.

#### 二. "%" that occupies a line exclusively means the end of M code information storage.

#### 三. The codes and meaning of X signal X0.0-X6.7

Format: Xxxxx + space + Chinese remark + EOB symbol (enter)

Example: "**X0000** X axis positive stroke limit signal"

Thereinto: "**xxxx**" means the value of X signal, i.e. "**0000** for **0.0**", "**0067** for **6.7**". and it is from 0000 to 0067 from up to down(i.e. X signal from 0.0 to 6.7), its sequence can't be altered.

“Space” generally includes 5 spaces, which cannot be adulterated by any other characters. The following is same as this.

### 四. The codes and meaning of Y signal Y0.0---Y5.7

Format: Yxxxx + space + Chinese notes + EOB symbol (enter)

Example: “**Y0000**     Z axis holding”

Thereinto: “xxxx” means the value of Y signal, i.e. “0000 for 0.0”, “0057 for 5.7”. and it is from 0000 to 0057 from up to down(i.e. Y signal from 0.0 to 5.7), its sequence can’t be altered.

### 五. The codes and meaning of K signal Y6.0---Y63.7

Format: Kxxxx + space + Chinese remark + EOB symbol(enter)

Example: “**K0060**     If X axis stroke switch is 1”

Thereinto : “xxxx” means the value of K signal, i.e. “0060 for 6.0”, “0637 for 63.7”. and it is from 0060 to 0637 from up to down(i.e. K signal from 6.0 to 63.7), its sequence can’t be altered.

“Space” generally includes 5 spaces, which cannot be adulterated by any other characters.

### 六. The codes and meaning of A signal A0.0-A31.7

Format: Axxxx + space + Chinese notes + EOB symbol(enter)

Example: “**A0000**     air pressure detection abnormal”

Thereinto: “xxxx” means the value of A signal, i.e. “0000 for 0.0”, “0317 for 31.7”. and

it is from 0000 to 0317 from up to down(i.e. A signal from 0.0 to 31.7), its sequence can’t be altered.

“Space” generally includes 5 spaces, which cannot be adulterated by any other characters.

### 七. end//end sign

**Note:** Every line of above information should be written in a set form, refer to system embedded file LadChixx.txt for details. The English files

**CNCmakers**

[www.CNCmakers.com](http://www.CNCmakers.com)